LECTURE NOTES

ON

FINITE ELEMENT METHODS

III B. Tech II semester (JNTUH-R15)

Prof. V.V.S. H. Prasad, Professor

Mr. C. Labesh Kumar, Assistant Professor

DEPARTMENT OF MECHANICAL ENGINEERING

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

DUNDIGAL, HYDERABAD - 500 043

1 Introduction to Finite Element Method

The digital computer has exerted a profound impact on engineering education, research, and practice. A versatile numerical method in the hands of engineers is the Finite Element Method (FEM). A general-purpose program based on FEM implemented on a computer provides a universal tool for engineering analysis, design optimization, and simulation. This chapter sets the stage for the study of finite elements and solution procedures that are described in detail in the subsequent chapters. Applications to solid mechanics and structural mechanics problems are stressed in these chapters. More advanced applications of FEM are identified in the last chapter as topics for further study.

1.1 Engineering Analysis

1.1.1 Objectives of engineering analysis

During the design and development of a product (as an assemblage of parts), the analyst is quite often required to: (i) calculate the displacements at certain points; (ii) calculate the entire distribution or displacement field; (iii) determine the stress distribution and hence predict strength; (iv) determine the natural frequencies and associated modes of vibration; (v) determine the critical buckling loads and the associated mode shapes; (vi) predict and plot forced vibration response; (vii) predict and plot transient response; (viii) predict temperature distribution and hence thermal stress distribution; (ix) predict crack growth, residual strength and fatigue life; (x) predict velocity, pressure and temperature distribution in fluids; (xi) study fluid–structure interactions (hydro-elasticity, aero-elasticity, etc.); (xii) study nonlinear effects (geometric and material nonlinearities); (xiii) determine electric and magnetic fields, and many more!

1.1.2 Methods of engineering analysis

To achieve the above objectives, the analyst has at his disposal three distinct approaches: (i) analytical methods; (ii) experimental techniques and (iii) numerical methods.

Analytical methods [1.1] provide quick closed form solutions. But, they treat only simple geometries and idealized support and loading conditions. Using experimental techniques, scaled models or prototypes can be tested. This approach is costly both in terms of the model, instrumentation, test facilities and the actual test itself. Numerical methods require very few restrictive assumptions; it can treat complex geometries and realistic support and loading conditions. They are far more cost effective than experimental techniques. The current interest in the engineering community on the development and application of computational tools based on numerical methods is thereby justified. This in fact was the motivation to develop the most versatile numerical method, namely, the finite element method (FEM).

The goal of analysis is to verify a design prior to manufacture. While there are several methods of engineering analysis, the most comprehensive is the finite element analysis (FEA).

The finite element method is essentially dependent for its success on the skillful use of digital computers. The method is put in the hands of professional engineers in the form of general-purpose programs.

It is in general possible to use the FEM to provide accurate numerical solutions to almost any mathematical problem or mathematically modelled physical problem in diverse fields like solid mechanics, mechanics of composites, fluid dynamics, heat transfer, etc. Many continuum mechanics problems arise in engineering and these are usually posed by appropriate differential equations and boundary conditions to be imposed on unknown functions. All such problems can also be dealt with by FEM.

1.1.3 History of finite element method

Finite element technology has emerged as a new discipline combining continuum mechanics with approximation theory, numerical analysis, and computer science. It draws from recent advances in each of these disciplines and is nourished by them, but it also stands as a viable branch of engineering in its own right.

In the pre-computer era, the finite element concept was used in the analysis of naturally discrete systems such as trusses, frames, electrical circuits, etc. The modern version of FEM was first used in engineering practice by discretizing a continuum using simple sub-domains with multiple connecting points (nodes) by Turner et al., [1.2], Argyris [1.3], and Clough [1.4]. The name finite element was first coined by Clough [1.4] in the year 1960. Parallel developments were also reported in applied mathematics literature.

The development of modern finite element technology therefore spans a period of forty years and may be divided into four stages, with each stage spanning eight to ten years.

The first stage is characterized by development of simple two- and three-dimensional continuum elements, mostly for solid mechanics applications. The variational approach for deriving element equations was introduced and all the elements developed belonged to the stiffness (displacement) model.

In the second stage, the variational approach was expanded to include multi-field and generalized variational principles with relaxed inter-element displacement continuity requirements leading to the mixed models and hybrid models. Weighted-residual methods were introduced to extend the finite element method for the analysis of field problems. The mathematical foundations of FEM received a great deal of attention during this stage. Efficient numerical methods for the solution of algebraic equations and for the extraction of eigen values were developed. Sub-structuring (super elements) and modal synthesis techniques for solving very large problems were introduced. A number of commercial FEM systems were developed and released for public use. Also, FEM penetrated other areas of engineering analysis such as heat transfer, fluid dynamics, biomechanics, geomechanics, aeromechanics, fracture mechanics, mechanics of laminated composite materials and structures, electromagnetism, etc. Further applications to nonlinear and time-dependent (transient) response and coupled-field problems (soil–structure, fluid–structure, etc.) were also made.

The third stage involved the development and applications of special elements. Singular elements for computational fracture mechanics, boundary-layer elements for viscous fluid flow analysis, infinite elements for modelling unbounded domains, rigid links, and gap elements for contact problems are some of the examples. Other activities included the development of the boundary element method, coupling of FEM with continuum mechanics methods such as Rayleigh–Ritz and Bubnov–Galerkin methods, and establishing equivalence and similarities between various finite element methods.

The fourth stage is characterized by new application fields, development of efficient algorithms and computational strategies for new computing systems (e.g., vector multi-processor and massively-parallel processors), widespread availability of commercial FEA software on personal computers, workstations and supercomputers. Also increasing attention was focused on quality assessment and control of finite element solutions. Strategies were proposed for adaptive refinement of finite element approximation in order to achieve optimal solutions.

The success of FEM is mainly attributed to its generality, versatility, ability to model and analyze complex geometries and robustness. To date there are approximately five-hundred user-friendly, widely distributed, well documented general purpose FEA programs and over two hundred pre and post processor packages [1.5].

The literature on finite element technology is nearly over-whelming. The first textbook on the FEM was published in 1967. Since then over 387 textbooks and monographs and over 338 conference proceedings have been published on the subject [1.6].

The Finite Element Method by O.C. Zienkiewicz and R.L. Taylor, now in its fifth edition, (in volumes 1–3) is the pre-eminent reference work [1.7].

1.1.4 Advantages of finite element method

In initiating the prediction of displacements, stresses, vibration frequencies, buckling loads, etc., for a given product or its parts, the analyst must first derive the governing equations. A basic difficulty in this approach, quite apart from the solvability of the derived equations is the ability of these equations to represent the design conditions. Complexities in geometry, applied loads, support conditions and material properties enter into this condition. A basic promise of the finite element method is that a system of matrix equations governing the behaviour can be formed automatically and solved efficiently, irrespective of the complexities of practical design conditions.

The underlying mathematics of FEM is simple to understand and the procedure easy to use. However, successful application of FEM in practice depends on the availability of a general-purpose finite element analysis software implemented on a digital computer. Commercial FEM systems abound. A handbook on the topic [1.5] lists and describes over fifty programs by name along with their availability. For example, these programs can solve limitless variety of problems in solid mechanics and structural mechanics, whether linear or nonlinear, static or dynamic, elastic or plastic.

A more subtle attribute of FEM is its ability to deal with complex material models. For example, the heterogeneous, anisotropic, nonlinear, inelastic models of laminated composite materials and sandwich construction are handled without any significant expansion of the cost or complexity of the numerical simulation process. FEM brings a number of special advantages to coupled thermal–structural analysis. A consistent methodology of finite element heat transfer analysis is available [1.8] for the computation of temperature distribution in solids and structures. It is possible to use the same general-purpose FEA program to predict both temperature distribution due to thermal input and thermal stresses arising from these temperatures. Also, in cases where the material properties are a function of temperature, it is possible to assign material properties to each finite element consistent with the temperature level of that element. It is a revolution you must be familiar with – the marriage of personal computers and finite element analysis programs. Over thirty-five PC-based FEM systems are currently available in the market [1.9]. Of these, some stand out as market leaders based on their performance, popularity and advertising. What makes FEA on a PC so successful is its affordability. This will change the face of engineering design in general and structural and mechanical design in particular. Technically, the size of a finite element model that a PC can handle is limited only by the capacity of the hard disk. Best of all, a PC-based FEM system is an excellent training tool for teaching FEM.

1.1.5 Variational principles and finite element methods

The finite element method applied to the numerical solution of solid mechanics problems can be regarded as applications of the known variational principles. There are several advantages: (i) the method is thereby put on a sound theoretical foundation; (ii) the requirements for compatibility are clarified as these are quite explicit in the statement of the energy principles; and (iii) greater flexibility in the design of finite elements comes about because of the variety of alternatives at hand.

In addition to the minimum potential energy and minimum complementary energy principles, more general mixed variational principles due to Hellinger–Reissner and Hu–Washizu are available. Also, modified forms of these with relaxed inter-element displacement or stress continuity are also available. Table 1.1 is particularly useful in classifying the many different avenues for developing finite element methods. Among the finite element methods identified in Table 1.1, the displacement method is certainly the best understood and most widely used. The subsequent discussion is therefore confined to the displacement method. Finite element analysis of solids and structures based on the principle of minimum potential energy employs a piece-wise Rayleigh–Ritz procedure. Note that the assumed displacement functions must satisfy sufficient continuity conditions within the domain under consideration and the kinematic boundary conditions. There is, however, no requirement that the force boundary conditions be a priori satisfied.

Table 1.1 *Variational principles and finite element methods*

Variational principle	Finite element method
Principle of minimum potential energy/Principle of virtual work (PMPE)	Stiffness method/Displacement method
Principle of minimum complementary energy (PMCE)	Equilibrium method/Force method
Hellinger–Reissner mixed variational principle	Mixed method I
Modified PMPE	Hybrid displacement method
Modified PMCE	Hybrid stress method
Modified mixed variational principle	Mixed method II
Hu–Washizu principle	Displacement method

1.1.6 Basic steps in finite element analysis

The first step is the *discretization* of a given domain using finite elements. The domain can be a solid, a liquid, a gas, or their combinations. A library of finite elements of different types, shapes and orders is available for this purpose. Each element has a finite number of nodes and each node a finite number of degrees of freedom, which are the fundamental unknowns. The elements are inter-connected at their nodes only and the finite element mesh (see Fig. 1.1) is generated using a pre-processor. Commercial pre-processors have the capability of automated mesh generation and adaptive mesh refinement. The second step is to *approximate the field variable(s)* over each element domain in terms of their nodal values using *interpolation functions* (also called *shape functions*). *Derivation of element equations*, which are necessary and sufficient to determine the vector of nodal degrees of freedom for each element, is the objective of the third step. Variational and weighted residual approaches are widely used in this step. *Computation of element matrices and vectors* involves numerical integration over each element domain. The fourth step involves *assembly of element equations*. In the fifth step, the governing matrix equations are appropriately modified to enforce *boundary, support, symmetry* and *constraint* conditions. Special elements such as springs, rigid links, and gap elements are made use of for this purpose. *Solution of the governing matrix equations* is accomplished in the sixth step. This will provide the vector of nodal degrees of freedom for the assemblage as well as for the individual elements. The final step is called *post-processing* where the numerical results are printed, plotted, displayed, and animated graphically. Interactive computer graphics are used for this purpose.

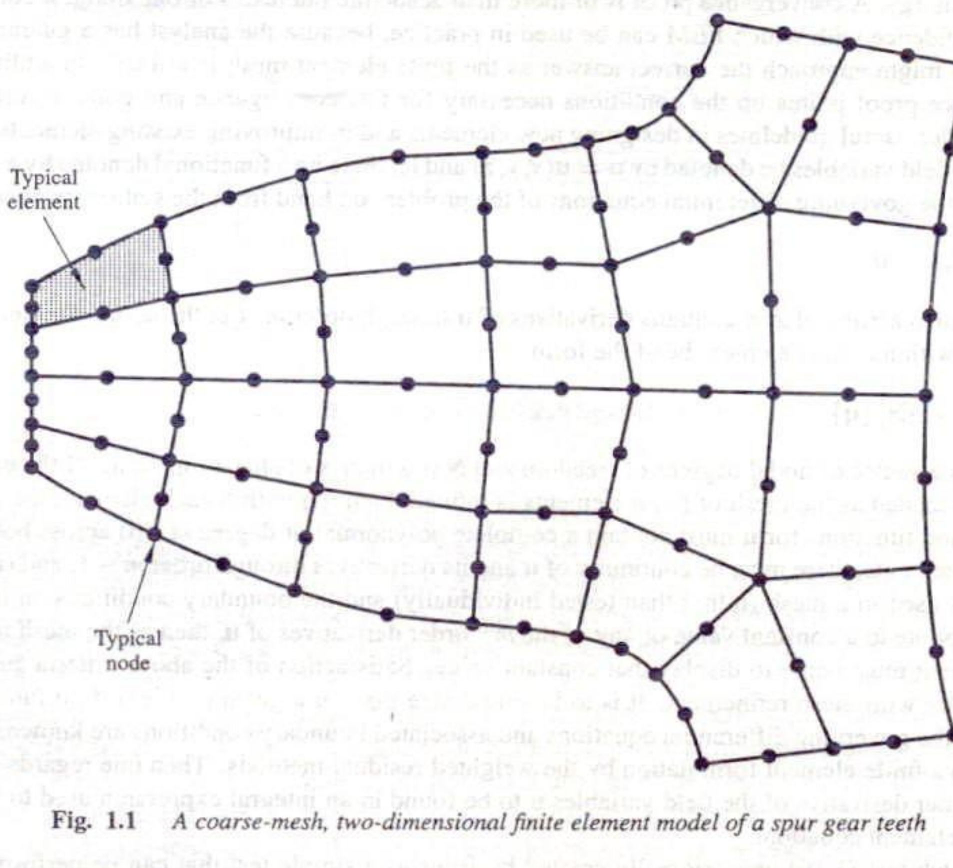


Fig. 1.1 *A coarse-mesh, two-dimensional finite element model of a spur gear teeth*

The above finite element procedure of artificially subdividing a given domain into convenient subdomains and assuming separate interpolation functions for each subdomain can be termed piece-wise Rayleigh–Ritz method.

Two approaches are available for refining a finite element model as the one shown in Fig. 1.1. The first approach provides more number of elements of the same type. This leads to the so-called h-convergence. In the second approach, the number of elements remains fixed, but the order of the interpolation functions used within each element is increased successively. This leads to p-convergence. In practice, a combination of h- and p-convergence is also used. Does an approximate numerical solution, obtained by the finite element method converge to the exact solution as the finite element mesh is uniformly refined? To ensure convergence, some criteria have to be satisfied. These are discussed in the next section.

1.1.7 Convergence criteria

If a particular problem in solid or structural mechanics is repeatedly analyzed, each time using a finer mesh of finite elements, we generate a sequence of approximate numerical solutions. How can we be assured that the sequence converges to the theoretically correct results? For conforming displacement model, a relatively simple proof of convergence can be given, based on the minimum property of the potential energy. A convergence proof is of more than academic interest. For one thing, it contributes to the confidence with which FEM can be used in practice, because the analyst has a guarantee that the results might approach the correct answer as the finite element mesh is refined. In addition, the convergence proof points up the conditions necessary for fast convergence and good accuracy, and thus provides useful guidelines in designing new elements and in improving existing elements.

Let the field variables be denoted by $\mathbf{u} = \mathbf{u}(x, y, z)$ and let there be a functional denoted by $\pi = \pi(\mathbf{u})$ that gives the governing differential equations of the problem on hand from the stationary condition

$$\partial\pi(\mathbf{u}) = 0.$$

Let us also assume that π contains derivatives of $\mathbf{u}$ through order m. Let the assumed interpolation functions within a finite element be of the form

$$\{\mathbf{u}\} = [\mathbf{N}]\{\mathbf{q}\}$$

where $\mathbf{q}$ is a vector of nodal degrees of freedom and $\mathbf{N}$ is a matrix of shape functions. If the exact $\mathbf{u}$ is to be approached as the mesh of finite elements is refined, then: (i) within each element, the assumed interpolation functions for $\mathbf{u}$ must contain a complete polynomial of degree m; (ii) across boundaries between elements, there must be continuity of $\mathbf{u}$ and its derivatives through order $m-1$; and (iii) if the element is used in a mesh (rather than tested individually) and the boundary conditions on the mesh are appropriate to a constant value of any of the m^{th} order derivatives of $\mathbf{u}$, then as the mesh is refined each element must come to display that constant value. Satisfaction of the above criteria guarantees convergence with mesh refinement. It is to be noted here that for a given problem if no functional π exists but the governing differential equations and associated boundary conditions are known, one can yet obtain a finite element formulation by the weighted residual methods. Then one regards m as the highest order derivative of the field variables $\mathbf{u}$ to be found in an integral expression used to generate the finite element equations.

The patch test [1.10] was originally created by Irons as a simple test that can be performed on a computer, so as to check the validity of a finite element formulation and its programmed implementation. If a particular element passes the patch test, we have the assurance that all convergence criteria noted above are met. Therefore, when this element is used to model any other structure or machine component, successive mesh refinement will produce a sequence of approximate solutions that converge to the exact solution. In other words, the patch test serves as a necessary and sufficient condition for convergence. An element that fails the patch test cannot be trusted.

1.1.8 Role of finite element analysis in computer-aided design

Figure 1.2 identifies the modules and their functions in a typical computer-aided design procedure.

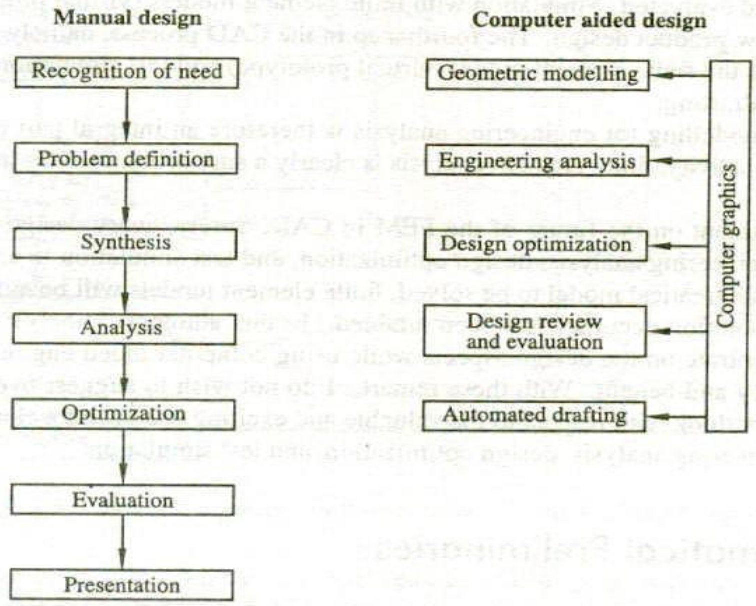


Fig. 1.2 *Computer-aided design: Modules*

We note that the first step is the geometric modelling of the product (an assemblage of parts). A large number of commercial CAD systems can be employed to perform this step. They are: Catia, UniGraphics, Pro/Engineer, I-Deas, Mechanical Desk Top, etc. In this step, for each part a mathematical model (the collection of all equations and data required to define the geometry) is generated and stored in the database. Given this information, the second step, the engineering analysis, may proceed.

In the second step, use of a general-purpose finite element analysis program (pre-processor, solver, post-processor) is now an accepted practice. A large number of commercial FEM systems are available for this purpose. They are NASTRAN, ANSYS, MARC, NISA, ALGOR, etc. This step generally is and should be performed by design engineers and not necessarily by engineering analysis specialists. Therefore, we recognize that the finite element method used must be robust, reliable and efficient. A basic pre-requisite to perform engineering analysis by FEM is a tractable mathematical model (the collection of all equations and data that can be used to predict the behaviour) of the product. FEM provides numerical solutions to the chosen mathematical model, which may be changed depending on the objectives of the analysis.

It is no longer sufficient to design a workable product which performs the desired functions. It has become essential to optimize the product design in order to maximize or minimize chosen variable(s) called objective function(s). This in fact is the third step in the CAD process. Keeping this in mind, commercial CAD and FEM systems have implemented FEM-based optimization methods as a module.

Simulation is the process of subjecting the part/product to various inputs, such as loads and environments, to determine how it behaves and thus predict the characteristics of the physical system. Though simulation may be carried out with scale models and prototypes, the cost and effort involved make it impossible to use these for product design since many feasible designs and operating conditions need to be considered and evaluated. Simulation with finite element models (virtual prototypes) is particularly valuable in new product design. The fourth step in the CAD process, namely design review and evaluation, relies on the finite element model (virtual prototype) and test simulation. The final step in CAD is automated drafting.

Finite element modelling for engineering analysis is therefore an integral part of CAD. Although an exciting field of activity, finite element analysis is clearly a supporting activity in the larger field of CAD.

Finally, we comment on the future of the FEM in CAD. Surely, every design engineer wants to use FEM-based engineering analysis, design optimization, and test simulation to enhance the product design. Given a mathematical model to be solved, finite element models will be automatically refined until the required solution accuracy has been attained. In this automated analysis environment, the engineer can concentrate on the design aspects while using computer aided engineering (CAE) tools with great efficiency and benefit. With these remarks I do not wish to suggest overconfidence but to express a realistic outlook with respect to the valuable and exciting use of finite elements and solution procedures in engineering analysis, design optimization, and test simulation.

1.2 Mathematical Preliminaries

1.2.1 Physical problems, mathematical models and finite element solutions

The finite element method is widely used to perform engineering analysis of physical systems. The derivation of an appropriate mathematical model of the physical problem is an important pre-requisite for engineering analysis to proceed. The finite element method provides numerical solutions to this mathematical model and therefore it is necessary to assess the accuracy of the numerical solution. If the accuracy criteria are not met, the finite element model has to be refined until sufficient accuracy is reached. Hence the development of an appropriate finite element model is crucial.

Once a mathematical model has been solved accurately by finite element analysis and the results interpreted with respect to the physical system, we may well decide to consider next a refined mathematical model in order to improve our understanding of the response of the physical system. Furthermore, a change in the physical problem statement itself may be necessary, and this in turn will also lead to additional mathematical models and their finite element solutions. This iterative solution process is shown in Fig. 1.3.

In summary, we should keep in mind that the crucial step in finite element analysis of physical systems is always choosing an appropriate mathematical model. Furthermore, the chosen mathematical model must be reliable and effective. In the process of analysis, the analyst has to judge not only the accuracy of the finite element solution but also its validity. Choosing the mathematical model,

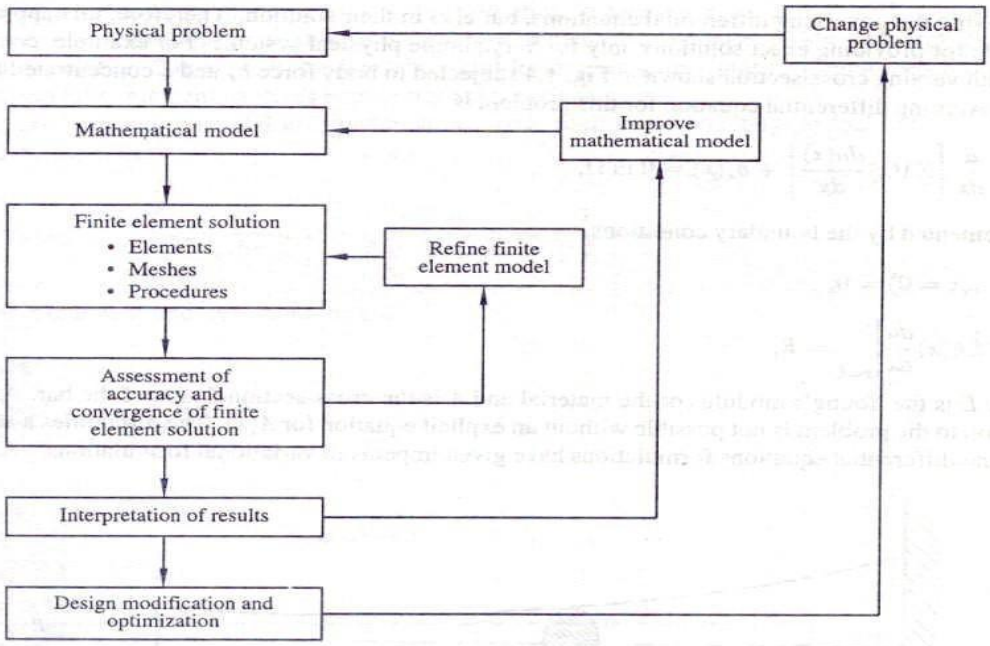


Fig. 1.3 *Physical problems, mathematical models, finite element solutions*

solving this model by appropriate finite element procedures, and judging the results are the fundamental ingredients of an engineering analysis.

Some classical techniques used for the formulation and solution of mathematical models of engineering systems is well documented (see Bathe [1.11]). Two categories of mathematical models are considered: lumped-parameter models and continuum-mechanics-based models. We also refer to these as discrete-systems and continuous-systems mathematical models. Discrete-system mathematical models lead to steady state problems, propagation problems and eigen value problems. Continuous-system mathematical models lead to differential equation formulations, variational formulations and weighted residual methods.

1.2.2 Differential equations formulations

Continuum-mechanics-based mathematical modelling of systems lead us to differential equations. The governing differential equations must be satisfied throughout the domain of the physical system, and before their solution can be attempted, they must be supplemented by boundary conditions and also by initial conditions. In initiating engineering analysis of a part/product, the analyst must first derive the governing differential equations. A basic difficulty in the approach, quite apart from the solvability of the derived equations, is the ability of these equations to represent the complexities in geometry, applied loads, support conditions and material properties. In summary, we face difficulties not only in deriving the governing differential equations, but also in their solution. Therefore, this approach is suitable for providing exact solutions only for very simple physical systems. For example, consider a bar with varying cross-section shown in Fig. 1.4 subjected to body force b_x and a concentrated load R. The governing differential equation for this problem is

$$\frac{d}{dx}\left[EA(x)\frac{du(x)}{dx}\right] + b_x(x) = 0 \text{ in } \Omega, \tag{1.1}$$

supplemented by the boundary conditions,

$$u(x=0) = 0, \tag{1.2}$$

$$EA(x)\left.\frac{du}{dx}\right|_{x=L} = R, \tag{1.3}$$

where E is the Young's modulus of the material and A is the cross-sectional area of the bar. An exact solution to the problem is not possible without an explicit equation for $A(x)$. The difficulties associated with the differential equations formulations have given impetus to variational formulations.

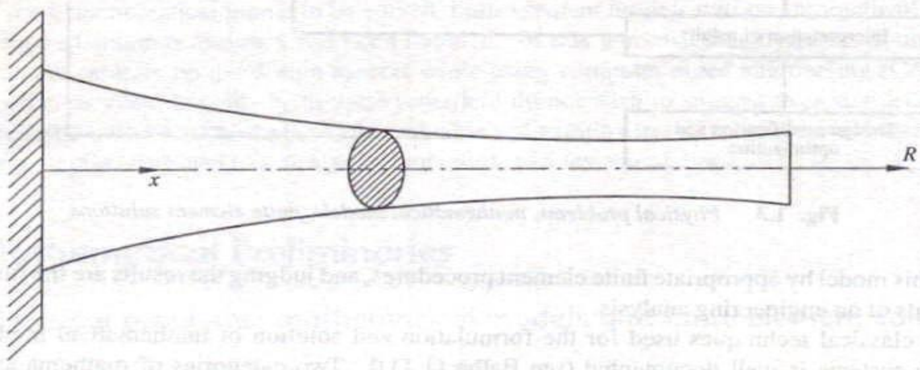


Fig. 1.4 *A bar subjected to axial loads*

1.2.3 Variational formulations

Continuum-mechanics-based mathematical modelling of solids and structures lead us to variational formulations. The essence of the approach is to calculate the total potential π of the system and to invoke the stationarity of π, i.e., $\partial\pi = 0$, with respect to the state variables. The variational formulations are effective for the solution of solid mechanics and structural mechanics problems by the Rayleigh–Ritz method. Indeed, the FEM for solid/structural mechanics problems can be regarded as a piecewise Rayleigh–Ritz method!

The total potential π is also called the functional of the problem. An important question then arises: How can we establish an appropriate functional corresponding to a physical problem?

For solid mechanics and structural mechanic problems, a number of functionals are applicable. For instance, we can employ the potential energy functional π_p, the complimentary energy functional π_c, the Hellinger–Reissner functional π_{HR}, the Hu–Washizu functional π_{HW}, and a number of modified forms of these with relaxed requirements on inter-element displacement or stress continuity which are also available. Table 1.1 is particularly useful in identifying the many different possibilities for developing finite element methods based on these functionals.

For example, the potential energy functional governing static buckling analysis of the problem shown in Fig. 1.5 is

$$\pi_p(w) = \frac{1}{2}\int_0^L EI\left(\frac{d^2w}{dx^2}\right)dx - \frac{P}{2}\int_0^L \left(\frac{dw}{dx}\right)^2 dx + \frac{1}{2}kw_L^2, \tag{1.4}$$

and the essential boundary conditions are

$$w(x=0) = 0, \tag{1.5}$$

$$\left.\frac{dw}{dx}\right|_{x=0} = 0. \tag{1.6}$$

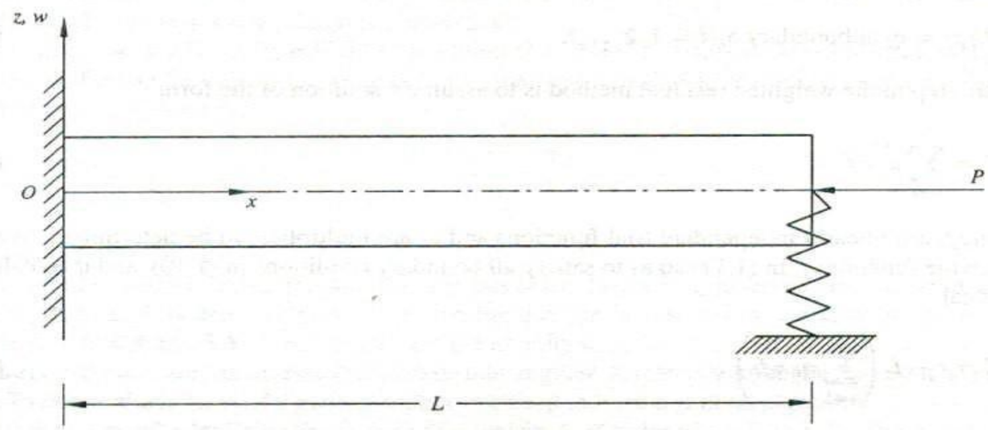


Fig. 1.5 *A column subjected to compressive load*

The basic step in the Rayleigh–Ritz method is to assume a solution of the form

$$w(x) = \sum_{i=1}^{n} a_i * f_i(x), \tag{1.7}$$

where $f_i(x)$ are linearly independent trial functions and a_i are multipliers to be determined. We have to assume the trial functions $f_i(x)$ such that the essential boundary conditions are a priori satisfied. The parameter a_i is determined from the equations

$$\frac{\partial \pi_p}{\partial a_i} = 0, i = 1, 2, 3, \ldots, n. \tag{1.8}$$

There are some classes of problems for which variational formulations are not available. This has given impetus to the development of weighted residual methods.

1.2.4 Weighted residual methods

In a previous section we have discussed the differential equations formulation of physical problems. For thermal analysis, fluid flow analysis, electromagnetic fields, etc., the governing differential equations, appropriate boundary and initial conditions are well known. However, closed form solutions are possible for simple systems only. For more complex systems, weighted residual methods must be employed. Indeed, the finite element method for field problems can be regarded as an extension of these.

Consider the analysis of a steady state field problem using its differential equations formulation,

$$L[\varphi] = r \text{ in domain } \Omega, \tag{1.9}$$

in which L is a linear differential operator, φ is the state variable to be calculated, and r is the forcing function. The solution to the problem must also satisfy the boundary conditions

$$B_i[\varphi] = q_i \text{ at boundary } S_i (i = 1, 2, \ldots). \tag{1.10}$$

The basic step in the weighted residual method is to assume a solution of the form

$$\bar{\varphi} = \sum_{i=1}^{n} a_i * f_i, \tag{1.11}$$

where the f_i are linearly independent trial functions and a_i are multipliers to be determined. We have to choose the functions f_i in (1.11) so as to satisfy all boundary conditions in (1.10), and then calculate the residual

$$R = r - L\left(\sum_{i=1}^{n} a_i * f_i\right). \tag{1.12}$$

For the exact solution the residual is of course zero. A good approximation to the exact solution would imply that R is small at all points of the solution domain Ω. The various weighted residual methods differ in the criteria that they employ to calculate a_i such that R is small. However, in all the methods we determine the a_i so as to make a weighted average of R vanish. In the Galerkin method, the parameters a_i are determined from the equation

$$\int_{\Omega} f_i * R d\Omega = 0 (i = 1, 2, 3, \ldots, n), \tag{1.13}$$

where Ω is the solution domain.

An important step in using a weighted residual method is the solution of the simultaneous equations for the parameters a_i. We note that since L is a linear operator, a linear set of equations in the parameters a_i is generated. In addition, the coefficient matrix is symmetric (and also positive definite) if L is a symmetric and also positive definite operator. The fundamental difficulty in using the weighted residual method in practice is because the trial functions must be $2m$ times differentiable and must satisfy all essential and natural boundary conditions, where $2m$ is the order of the highest derivative present in the problem governing the differential equations. Therefore, the weighted residual method is used in the context of the FEM in a different form, namely, in a form that allows the use of trial functions which are to be m times differentiable only and do not need to satisfy a priori the natural boundary conditions. The procedure would be to weigh the governing differential equations(s) in the domain with suitable weight function(s); integrate the resulting equation(s) with a transformation using integration by parts (or more generally using the divergence theorem); and substituting the natural boundary conditions. This leads us to a weak statement.

1.2.5 Matrix algebra

The use of vectors and matrices is of fundamental importance in engineering analysis by finite element method. The objective of this section is to present the fundamentals, with emphasis on those aspects that are important in finite element analysis.

Matrices are an ordered array of numbers that are subjected to specific rules of addition, multiplication, and inversion. Whenever the elements of a matrix obey a certain law, we can consider the matrix to be of special form symmetric, diagonal, banded, etc.

Two matrices $\mathbf{A}$ and $\mathbf{B}$ can be multiplied to obtain $\mathbf{C} = \mathbf{AB}$ if and only if the number of columns in $\mathbf{A}$ is equal to the number of rows in $\mathbf{B}$. If $\mathbf{A}$ is of order $p \times m$ and $\mathbf{B}$ is of order $m \times q$, then for each element of matrix $\mathbf{C}$, we have,

$$\mathbf{C}_{ij} = \sum_{k=1}^{m} a_{ik} b_{kj}, \tag{1.14}$$

and $\mathbf{C}$ is of the order $p \times q$.

With regard to matrix division, it strictly does not exist. Instead, an inverse matrix is defined. The inverse of a matrix $\mathbf{A}$ is denoted by $\mathbf{A}^{-1}$. Assuming that the inverse exists, the elements of $\mathbf{A}^{-1}$ are such that $\mathbf{A}^{-1}\mathbf{A} = \mathbf{I}$ and $\mathbf{A}\mathbf{A}^{-1} = \mathbf{I}$, where $\mathbf{I}$ is the identity matrix.

A matrix that possesses an inverse is said to be nonsingular. A matrix without an inverse is a singular matrix. To obtain the inverse of a general matrix, we need to have a general algorithm.

A practical way of calculating the inverse of a matrix $\mathbf{A}$ of order $n \times n$ is to solve the n systems of equation

$$\mathbf{AX} = \mathbf{I}, \tag{1.15}$$

where $\mathbf{I}$ is the identity matrix of order n and we have $\mathbf{X} = \mathbf{A}^{-1}$. For the solution of equations in (1.15), one can use the well known Gaussian elimination algorithm.

The trace and determinant of a matrix are defined only if the matrix is square. The trace of the matrix $\mathbf{A}$ is denoted as $\text{tr}(A)$ and equal to

$$\text{tr} = \sum_{i=1}^{n} a_{ii},$$

where n is the order of $\mathbf{A}$.

The determinant of an $n \times n$ matrix $\mathbf{A}$ is denoted as $\det A$ and is given by the recurrence relation

$$\det A = \sum_{j=1}^{n} (-1)^{1+j} a_{1j} \det A_{1j},$$

where A_{ij} is the $(n-1)(n-1)$ matrix obtained by eliminating the 1st row and jth column from the matrix $\mathbf{A}$.

A set of simultaneous linear algebraic equations may be symbolized as

$$\mathbf{AX} = \mathbf{b}. \tag{1.16}$$

Solution for $\mathbf{X}$ may be symbolized as

$$\mathbf{X} = \mathbf{A}^{-1}\mathbf{b}.$$

A square matrix is called singular if its determinant is zero. If $\mathbf{A}$ in (1.16) is singular, there is no unique solution vector $\mathbf{X}$, and standard equation solvers will fail. Let $\mathbf{A}$ be an $n \times n$ matrix and $\mathbf{X}$ an $n \times l$ column vector. Also let $\mathbf{X} \neq 0$, which means that at least one coefficient $\mathbf{X}_i$ is nonzero. Then for all $\mathbf{X}$,

if $\mathbf{X}^T\mathbf{AX} > 0$, $\mathbf{A}$ is called positive definite.
if $\mathbf{X}^T\mathbf{AX} \geq 0$, $\mathbf{A}$ is called positive semi-definite.

1.3 Numerical Methods in Finite Element Analysis

1.3.1 Interpolation functions

In the finite element method, there is a need to approximate the field variables over each element domain in terms of their nodal values using interpolation functions (also called shape functions). However, to ensure monotonic convergence of finite element solutions, the interpolation functions should a priori satisfy the so-called convergence criteria. The two requirements for monotonic convergence are that the elements (or the mesh) must be compatible and complete. For one-dimensional elements, C^0 continuous Lagrange and C^1 continuous Hermite polynomial functions, well known in the mathematics literature, are used as interpolation functions. For two- and three-dimensional continuum elements and for structural elements (beam, plate, shell), procedures to derive appropriate interpolation functions are outlined in the subsequent chapters.

1.3.2 Numerical integration techniques

An important aspect of finite element analysis is the use of numerical integration techniques to compute element matrices and vectors. The required integrals in the one-, two- and three-dimensional cases respectively, can be written as

$$I = \int F(\xi)d\xi;$$

$$I = \int F(\xi, \eta) d\xi d\eta; \text{ and}$$

$$I = \int F(\xi, \eta, \zeta) d\xi d\eta d\zeta.$$

These integrals are evaluated numerically using

$$I = \int F(\xi)\, d\xi = \sum_{i=1}^{n} W_i F(\xi_i);$$

$$I = \int F(\xi, \eta)\, d\xi d\eta = \sum_{i=1}^{n} \sum_{j=1}^{n} W_i W_j F(\xi_i, \eta_j);$$

$$I = \int F(\xi, \eta, \zeta) d\xi d\eta d\zeta = \sum_{i=1}^{n} \sum_{j=1}^{n} \sum_{k=1}^{n} W_i W_j W_k F(\xi_i, \eta_j, \zeta_k),$$

where the W_i, W_j, and W_k are the weighting factors, and (ξ_i, η_j, ζ_k) denote sampling point locations and n denotes number of sampling points.

In finite element analysis we integrate matrices, which means that each element of the matrix considered is integrated individually. Hence for the derivation of numerical integration formulas, we need to consider a typical element of a matrix, which we denote as F. A very important numerical integration technique in which both the positions of the sampling points and the weights have been optimized is the Gauss quadrature formula. For the one-dimensional case, we have

$$I = \int F(\xi) d\xi = \sum_{i=1}^{n} W_i F(\xi_i).$$

We require n sampling points (also called Gauss points) to integrate exactly a polynomial of order at most $(2n - 1)$. Polynomials of orders less than $(2n - 1)$ are also integrated exactly.

A great deal of research has been done on the development of suitable numerical integration formulas for quadrilateral and triangular domains in two-dimensions, hexahedral and tetrahedral domains in three-dimensions.

1.3.3 Static analysis—solution of equilibrium equations

The computational efficiency of finite element analysis depends on the numerical methods used for the solution of the governing matrix equations. In this section, we are concerned with the solution of the simultaneous equations that arise in static analysis of solids and structures. Specifically, we discuss the solution of the equations that arise in linear static analysis

$$[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}, \tag{1.17}$$

where $\mathbf{K}$ is the stiffness matrix, $\mathbf{Q}$ the vector of nodal degrees of freedom, and $\mathbf{F}$ the vector of nodal forces.

There are two different approaches for the solution of the equations in (1.17) – direct solution techniques and iterative solution techniques.

The most effective direct solution techniques currently used are basically applications of Gauss elimination algorithm. The basic algorithm can be applied to any set of simultaneous linear equations. Its effectiveness in FEA depends on the specific properties of the assembled matrix: symmetry, bandedness, sparsity and positive definiteness.

The mathematical operations of Gauss elimination reduce the matrix **K** to upper triangular form, i.e., a form in which all elements below the leading diagonal are zero. Starting with the last equation, it is then possible to calculate by back substution all unknowns in the order $Q_n, Q_{n-1}, \ldots, Q_1$. A very important aspect of the computer implementation of the Gauss elimination procedure is referred to as the active column solution or the skyline reduction method. The use of bandwidth minimization procedures can be very important in practice because the mean half-bandwidth of a stiffness matrix may initially be rather large as a result of the element and nodal point generation schemes used. Furthermore, the active columns of the matrix **K** are stored in a one-dimensional array. Cholesky factorization, static condensation, sub structuring, and frontal solver are some other schemes that are in principle, applications of the basic Gauss elimination procedure.

It is informative to note that during the initial developments of the finite element method, iterative solution algorithms have been employed. A basic disadvantage of an iterative solution is that the time of solution can be estimated only very approximately because the number of iterations required for convergence depends on the condition number of the stiffness matrix **K**. It is primarily for this reason that the use of iterative solvers in finite element analysis was largely abandoned, while the direct solvers have been refined and rendered extremely effective. The Gauss-Seidel iterative procedure continues to find use. However, the conjugate gradient method is particularly attractive.

The finite element equations to be solved in nonlinear analysis of solids and structures are, at time $t + \Delta t$,

$$^{t+\Delta t}\mathbf{R} - {}^{t+\Delta t}\mathbf{F} = 0, \tag{1.18}$$

where the vector $^{t+\Delta t}\mathbf{F}$ stores the externally applied nodal loads and $^{t+\Delta t}\mathbf{R}$ is the vector of nodal forces that are equivalent to the element stresses. Both vectors in (1.18) are evaluated using the principle of virtual displacements.

Since the nodal point forces $^{t+\Delta t}\mathbf{R}$ depend nonlinearly on the nodal point displacements, it is necessary to iterate in the solution of (1.18). The most frequently used iteration scheme for the solution of nonlinear finite element equations are the Newton-Raphson and other closely related techniques. An important requirement of nonlinear finite element analysis is frequently the calculation of the collapse load of a structure. A load-displacement constraint method proposed by E.Riks can be used for this purpose.

1.3.4 Vibration analysis—solution of eigen problems

The finite element equations to be solved in vibration analysis of solids and structures are

$$\mathbf{K}\phi = \lambda \mathbf{M}\phi, \tag{1.19}$$

where **K** and **M** are, respectively, the stiffness matrix and mass matrix of the finite element model. The eigen values λ_i and the eigen vectors ϕ_r are the natural frequencies (radians/s) squared (ω_i^2) and the corresponding mode shape vectors, respectively. We concentrate in particular on the calculation of the smallest eigen values $\lambda_1, \lambda_2, \ldots, \lambda_p$ and corresponding eigen vectors $\phi_1, \phi_2, \ldots, \phi_p$.

The solution methods considered here can be subdivided into four groups: the vector iteration methods; the transformation methods; polynomial iteration techniques; and sturm sequence methods. A number of solution algorithms have been developed within each of these four groups. In addition, the Lanc'zos method and the subspace iteration methods are also available.

1.3.5 Dynamic response analysis—solution of equations of motion

The finite element equations for dynamic response analysis are

$$\mathbf{M\ddot{Q}(t) + C\dot{Q}(t) + KQ(t) = F(t)}, \tag{1.20}$$

where **M**, **C**, and **K** are the mass, damping and stiffness matrices; **F** is the vector of externally applied, time-dependent, nodal point loads, $\mathbf{Q}$, $\mathbf{\dot{Q}}$, and $\mathbf{\ddot{Q}}$ are the displacement, nodal point velocity and nodal point acceleration vectors of the finite element model, all of them being time-dependent.

Mathematically, (1.20) represents a system of linear differential equations of second order with constant coefficients. However, the procedures proposed for the solution of general systems of differential equations can become very expensive if the order of the matrices is large, unless specific advantage is taken of the special characteristics of the coefficient matrices **K**, **C** and **M**. In practical finite element analysis, the procedures considered are divided into two groups: direct integration and mode superposition. The central different method, the Houbolt method, the Wilson θ-method, the new-mark method, belong to the direct integration methods group.

The dynamic response analysis by mode superposition requires, first the solution of the eigen values and eigen vectors of the problem, then the solution of the decoupled equilibrium equations, and finally the superposition of the response in each eigen vector. In practice, the eigen vectors are the free vibration mode shapes of the finite element model.

The choice between mode superposition analysis and direct integration methods is merely one of numerical effectiveness.

Nonlinear dynamic response analysis of a finite element model is in essence, performed using the incremental formulations, the iterative solution methods, and the time integration algorithms (explicit integration and implicit integration). The application of mode superposition in nonlinear dynamic response analysis can be effective if only a relatively few modes shapes need to be considered.

1.3.6 Linear buckling analysis—solution methods

The matrix equation for linear buckling analysis of structures by the finite element method is

$$[\mathbf{K} + \lambda\mathbf{K}_\sigma]\{\mathbf{dQ}\} = \{\mathbf{0}\}, \tag{1.21}$$

where the initial stress stiffness matrix $\mathbf{K}_\sigma$ is calculated from an arbitrarily chosen level of membrane stress state, and λ is the factor by which this level must be increased or decreased in order to produce buckling. At the critical (buckling) load, there is a bifurcation in a load versus displacement plot. Two infinitesimally close equilibrium states are possible — the unbuckled state and the buckled state, without any change in the applied loads F. Nodal displacement increments $\{\mathbf{dQ}\}$ are departures from the configuration **Q** that exist just before buckling. The right hand side of (1.21) is the corresponding change in applied nodal point loads and is therefore a null vector. Equation (1.21) is an eigen problem.

The computed value of λ may be positive or negative, depending on the state of membrane stress used to construct $\mathbf{K}_\sigma$. Membrane stresses may be known at the outset, or they may have to be computed. Linear buckling analysis uses $\mathbf{K}$ and $\mathbf{K}_\sigma$ based on the original, undeformed geometry of the structure and often overestimates the actual buckling load. Most practical buckling problems are nonlinear, and buckling analysis should be based on the tangent stiffness that prevails at the instant of buckling. These considerations are automatically incorporated in the nonlinear finite element analysis procedures.

The reader is advised to refer Bathe [1.11] for algorithmic details. However, computer implementation of these numerical methods and its integration into a commercial FEM system that can be executed on any computer, personal to supercomputer, is a challenge and involves many human years' effort.

1.4 References

[1.1] Wang, C.T. *Applied Elasticity.* McGraw Hill, New York, 1953.

[1.2] Turner, M.J., Clough, R.W., Martin, H.C., and Topp, L.J. Stiffness and defection analysis of complex structures. *Journal of Aeronautical Sciences*, 23:805–823, 1956.

[1.3] Argyris, J.J. *Energy Theorems and Structural Analysis*. Butterworth, London, 1960.

[1.4] Clough, R.W. The finite element method in plane stress analysis. In *Proceedings of the Second ASC Conference on Electronic Computation*, Pittsburgh, PA, Sept. 1960.

[1.5] Brebbia, C.A., ed., *Finite Element Systems*. Springer Verlag, Berlin, 1985.

[1.6] Noor, A.K. Bibliography of books and monographs on finite element technology. *Applied Mechanics Reviews*, 4(6), June 1991.

[1.7] Zienkiewicz, O.C. and Taylor, R.L. *The Finite Element Method*, volume 1–3. Arnold Publishers, 5th ed., 2000.

[1.8] Baker, A.J. and Pepper, O.W. *Finite Elements 1–2–3*. McGraw Hill, New York, 1991.

[1.9] McLean, D.M. An overview of PC-based finite element programs in the USA. *Finite Element News*, June 1988.

[1.10] Taylor, R.L., Simo, J.C., Zienkiewicz, O.C., and Chan, C.E. The patch tests—condition for assessing FEM convergence. *International Journal For Numerical Methods in Engineering*, 22:39–62, 1986.

[1.11] Bathe, K.J. *Finite Element Procedures*. Prentice Hall of India, New Delhi, 1996.

1.5 Computational Problems

No specific computational problems are suggested in this chapter. However, students may wish to get acquainted with the FEA software chosen for the course by using its pre-processor to create simple geometries and discretize them using finite elements.

Unit-II
One Dimensional Elements

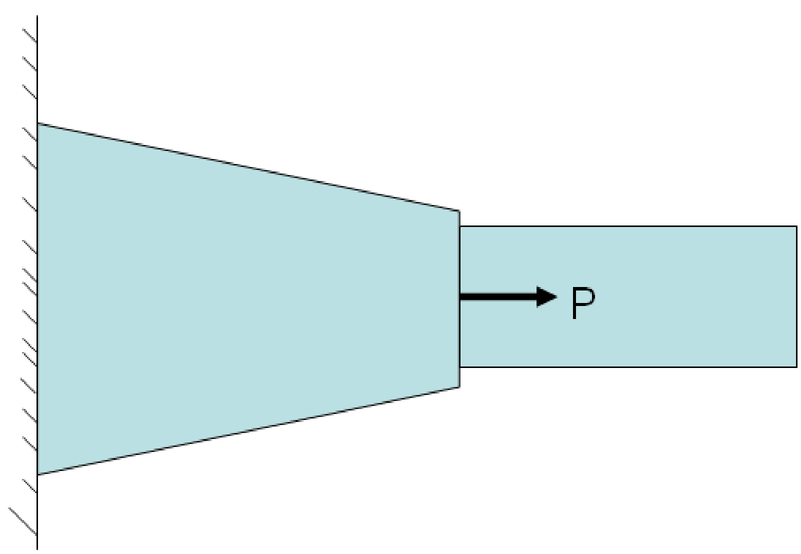
In the finite element method elements are grouped as 1D, 2D and 3D elements. Beams and plates are grouped as structural elements. One dimensional elements are the line segments which are used to model bars and truss. Higher order elements like linear, quadratic and cubic are also available. These elements are used when one of the dimension is very large compared to other two. 2D and 3D elements will be discussed in later chapters.

Seven basic steps in Finite Element Method

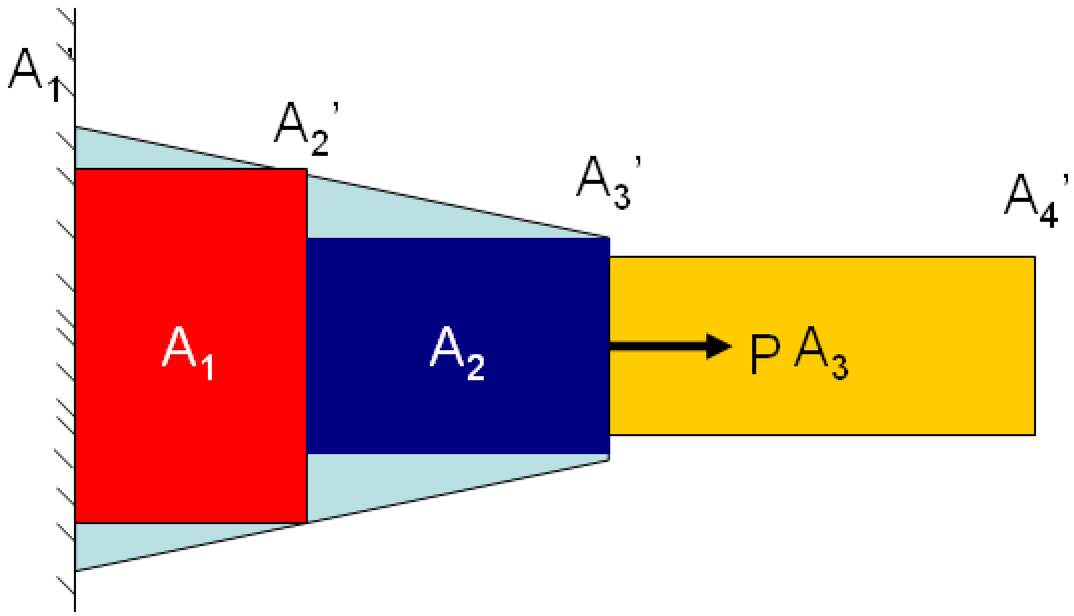
These seven steps include

- Modeling
- Discretization
- Stiffness Matrix
- Assembly
- Application of BC's
- Solution
- Results

Let's consider a bar subjected to the forces as shown

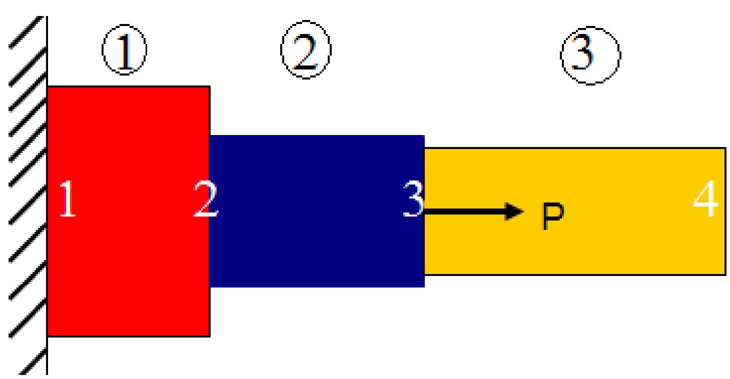


First step is the modeling lets us model it as a stepped shaft consisting of discrete number of elements each having a uniform cross section. Say using three finite elements as shown. Average c/s area within each region is evaluated and used to define elemental area with uniform cross-section.

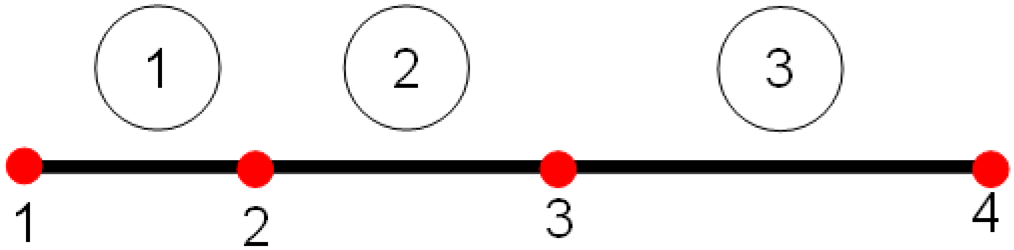


$A_1 = A_1' + A_2' / 2$ similarly A_2 and A_3 are evaluated

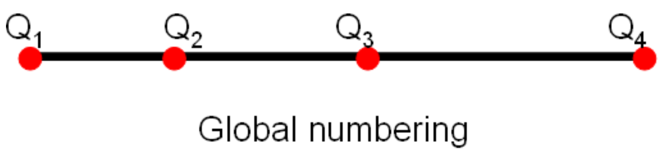
Second step is the Discretization that includes both node and element numbering, in this model every element connects two nodes, so to distinguish between node numbering and element numbering elements numbers are encircled as shown.

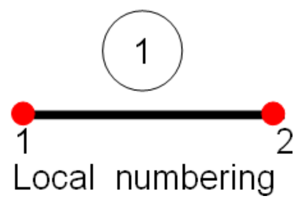


Above system can also be represented as a line segment as shown below.



Here in 1D every node is allowed to move only in one direction, hence each node as one degree of freedom. In the present case the model as four nodes it means four dof. Let Q1, Q2, Q3 and Q4 be the nodal displacements at node 1 to node 4 respectively, similarly F1, F2, F3, F4 be the nodal force vector from node 1 to node 4 as shown. When these parameters are represented for a entire structure use capitals which is called global numbering and for representing individual elements use small letters that is called local numbering as shown.





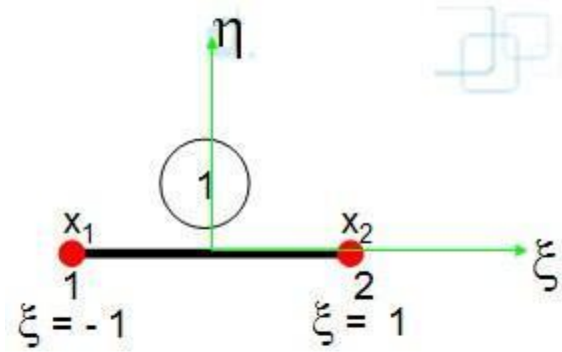
This local and global numbering correspondence is established using element connectivity element as shown

Elements e	Nodes 1 (Local)	Nodes 2 (Local)
1	1	2
2	2	3
3	3	4

Local ← (node numbers 1, 2 in header); Global } (node entries in table body)

Element Connectivity table

Now let's consider a single element in a natural coordinate system that varies in ξ and η, x_1 be the x coordinate of node 1 and x_2 be the x coordinate of node 2 as shown below.



Let us assume a polynomial

$$X = a_o + a_1 \xi$$

Now

$$@\ x = x_1 \qquad \xi = -1$$

$$@\ x = x_2 \qquad \xi = 1$$

After applying these conditions and solving for constants we have

$$x_1 = a_0 - a_1$$

$$x_2 = a_0 + a_1$$

$a_0 = x_1 + x_2/2$ $\qquad$ $a_1 = x_2 - x_1/2$

Substituting these constants in above equation we get

$$X = a_o + a_1 \xi$$

$$X = \frac{x_1 + x_2}{2} + \frac{x_2 - x_1}{2} \xi$$

$$X = \frac{1-\xi}{2} x_1 + \frac{1+\xi}{2} x_2$$

$$X = N_1 X_1 + N_2 X_2$$

$$N_1 = \frac{1-\xi}{2} \qquad N_2 = \frac{1+\xi}{2}$$

Where N_1 and N_2 are called shape functions also called as interpolation functions.

These shape functions can also be derived using nodal displacements say q1 and q2 which are nodal displacements at node1 and node 2 respectively, now assuming the displacement function and following the same procedure as that of nodal coordinate we get

$$U = \alpha_o + \alpha_1 \xi$$

$$U = \frac{1-\xi}{2} q_1 + \frac{1+\xi}{2} q_2$$

$$U = N_1 q_1 + N_2 q_2$$

$$= [N_1 \quad N_2] \begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix}$$

$$U = Nq$$

$$U = Nq$$

Where N is the shape function matrix and q is displacement matrix. Once the displacement is known its derivative gives strain and corresponding stress can be determined as follows.

$$U = N\,q$$

$$\varepsilon = \frac{du}{dx} = \frac{du}{d\xi}\,\frac{d\xi}{dx}$$

$$\varepsilon = \frac{q_2 - q_1}{2}\,\frac{2}{x_2 - x_1}$$

$$\varepsilon = \frac{q_2 - q_1}{L} \qquad \text{where } L = x_2 - x_1$$

$$\varepsilon = \frac{1}{L}\,[-1 \quad 1]\begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix}$$

$$\varepsilon = B\,q$$

$$\text{where } B = \frac{1}{L}\,[-1 \quad 1]$$ element strain displacement matrix

$$\sigma = E\,\varepsilon = B\,q\,E$$

From the potential approach we have the expression of Π as

From the potential energy concept

$$\pi = \tfrac{1}{2}\int_V \sigma^T \varepsilon\, dv - \int_V u^T f_b\, dv - \int_S u^T T\, ds - \sum_{i=1}^{n} u_i\, p_i$$

Since body is divide

$$\pi_e = \int_e u_e - w_e \, dv$$

$$\pi = ½ \int B^T q^T E B q \, dv - \sum_{i=1}^{n} u_i p_i$$

Now total potential energy

$$\pi = \sum \pi_e = ½ Q^T \left(\int B^T EBAL \right) Q - \sum Qi^T Fi$$

$$\Pi = ½ Q^T K Q - Q^T F$$

To extremise the potential energy

$$\frac{d\pi}{dQ^T} = 0 = KQ - F$$

Third step in FEM is finding out stiffness matrix from the above equation we have the value of K as

$$K = \int_V B^T E B \, dv \qquad \text{where } B = \frac{1}{L} [-1 \quad 1]$$

For an element

$$K = \int_e B^T E B A \, dx$$

But

$$\frac{dx}{d\xi} = L/2$$

Therefore now substituting the limits as -1 to +1 because the value of ξ varies between -1 & 1 we have

$$K = \int_{-1}^{+1} B^T E\, B\, A \frac{L}{2} d\xi$$

Integration of above equations gives K which is given as

$$K = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

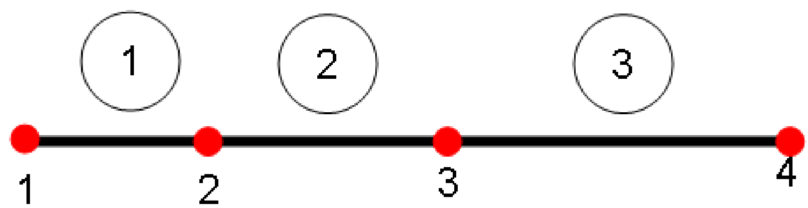
Fourth step is assembly and the size of the assembly matrix is given by number of nodes X degrees of freedom, for the present example that has four nodes and one degree of freedom at each node hence size of the assembly matrix is 4 X 4. At first determine the stiffness matrix of each element say k_1, k_2 and k_3 as

$$K_1 = \frac{A_1 E_1}{L_1} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \begin{bmatrix} \frac{A_1 E_1}{L_1} & -\frac{A_1 E_1}{L_1} \\ -\frac{A_1 E_1}{L_1} & \frac{A_1 E_1}{L_1} \end{bmatrix}$$

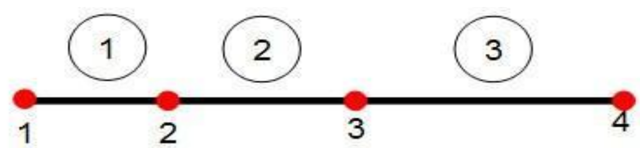
Similarly determine k_2 and k_3

$$K_2 = \begin{bmatrix} \frac{A_2E_2}{L_2} & -\frac{A_2E_2}{L_2} \\ -\frac{A_2E_2}{L_2} & \frac{A_2E_2}{L_2} \end{bmatrix} \quad K_3 = \begin{bmatrix} \frac{A_3E_3}{L_3} & -\frac{A_3E_3}{L_3} \\ -\frac{A_3E_3}{L_3} & \frac{A_3E_3}{L_3} \end{bmatrix}$$

The given system is modeled as three elements and four nodes we have three stiffness matrices.



Since node 2 is connected between element 1 and element 2, the elements of second stiffness matrix (k_2) gets added to second row second element as shown below similarly for node 3 it gets added to third row third element



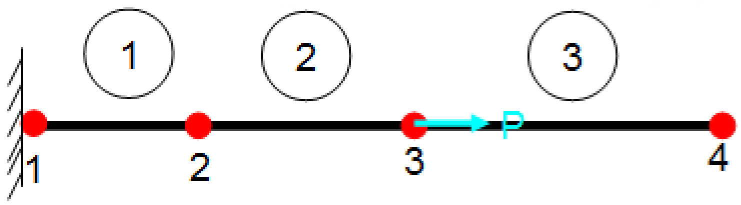
$$\begin{array}{c} \\ \\ \end{array}\begin{array}{cccc} 1 & 2 & 3 & 4 \end{array}$$

$$\begin{bmatrix} \frac{A_1E_1}{L_1} & -\frac{A_1E_1}{L_1} & 0 & 0 \\ -\frac{A_1E_1}{L_1} & \frac{A_1E_1}{L_1}+\frac{A_2E_2}{L_2} & -\frac{A_2E_2}{L_2} & 0 \\ 0 & -\frac{A_2E_2}{L_2} & \frac{A_2E_2}{L_2}+\frac{A_3E_3}{L_3} & -\frac{A_3E_3}{L_3} \\ 0 & 0 & -\frac{A_3E_3}{L_3} & \frac{A_3E_3}{L_3} \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array}$$

Fifth step is applying the boundary conditions for a given system. We have the equation of equilibrium KQ=F

K = global stiffness matrix
Q = displacement matrix
F= global force vector

Let Q1, Q2, Q3, and Q4 be the nodal displacements at node 1 to node 4 respectively. And F1, F2, F3, F4 be the nodal load vector acting at node 1 to node 4 respectively.



Given system is fixed at one end and force is applied at other end. Since node 1 is fixed displacement at node 1 will be zero, so set q1 =0. And node 2, node 3 and node 4 are free to move hence there will be displacement that has to be determined. But in the load vector because of fixed node 1 there will reaction force say R1. Now replace F1 to R1 and also at node 3 force P is applied hence replace F3 to P. Rest of the terms are zero.

After applying BC,s

$$\begin{pmatrix} \frac{A_1E_1}{L_1} & -\frac{A_1E_1}{L_1} & 0 & 0 \\ -\frac{A_1E_1}{L_1} & \frac{A_1E_1}{L_1}+\frac{A_2E_2}{L_2} & -\frac{A_2E_2}{L_2} & 0 \\ 0 & -\frac{A_2E_2}{L_2} & \frac{A_2E_2}{L_2}+\frac{A_3E_3}{L_3} & -\frac{A_3E_3}{L_3} \\ 0 & 0 & -\frac{A_3E_3}{L_3} & \frac{A_3E_3}{L_3} \end{pmatrix} \begin{pmatrix} 0 \\ Q_2 \\ Q_3 \\ Q_4 \end{pmatrix} = \begin{pmatrix} R_1 \\ 0 \\ P \\ 0 \end{pmatrix}$$

Sixth step is solving the above matrix to determine the displacements which can be solved either by

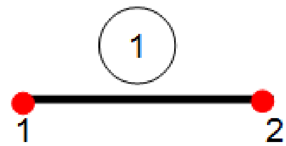
- Elimination method
- Penalty approach method

Details of these two methods will be seen in later sections.

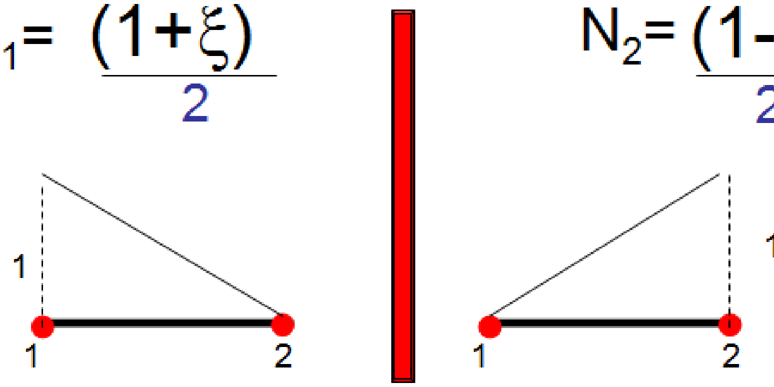
Last step is the presentation of results, finding the parameters like displacements, stresses and other required parameters.

Body force distribution for 2 noded bar element

We derived shape functions for 1D bar, variation of these shape functions is shown below .As a property of shape function the value of N_1 should be equal to 1 at node 1 and zero at rest other nodes (node 2).



$$N_1 = \frac{(1+\xi)}{2} \qquad N_2 = \frac{(1-\xi)}{2}$$



From the potential energy of an elastic body we have the expression of work done by body force as

$$\int_V u^T f_b \, dv$$

$$U = N_1 q_1 + N_2 q_2$$

For an element

$$\int_e u^T f_b A \, dx$$

Where f_b is the body acting on the system. We know the displacement function $U = N_1q_1 + N_2q_2$ substitute this U in the above equation we get

$$= A f_b \int_e (N_1q_1 + N_2q_2)\, dx$$

$$= A f_b \int_e [N_1 \;\; N_2] \begin{pmatrix} q_1 \\ q_2 \end{pmatrix} dx$$

$$= A f_b \int_e \underbrace{[q_1 \;\; q_2]}_{qT} \begin{pmatrix} N_1 \\ N_2 \end{pmatrix} dx$$

$$= A f_b \; qT \int_e \begin{pmatrix} N_1 \\ N_2 \end{pmatrix} dx$$

$$= qT \begin{pmatrix} A f_b \int_e N_1 \, dx \\ A f_b \int_e N_2 \, dx \end{pmatrix}$$

Now

$$\int_e N_1 \, dx = \int_e \frac{1-\xi}{2} \, dx$$

but $\frac{dx}{d\xi} = l_{e/2}$

$$= \int_{-1}^{+1} \frac{1-\xi}{2} \frac{l_e}{2} \, d\xi = \frac{l_e}{2}$$

Similarly

$$\int_e N_2 \, dx = \frac{l_e}{2}$$

Therefore

$$\int u^T f_b A \, dx = q^T \underbrace{A f_b \frac{l_e}{2} \begin{bmatrix} 1 \\ 1 \end{bmatrix}}_{f_e}$$

This amount of body force will be distributed at 2 nodes hence the expression as 2 in the denominator.

Surface force distribution for 2 noded bar element

Now again taking the expression of work done by surface force from potential energy concept and following the same procedure as that of body we can derive the expression of surface force as

$$\int_s u^T T\, ds = \int_e u^T T\, dx$$

$$= q^T \underbrace{\frac{l_e T}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}}_{T_e}$$

Where T_e is element surface force distribution.

Methods of handling boundary conditions

We have two methods of handling boundary conditions namely Elimination method and penalty approach method. Applying BC's is one of the vital role in FEM improper specification of boundary conditions leads to erroneous results. Hence BC's need to be accurately modeled.

Elimination Method: let us consider the single boundary conditions say $Q_1 = a_1$.Extremising Π results in equilibrium equation.

$Q = [Q_1, Q_2, Q_3 \ldots\ldots\ldots Q_N]^T$ be the displacement vector and

$F = [F_1, F_2, F_3 \ldots\ldots\ldots\ldots F_N]^T$ be load vector

Say we have a global stiffness matrix as

$$K = \begin{matrix} K_{11} & K_{12} \ldots\ldots\ldots\ldots K_{1N} \\ K_{21} & K_{22} \ldots\ldots\ldots\ldots K_{2N} \\ . & \\ . & \\ . & \\ K_{N1} & K_{N2} \ldots\ldots\ldots\ldots K_{NN} \end{matrix}$$

Now potential energy of the form $\Pi = \frac{1}{2} Q^T K Q - Q^T F$ can written as

$$
\begin{aligned}
\Pi = \frac{1}{2} (&Q_1K_{11}Q_1 + Q_1K_{12}Q_2 + \ldots + Q_1K_{1N}Q_N \\
&+ Q_2K_{21}Q_1 + Q_2K_{22}Q_2 + \ldots + Q_2K_{2N}Q_N \\
&\ldots \\
&\ldots \\
&+ Q_NK_{N1}Q_1 + Q_NK_{N2}Q_2 + \ldots + Q_NK_{NN}Q_N) \\
&- (Q_1F_1 + Q_2F_2 + \ldots + Q_NF_N)
\end{aligned}
$$

Substituting $Q_1 = a_1$ we have

$$
\begin{aligned}
\Pi = \frac{1}{2} (&a_1K_{11}a_1 + a_1K_{12}Q_2 + \ldots + a_1K_{1N}Q_N \\
&+ Q_2K_{21}a_1 + Q_2K_{22}Q_2 + \ldots + Q_2K_{2N}Q_N \\
&\ldots \\
&\ldots \\
&+ Q_NK_{N1}a_1 + Q_NK_{N2}Q_2 + \ldots + Q_NK_{NN}Q_N) \\
&- (a_1F_1 + Q_2F_2 + \ldots + Q_NF_N)
\end{aligned}
$$

Extremizing the potential energy

ie $d\Pi/dQi = 0$ gives

Where $i = 2, 3 \ldots N$

$$
\begin{aligned}
K_{22}Q_2 + K_{23}Q_3 + \ldots + K_{2N}Q_N &= F_2 - K_{21}a_1 \\
K_{32}Q_2 + K_{33}Q_3 + \ldots + K_{3N}Q_N &= F_3 - K_{31}a_1 \\
&\ldots \\
K_{N2}Q_2 + K_{N3}Q_3 + \ldots + K_{NN}Q_N &= F_N - K_{N1}a_1
\end{aligned}
$$

Writing the above equation in the matrix form we get

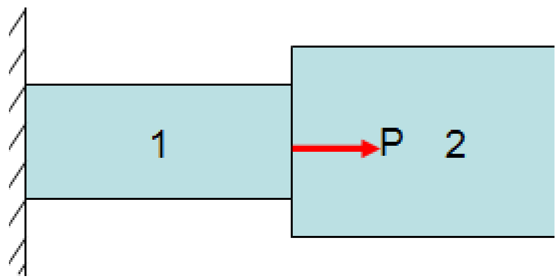
$$
\begin{matrix}
K_{22} & K_{23} \ldots K_{2N} \\
K_{32} & K_{33} \ldots K_{2N} \\
. & \\
. & \\
. & \\
K_{N2} & K_{N3} \ldots K_{NN}
\end{matrix}
\quad
\begin{matrix}
Q_2 \\ Q_3 \\ . \\ . \\ \\ Q_N
\end{matrix}
\quad = \quad
\begin{matrix}
F_2 - K_{21}a_1 \\ F_3 - K_{31}a_1 \\ \\ \\ \\ F_N - K_{N1}a_1
\end{matrix}
$$

Now the N X N matrix reduces to N-1 x N-1 matrix as we know $Q_1=a_1$ ie first row and first column are eliminated because of known Q_1. Solving above matrix gives displacement components. Knowing the displacement field corresponding stress can be calculated using the relation $\sigma = \varepsilon Bq$.

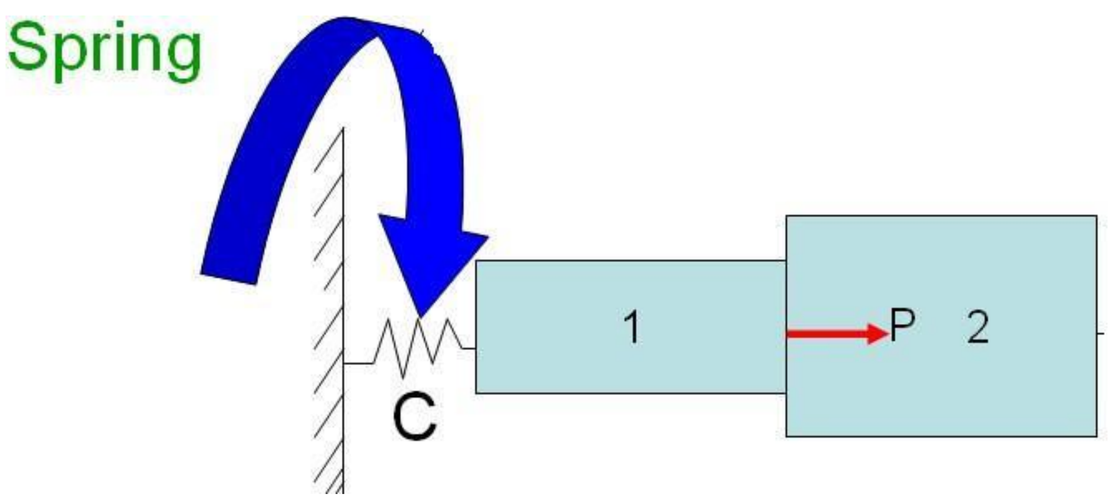
Reaction forces at fixed end say at node1 is evaluated using the relation

$$R_1= K_{11}Q_1+K_{12}Q_2+\ldots\ldots\ldots\ldots+K_{1N}Q_N-F_1$$

Penalty approach method: let us consider a system that is fixed at both the ends as shown



In penalty approach method the same system is modeled as a spring wherever there is a support and that spring has large stiffness value as shown.



Let a_1 be the displacement of one end of the spring at node 1 and a_3 be displacement at node 3. The displacement Q_1 at node 1 will be approximately equal to a_1, owing to the relatively small resistance offered by the structure. Because of the spring addition at the support the strain energy also comes into the picture of Π equation .Therefore equation Π becomes

$$\Pi = \frac{1}{2} Q^T K Q + \frac{1}{2} C (Q_1 - a_1)^2 - Q^T F$$

The choice of C can be done from stiffness matrix as

$$C = \max [K_{ij}] \times 10^4$$

We may also choose 10^5 &10^6 but 10^4 found more satisfactory on most of the computers.

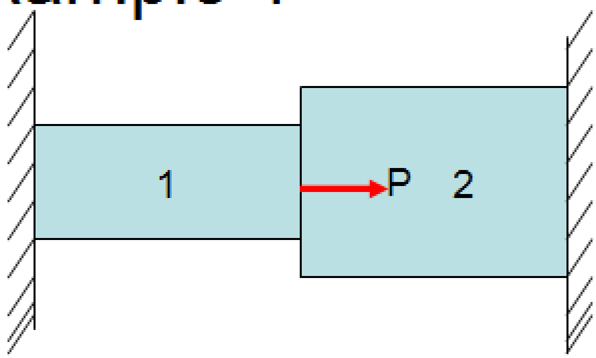
Because of the spring the stiffness matrix has to be modified ie the large number c gets added to the first diagonal element of K and Ca_1 gets added to F_1 term on load vector. That results in.

$$\begin{pmatrix} K_{11} + C & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{pmatrix} \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} = \begin{pmatrix} F1 \\ F2 \\ F3 \end{pmatrix} + C\,a1$$

A reaction force at node 1 equals the force exerted by the spring on the system which is given by

$$\text{Reaction forces} = -C(Q_1 - a_1)$$

Example 1



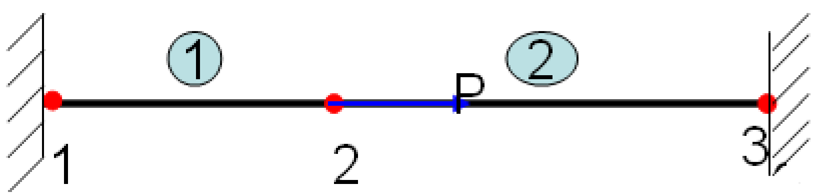
$A_1 = 900mm^2$ $A_2 = 1200mm^2$

$E_1 = 70 \times 10^9\ N/m^2$ $E_2 = 200 \times 10^9\ N/m^2$

$L_1 = 200mm$ $L_2 = 300mm$

$P = 300\ KN$

To solve the system again the seven steps of FEM has to be followed, first 2 steps contain modeling and discretization. this result in



Third step is finding stiffness matrix of individual elements

$$K_1 = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{900 \times 0.75 \times 10^5}{200}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^5 \begin{array}{c} \begin{matrix} 1 & \quad 2 \end{matrix} \\ \begin{bmatrix} 3.15 & -3.15 \\ -3.15 & 3.15 \end{bmatrix} \end{array} \begin{matrix} 1 \\ 2 \end{matrix}$$

Similarly

$$K_2 = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^5 \begin{array}{c} \begin{array}{cc} 2 & 3 \end{array} \\ \begin{bmatrix} 8 & -8 \\ -8 & 8 \end{bmatrix} \end{array} \begin{array}{c} 2 \\ 3 \end{array}$$

Next step is assembly which gives global stiffness matrix

$$K = \begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 3.15 & -3.15 & 0 \\ -3.15 & 3.15+8 & -8 \\ 0 & -8 & 8 \end{bmatrix} \end{array} 10^5 \begin{array}{c} 1 \\ 2 \\ 3 \end{array}$$

Now determine global load vector

$$F = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 300 \times 10^3 \\ R_3 \end{Bmatrix}$$

We have the equilibrium condition KQ=F

$$10^5 \begin{bmatrix} 3.15 & -3.15 & 0 \\ -3.15 & 3.15+8 & -8 \\ 0 & -8 & 8 \end{bmatrix} \begin{Bmatrix} Q1 \\ Q2 \\ Q3 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 300 \times 10^3 \\ R_3 \end{Bmatrix} \begin{matrix} \\ -(-3.15 \times 10^5 \times Q1) \\ -(0 \times Q1) \end{matrix}$$

After applying elimination method we have Q2 = 0.26mm

Once displacements are known stress components are calculated as follows

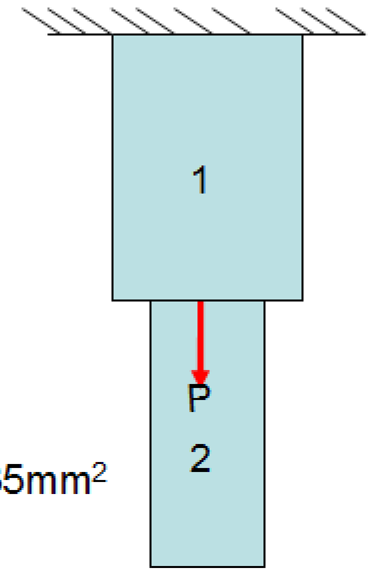
For element 1

$$\sigma_1 = E_1 \frac{1}{L_1} [-1 \quad 1] \begin{Bmatrix} Q1 \\ Q2 \end{Bmatrix} = 94.17 \text{ N/mm}^2$$

For element 2

$$\sigma_2 = E_2 \frac{1}{L_2} [-1 \quad 1] \begin{Bmatrix} Q2 \\ Q3 \end{Bmatrix} = -179.34 \text{ N/mm}^2$$

Example 2



$E_1 = 2.06 \times 10^5$ MPa

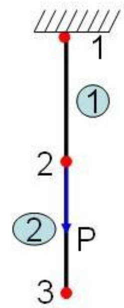
$A_1 = 3387.09 mm^2$ $A_2 = 2419.35 mm^2$

$L_1 = L_2 = 304.8 mm$

P = 444.8N

Body force = $f_b = 7.69 \times 10^{-5} N/mm^3$

Solution:



$$K_1 = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^6 \begin{bmatrix} 2.28 & -2.28 \\ -2.28 & 2.28 \end{bmatrix} \begin{matrix} 1 \\ 2 \end{matrix}$$

(columns 1, 2)

$$K_2 = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^6 \begin{bmatrix} 1.63 & -1.63 \\ -1.63 & 1.63 \end{bmatrix} \begin{matrix} 2 \\ 3 \end{matrix}$$

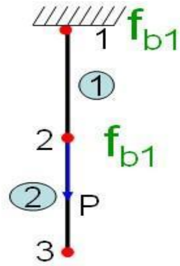
(columns 2, 3)

$$K = \begin{bmatrix} 2.28 & -2.28 & 0 \\ -2.28 & 2.28+1.63 & -1.63 \\ 0 & -1.63 & 1.63 \end{bmatrix} 10^6 \begin{matrix} 1 \\ 2 \\ 3 \end{matrix}$$

(columns 1, 2, 3)

Body force terms

Element 1



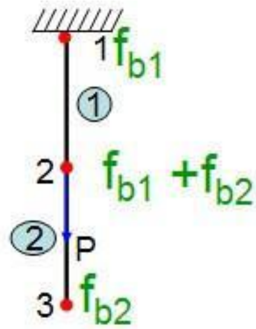
$$f_{b1} = \frac{A_1 f_b L_1}{2}\begin{Bmatrix}1\\1\end{Bmatrix}\begin{matrix}1\\2\end{matrix}$$

$$= \frac{3387.09 \times 7.69 \times 10^{-5} \times 304.8}{2}\begin{Bmatrix}1\\1\end{Bmatrix}\begin{matrix}1\\2\end{matrix}$$

$$= \begin{Bmatrix}39.69\\39.69\end{Bmatrix}\begin{matrix}1\\2\end{matrix}$$

Body force terms

Element 2

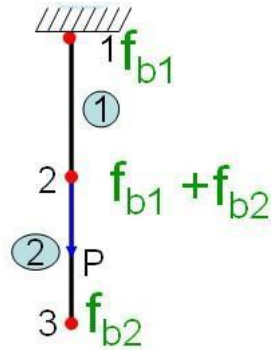


$$f_{b2} = \frac{A_2 f_b L_2}{2}\begin{Bmatrix}1\\1\end{Bmatrix}\begin{matrix}2\\3\end{matrix}$$

$$= \frac{2419.35 \times 7.69 \times 10^{-5} \times 304.8}{2}\begin{Bmatrix}1\\1\end{Bmatrix}\begin{matrix}2\\3\end{matrix}$$

$$= \begin{Bmatrix}28.3\\28.3\end{Bmatrix}\begin{matrix}2\\3\end{matrix}$$

Global load vector:



$$F = \begin{pmatrix} F_1 \\ F_2 \\ F_3 \end{pmatrix} = \begin{pmatrix} f_{b1} \\ p + f_{b1} + f_{b2} \\ fb2 \end{pmatrix} = \begin{pmatrix} 39.69 \\ 512.8 \\ 28.3 \end{pmatrix}$$

We have the equilibrium condition KQ=F

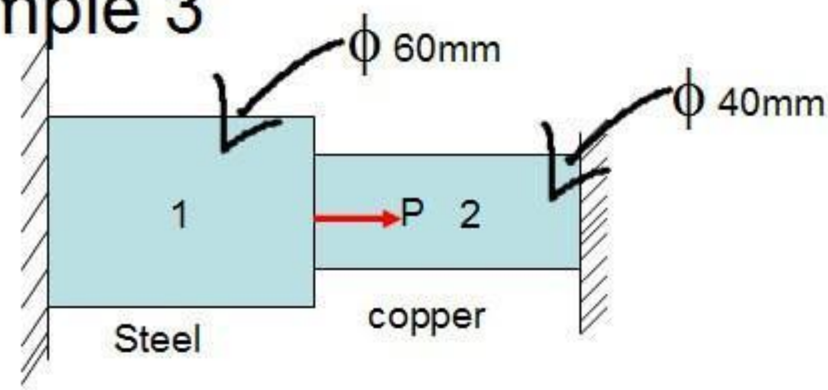
$$10^6 \begin{pmatrix} 2.28 & -2.28 & 0 \\ -2.28 & 6.92 & -16.3 \\ 0 & -1.63 & 1.63 \end{pmatrix} \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} = \begin{pmatrix} 39.69 + R1 \\ 512.8 \\ 28.3 \end{pmatrix}$$

$Q2 = 0.23 \times 10^{-3}$mm

$Q3 = 2.5 \times 10^{-4}$mm

After applying elimination method and solving matrices we have the value of displacements as $Q2 = 0.23 \text{ X } 10^{-3}$mm & $Q3 = 2.5\text{X}10^{-4}$mm

Example 3

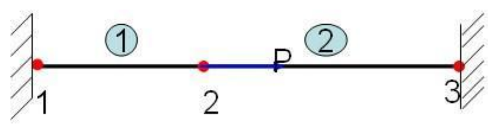


$E_1 = 2 \times 10^5$ MPa $E_2 = 1 \times 10^5$ MPa

$L_1 = 800$ mm $L_2 = 500$mm

$P = 100$ KN

Solution:



$$A_1 = \pi/4\,(60)^2 = 2827.43\text{mm}^2$$

$$A_2 = \pi/4\,(40)^2 = 1256.63\text{mm}^2$$

$$K_1 = \frac{A_1E_1}{L_1}\begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix} = \frac{2827.43 \times 2 \times 10^5}{800}\begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix} = 10^5 \begin{bmatrix} 7.06 & -7.06 \\ -7.06 & 7.06 \end{bmatrix} \begin{matrix} 1 \\ 2 \end{matrix}$$

(columns 1, 2)

$$K_2 = \frac{A_2E_2}{L_2}\begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix} = 10^5 \begin{bmatrix} 2.51 & -2.51 \\ -2.51 & 2.51 \end{bmatrix} \begin{matrix} 2 \\ 3 \end{matrix}$$

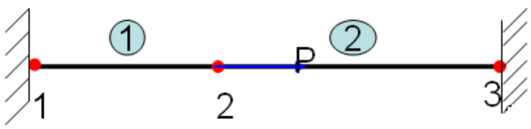
(columns 2, 3)

Global stiffness matrix

$$K = \begin{bmatrix} 7.07 & -7.07 & 0 \\ -7.07 & 9.583 & -2.513 \\ 0 & -2.513 & 2.513 \end{bmatrix} 10^5$$

(Columns and rows are numbered 1, 2, 3.)

Global load vector:



$$F = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 100 \times 10^3 \\ 0 \end{Bmatrix}$$

Equilibrium Equation

$$K\,Q = F$$

$$K = \begin{bmatrix} 7.07 & -7.07 & 0 \\ -7.07 & 9.583 & -2.513 \\ 0 & -2.513 & 2.513 \end{bmatrix} 10^5 \begin{Bmatrix} Q1 \\ Q2 \\ Q3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 100 \times 10^3 \\ 0 \end{Bmatrix}$$

$$C = \max [\, K_{ij} \,] \times 10^4 = 9.583 \times 10^5 \times 10^4$$

Modification required

$$\begin{pmatrix} 7.07 + C & -7.07 & 0 \\ -7.07 & 9.583 & -2.513 \\ 0 & -2.513 & 2.513 + C \end{pmatrix} 10^5 \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} = \begin{pmatrix} 0 + C\,a1 \\ 100 \times 10^3 \\ 0 + C\,a3 \end{pmatrix}$$

After Modification

$$\begin{pmatrix} 9.583 \times 10^4 & -7.07 & 0 \\ -7.07 & 9.583 & -2.513 \\ 0 & -2.513 & 9.583 \times 10^4 \end{pmatrix} 10^5 \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} = \begin{pmatrix} 0 \\ 100 \times 10^3 \\ 0 \end{pmatrix}$$

Solving the matrix we have

$Q1 = 7.698 \times 10^{-6}mm$, $Q2 = 0.104mm$, $Q3 = 2.736 \times 10^{-6}mm$

Reaction forces

@ node 1

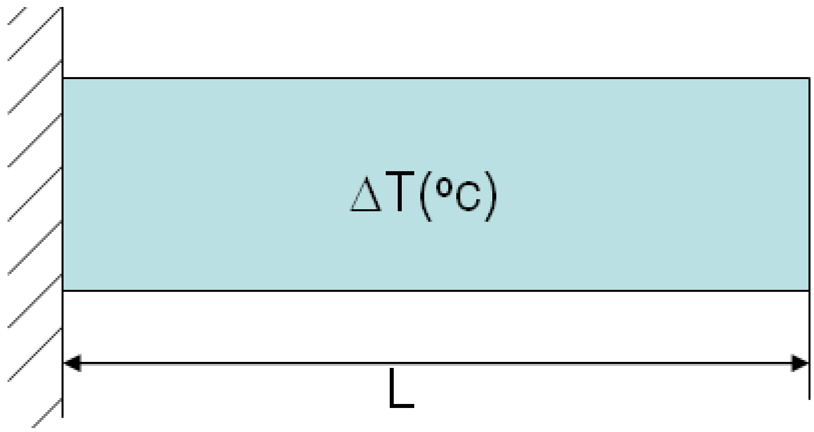
$$R_1 = C(Q1 - a1) = -73597.44N$$

@ node 3

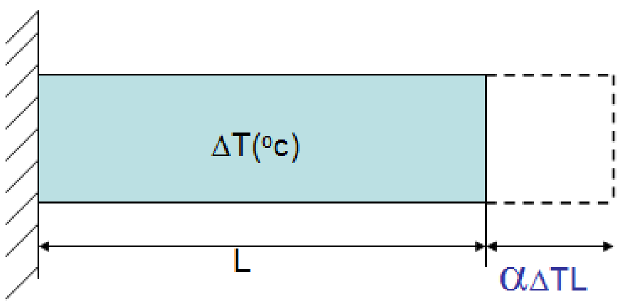
$$R_3 = C(Q3 - a3) = -26219.08N$$

Temperature effect on 1D bar element

Lets us consider a bar of length L fixed at one end whose temperature is increased to ΔT as shown.



Because of this increase in temperature stress induced are called as thermal stress and the bar gets expands by a amount equal to αΔTL as shown. The resulting strain is called as thermal strain or initial strain

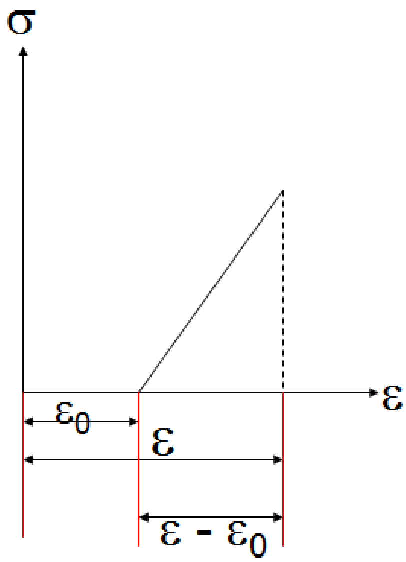


α = coefficient of thermal expansion

$$\varepsilon_o = \frac{\alpha \Delta T L}{L} = \alpha \Delta T$$

ε_o → Thermal strain (initial strain)

In the presence of this initial strain variation of stress strain graph is as shown below



Hooke's law

$$\frac{\text{Stress}}{\text{Strain}} = \frac{\sigma}{\varepsilon - \varepsilon_0} = E$$

$$\sigma = (\varepsilon - \varepsilon_0)\, E$$

We know that

Strain energy in a bar

$$U = \tfrac{1}{2} \int \sigma^T \varepsilon \, dv$$

For an element

$$U = \tfrac{1}{2} \int_e \sigma^T \varepsilon \, A \, dx$$

Therefore

$$U = \tfrac{1}{2} \int_e E \, (\varepsilon - \varepsilon_0)^T (\varepsilon - \varepsilon_0) \, A \, dx$$

$$U = \tfrac{1}{2} \int_e E \, (Bq - \varepsilon_0)^T (Bq - \varepsilon_0) \, A \, dx$$

$$U = \tfrac{1}{2} \int_e E\,(Bq - \varepsilon_0)^T (Bq - \varepsilon_0)\, A\, dx$$

But $dx/d\xi = L_e/2$

$$U = \tfrac{1}{2} EA \int_e (Bq - \varepsilon_0)^T (Bq - \varepsilon_0)\, Le/2\, d\xi$$

$$U = \tfrac{1}{2} EA/2 \int_e (Bq - \varepsilon_0)^T (Bq - \varepsilon_0)\, Le\, d\xi$$

$$U = \tfrac{1}{2} EA/2 \int_e (B^T q^T - \varepsilon_0)(Bq - \varepsilon_0)\, Le\, d\xi$$

$$U = \tfrac{1}{2} Le\, EA/2 \int_e [B^T q^T Bq - B^T q^T \varepsilon_0 - Bq\varepsilon_0 + \varepsilon_0^2]\, d\xi$$

$$U = \tfrac{1}{2} Le\, EA/2 \int_e [B^T q^T Bq - \varepsilon_0 (B^T q^T + Bq) + \varepsilon_0^2]\, d\xi$$

Therefore

$$U = \tfrac{1}{2} Le\, EA/2 \int_e [B^T q^T Bq - \varepsilon_0\, 2B^T q^T + \varepsilon_0^2]\, d\xi$$

Integrating individual terms

$$U = \frac{1}{2} q^T EA \frac{Le}{2} \int_e [B^T B\, d\xi]\, q$$

Stiffness matrix

$$- \frac{1}{2} q^T EA \frac{Le}{2} \varepsilon_0 \int_e 2B^T\, d\xi$$

Thermal load vector

$$+ \frac{1}{2} EA \frac{Le}{2} \int_e \varepsilon_0^2\, d\xi \rightarrow 0$$

Extremizing the potential energy first term yields stiffness matrix, second term results in thermal load vector and last term eliminates that do not contain displacement filed

Thermal load vector

From the above expression taking the thermal load vector lets derive what is the effect of thermal load.

$$\theta_e = \frac{1}{2} \, EA \frac{Le}{2} \varepsilon_0 \int_e 2B^T \, d\xi$$

$$= \frac{1}{\;} \, EA \frac{Le}{2} \varepsilon_0 \int_e B^T \, d\xi$$

We know that $B^T = \frac{1}{L} \begin{bmatrix} -1 \\ 1 \end{bmatrix}$

$$\theta_e = \frac{EA}{2} \varepsilon_0 \int_{-1}^{1} \begin{bmatrix} -1 \\ 1 \end{bmatrix} d\xi$$

$$= \frac{EA}{2} \varepsilon_0 \begin{bmatrix} -2 \\ 2 \end{bmatrix}$$

$$= EA \; \varepsilon_0 \begin{bmatrix} -1 \\ 1 \end{bmatrix}$$

$$\theta = EA\alpha\Delta T \begin{bmatrix} -1 \\ 1 \end{bmatrix}$$

Stress component because of thermal load

$$\sigma = (\varepsilon - \varepsilon_0)\, E$$

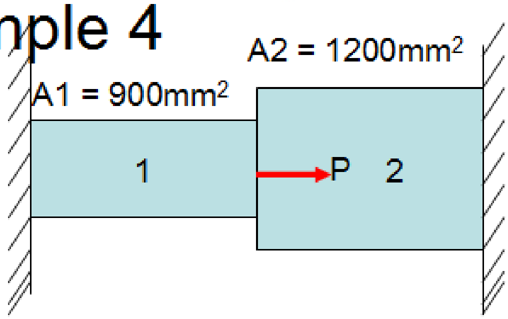
We know $\varepsilon = Bq$ and $\varepsilon_o = \alpha\Delta T$ substituting these in above equation we get

$$= (Bq - \alpha\Delta T)\, E$$

$$= E\, Bq - E\, \alpha\Delta T$$

$$\sigma = E\, \frac{1}{L} [-1 \quad 1] q - E\, \alpha\Delta T$$

Example 4



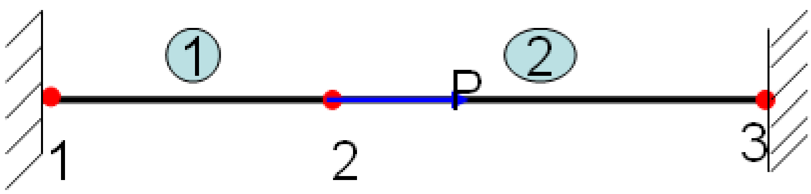
$\alpha_1 = 23 \times 10^{-6}$ Per ^0C $\alpha_2 = 11.7 \times 10^{-6}$ Per ^0C

$E_1 = 70 \times 10^9\ N/m^2$ $E_2 = 200 \times 10^9\ N/m^2$

$L_1 = 200mm$ $L_2 = 300mm$

P = 300 KN is applied at 20^0c ,the temperature is then raised to 60^0c

Solution:



$$K_1 = \frac{A_1E_1}{L_1}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{900x70x\ 10^3}{200}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^3 \begin{bmatrix} 315 & -315 \\ -315 & 315 \end{bmatrix}\begin{matrix} 1 \\ 2 \end{matrix}$$

(column labels: 1, 2)

$$K_2 = \frac{A_2E_2}{L_2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^3 \begin{bmatrix} 800 & -800 \\ -800 & 800 \end{bmatrix}\begin{matrix} 2 \\ 3 \end{matrix}$$

(column labels: 2, 3)

Global stiffness matrix:

$$K = \begin{bmatrix} 315 & -315 & 0 \\ -315 & 1115 & -800 \\ 0 & -800 & 800 \end{bmatrix} 10^3$$

(Column labels: 1, 2, 3; row labels: 1, 2, 3)

Thermal load vector:

We have the expression of thermal load vector given by

$$\theta = EA\alpha\Delta T \begin{Bmatrix} -1 \\ 1 \end{Bmatrix}$$

Element 1

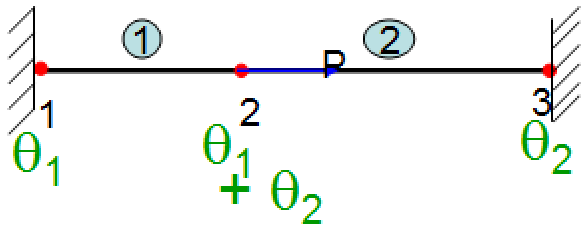
$$\theta_1 = 70 \times 10^3 \times 900 \times 23 \times 10^{-6} \times 40 \begin{Bmatrix} -1 \\ 1 \end{Bmatrix} \begin{matrix} 1 \\ 2 \end{matrix}$$

$$\theta_1 = 10^3 \begin{Bmatrix} -57.96 \\ 57.96 \end{Bmatrix} \begin{matrix} 1 \\ 2 \end{matrix}$$

Similarly calculate thermal load distribution for second element

$$\theta_2 = 10^3 \begin{Bmatrix} -112.32 \\ 112.32 \end{Bmatrix} \begin{matrix} 2 \\ 3 \end{matrix}$$

Global load vector:



$$F = \begin{pmatrix} F_1 \\ F_2 \\ F_3 \end{pmatrix} = \begin{pmatrix} \theta_1 \\ P + \theta_2 + \theta_1 \\ \theta_2 \end{pmatrix} = \begin{pmatrix} -57.96 \\ 245.64 \\ 112.32 \end{pmatrix} 10^3$$

From the equation KQ=F we have

$$\begin{pmatrix} 315 & -315 & 0 \\ -315 & 1115 & -800 \\ 0 & -800 & 800 \end{pmatrix} 10^3 \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} = \begin{pmatrix} -57.96 \\ 245.64 \\ 112.32 \end{pmatrix} 10^3 \; -(-315 \times 10^3)Q1 - (-8 \times 10^5)Q3 \quad -(0)Q1$$

After applying elimination method and solving the matrix we have

Q2= 0.22mm

Stress in each element:

For element 1

$$\sigma_1 = E_1 \frac{1}{L_1} [-1 \quad 1] \begin{Bmatrix} Q1 \\ Q2 \end{Bmatrix} - E_1 \alpha_1 \Delta T$$

$= 12.60MPa$

For element 2

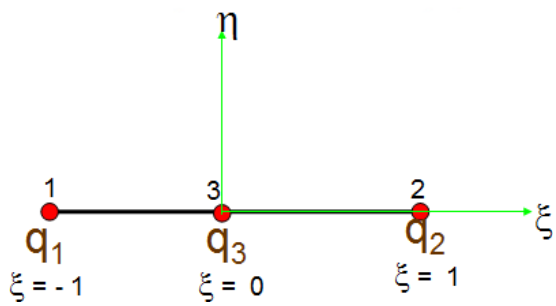
$$\sigma_2 = E_2 \frac{1}{L_2} [-1 \quad 1] \begin{Bmatrix} Q2 \\ Q3 \end{Bmatrix} - E_2 \alpha_2 \Delta T$$

$= -240.27MPa$

Quadratic 1D bar element

In the previous sections we have seen the formulation of 1D linear bar element , now lets move a head with quadratic 1D bar element which leads to for more accurate results . linear element has two end nodes while quadratic has 3 equally spaced nodes ie we are introducing one more node at the middle of 2 noded bar element.

Consider a quadratic element as shown and the numbering scheme will be followed as left end node as 1, right end node as 2 and middle node as 3.



Let's assume a polynomial as

$$U = \alpha_o + \alpha_1 \xi + \alpha_2 \xi^2$$

Now applying the conditions as

@ node 1	$u = q_1$	$\xi = -1$
@ node 2	$u = q_2$	$\xi = 1$
@ node 3	$u = q_3$	$\xi = 0$

ie

$$q_1 = \alpha_o - \alpha_1 + \alpha_2$$

$$q_2 = \alpha_o + \alpha_1 + \alpha_2$$

$$q_3 = \alpha_o$$

Solving the above equations we have the values of constants

$$\alpha_1 = \frac{q_2 - q_1}{2} \qquad \alpha_2 = \frac{q_1 + q_2 - 2q_3}{2}$$

And substituting these in polynomial we get

$$U = \alpha_o + \alpha_1 \xi + \alpha_2 \xi^2$$

$$= q_3 + \left(\frac{q_2 - q_1}{2}\right)\xi + \left(\frac{q_1 + q_2 - 2q_3}{2}\right)\xi^2$$

$$= \frac{\xi(\xi-1)}{2} q_1 + \frac{\xi(\xi+1)}{2} q_2 + (1-\xi^2) q_3$$

Or

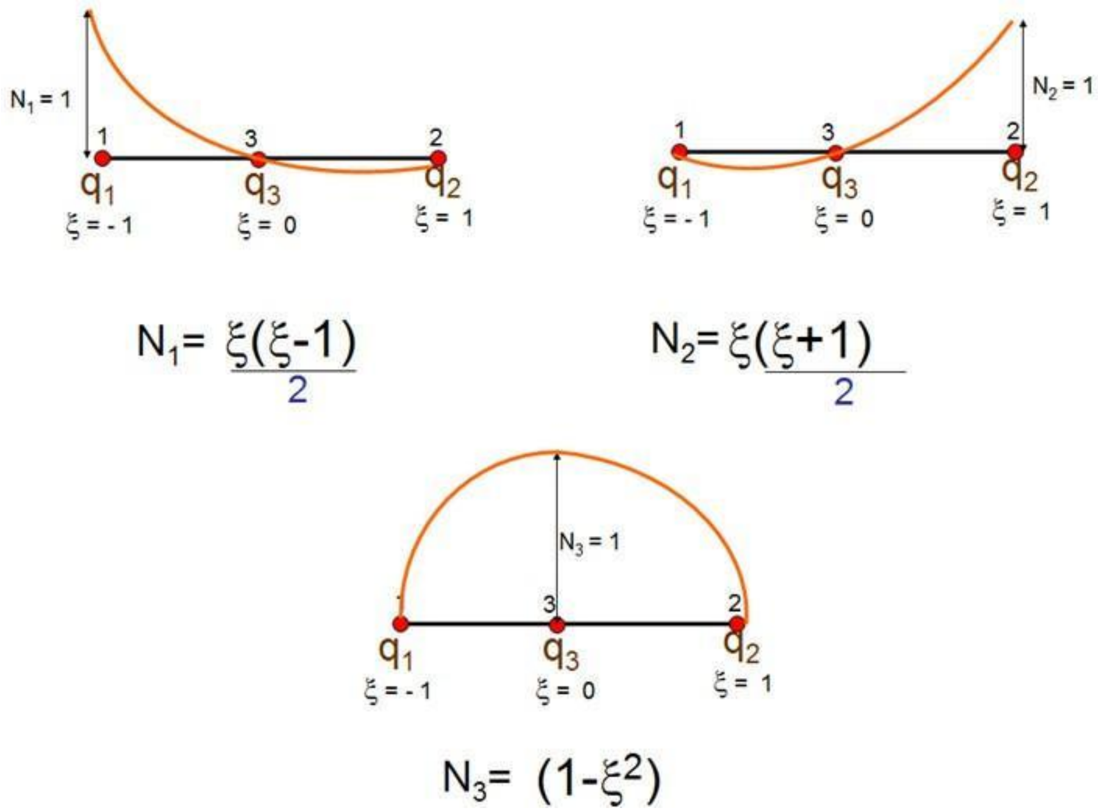
$$U = N_1 q_1 + N_2 q_2 + N_3 q_3$$

Where N1 N_2 N_3 are the shape functions of quadratic element

$$N_1 = \frac{\xi(\xi-1)}{2}$$

$$N_2 = \frac{\xi(\xi+1)}{2}$$

$$N_3 = (1-\xi^2)$$



Graphs show the variation of shape functions within the element .The shape function N_1 is equal to 1 at node 1 and zero at rest other nodes (2 and 3). N_2 equal to 1 at node 2 and zero at rest other nodes(1 and 3) and N_3 equal to 1 at node 3 and zero at rest other nodes(1 and 2)

Element strain displacement matrix If the displacement field is known its derivative gives strain and corresponding stress can be determined as follows

WKT

$$U = N_1 q_1 + N_2 q_2 + N_3 q_3$$

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

$$= \frac{du}{d\xi} \frac{d\xi}{dx}$$ By chain rule

Now

$$\frac{du}{d\xi} = \frac{d[N_1 q_1 + N_2 q_2 + N_3 q_3]}{d\xi}$$

Splitting the above equation into the matrix form we have

$$\frac{du}{d\xi} = \frac{d[N_1\ N_2\ N_3]}{d\xi} \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix}$$

$$\frac{du}{d\xi} = \begin{pmatrix} \frac{(2\xi-1)}{2} & \frac{(2\xi+1)}{2} & -2\xi \end{pmatrix} \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix}$$

Therefore

$$\varepsilon = \frac{du}{dx} = \frac{du}{d\xi}\frac{d\xi}{dx}$$

$$= \left(\frac{(2\xi-1)}{2} \quad \frac{(2\xi-1)}{2} \quad -2\xi \right) \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix} \frac{d\xi}{dx}$$

$$= \frac{2}{le}\left(\frac{(2\xi-1)}{2} \quad \frac{(2\xi+1)}{2} \quad -2\xi \right) \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \end{Bmatrix}$$

$$\varepsilon = Bq$$

B is element strain displacement matrix for 3 noded bar element

Stiffness matrix:

We know the stiffness matrix equation

$$K = \int_v B^T E B \, dv$$

For an element

$$K = \int_e B^T E B A \, dx$$

$$= \int_e B^T E B \frac{A L_e}{2} d\xi$$

Taking the constants outside the integral we get

$$K = \frac{E\,A\,L_e}{2} \int_e B^T\,B\,d\xi$$

Where

$$B = \frac{2}{le}\begin{bmatrix} \frac{(2\xi-1)}{2} & \frac{(2\xi+1)}{2} & -2\xi \end{bmatrix}$$

and B^T

$$B^T = \frac{2}{le}\begin{bmatrix} \frac{(2\xi-1)}{2} \\ \frac{(2\xi+1)}{2} \\ -2\xi \end{bmatrix}$$

Now taking the product of B^T X B and integrating for the limits -1 to +1 we get

$$K = \frac{E\,A\,L_e}{2} \int_e B^T\,B\,d\xi$$

$$= \frac{E\,A\,L_e}{2} \int_{-1}^{+1} \frac{4}{L_e^2} \begin{bmatrix} \frac{1}{4}(2\xi-1)^2 & \frac{1}{4}(2\xi-1)(2\xi+1) & -(2\xi-1)\xi \\ \frac{1}{4}(2\xi-1)(2\xi+1) & \frac{1}{4}(2\xi+1)^2 & -(2\xi+1)\xi \\ -(2\xi-1)\xi & -(2\xi+1)\xi & 4\xi^2 \end{bmatrix}$$

Integration of a matrix results in

$$K = \frac{EA}{3L_e}\begin{pmatrix} 7 & 1 & -8 \\ 1 & 7 & -8 \\ -8 & -8 & 16 \end{pmatrix}$$

Body force term & surface force term can be derived as same as 2 noded bar element and for quadratic element we have

Body force:

$$f_e = A f_b l_e \begin{pmatrix} 1/6 \\ 1/6 \\ 2/3 \end{pmatrix}$$

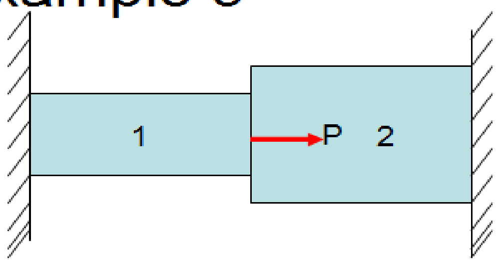
Surface force term:

$$T_e = T \quad l_e \begin{pmatrix} 1/6 \\ 1/6 \\ 2/3 \end{pmatrix}$$

This amount of body force and surface force will be distributed at three nodes as the element as 3 equally spaced nodes.

Problems on quadratic element

Example 5



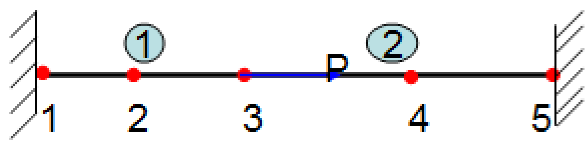
$A_1 = 600 mm^2$ $A_2 = 800 mm^2$

$E = 2 \times 10^5 \ N/mm^2$

$L_1 = 150 mm$ $L_2 = 220 mm$

P = 30 KN

Solution:



$$K_1 = 10^5 \begin{array}{c} \begin{array}{ccc} 1 & 3 & 2 \end{array} \\ \begin{pmatrix} 18.6 & 2.6 & -21.3 \\ 2.6 & 18.6 & -21.3 \\ -21.3 & -21.3 & 42.6 \end{pmatrix} \end{array} \begin{array}{c} 1 \\ 3 \\ 2 \end{array}$$

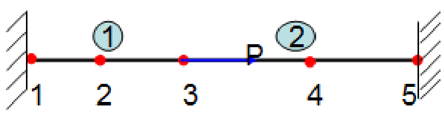
$$K_2 = 10^5 \begin{array}{c} \begin{array}{ccc} 3 & 5 & 4 \end{array} \\ \begin{pmatrix} 16.9 & 2.42 & -19.3 \\ 2.42 & 16.9 & -19.3 \\ -19.3 & -19.3 & 38.7 \end{pmatrix} \end{array} \begin{array}{c} 3 \\ 5 \\ 4 \end{array}$$

Global stiffness matrix

$$
K = 10^5 \begin{pmatrix} 18.6 & -21.3 & 2.6 & 0 & 0 \\ -21.3 & 42.6 & -21.3 & 0 & 0 \\ 2.6 & -21.3 & 35.5 & -19.3 & 2.4 \\ 0 & 0 & -19.3 & 38.7 & -19.3 \\ 0 & 0 & 2.4 & 19.3 & 16.9 \end{pmatrix}
$$

(Columns numbered 1, 2, 3, 4, 5; rows numbered 1, 2, 3, 4, 5.)

Global load vector



$$
F = \begin{pmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{pmatrix} = \begin{pmatrix} R_1 \\ 0 \\ P \\ 0 \\ R_5 \end{pmatrix}
$$

By the equilibrium equation KQ=F, solving the matrix we have Q2, Q3 and Q4 values

$$10^5 \begin{pmatrix} 18.6 & -21.3 & 2.6 & 0 & 0 \\ -21.3 & 42.6 & -21.3 & 0 & 0 \\ 2.6 & -21.3 & 35.5 & -19.3 & 2.4 \\ 0 & 0 & -19.3 & 38.1 & -19.3 \\ 0 & 0 & 2.4 & 19.3 & -16.9 \end{pmatrix} \begin{pmatrix} Q_1 \\ Q_2 \\ Q_3 \\ Q_4 \\ Q_5 \end{pmatrix} = \begin{pmatrix} R_1 \\ 0 \\ P \\ 0 \\ R_5 \end{pmatrix}$$

$Q_2 = 1.25 \times 10^{-7}$ mm

$Q_3 = 2.14 \times 10^{-3}$ mm

$Q_5 = 5.13 \times 10^{-3}$ mm

Stress components in each element

For element 1 @ node 1

$$\sigma_{1/1} = \frac{2}{l_1} \begin{pmatrix} \frac{(2\xi-1)}{2} & \frac{(2\xi+1)}{2} & -2\xi \end{pmatrix} \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} E$$

$$\sigma_{1/1} = \frac{2}{150} \begin{pmatrix} -3/2 & -\frac{1}{2} & 2 \end{pmatrix} \begin{pmatrix} 0 \\ 0.01 \\ 0.02 \end{pmatrix} 2 \times 10^5$$

$= 93.1$ N/mm^2

For element 1 @ node 2

$$\sigma_{1/2} = \frac{2}{l_1} \begin{pmatrix} \frac{(2\xi-1)}{2} & \frac{(2\xi+1)}{2} & -2\xi \end{pmatrix} \begin{pmatrix} Q1 \\ Q2 \\ Q3 \end{pmatrix} E$$

$$\sigma_{1/2} = \frac{2}{150} \begin{pmatrix} -\frac{1}{2} & \frac{1}{2} & 0 \end{pmatrix} \begin{pmatrix} 0 \\ 0.01 \\ 0.02 \end{pmatrix} 2 \times 10^5$$

$= 13.33$ N/mm^2

For element 1 @ node 3

$$\sigma_{1/3} = \frac{2}{l_1}\left[\frac{(2\xi-1)}{2} \quad \frac{(2\xi+1)}{2} \quad -2\xi\right]\begin{Bmatrix} Q1 \\ Q2 \\ Q3 \end{Bmatrix} E$$

$$\sigma_{1/3} = \frac{2}{150}\left[\tfrac{1}{2} \quad 3/2 \quad -2\right]\begin{Bmatrix} 0 \\ 0.01 \\ 0.02 \end{Bmatrix} 2 \times 10^5$$

$= -66.5\ N/mm^2$

For element 2 @ node 3

$$\sigma_{2/3} = \frac{2}{l_2}\left[\frac{(2\xi-1)}{2} \quad \frac{(2\xi+1)}{2} \quad -2\xi\right]\begin{Bmatrix} Q3 \\ Q4 \\ Q5 \end{Bmatrix} E$$

$$\sigma_{2/3} = \frac{2}{220}\left[-3/2 \quad -\tfrac{1}{2} \quad 2\right]\begin{Bmatrix} 0.02 \\ 0.01 \\ 0 \end{Bmatrix} 2 \times 10^5$$

$= -63.63\ N/mm^2$

For element 2 @ node 4

$$\sigma_{2/4} = \frac{2}{l_2}\left[\frac{(2\xi-1)}{2} \quad \frac{(2\xi+1)}{2} \quad -2\xi\right]\begin{Bmatrix} Q3 \\ Q4 \\ Q5 \end{Bmatrix} E$$

$$\sigma_{2/4} = \frac{2}{220}\left[-\tfrac{1}{2} \quad \tfrac{1}{2} \quad 0\right]\begin{Bmatrix} 0.02 \\ 0.01 \\ 0 \end{Bmatrix} 2 \times 10^5$$

$= -9.09\ N/mm^2$

For element 2 @ node 5

$$\sigma_{2/5} = \frac{2}{l_1}\begin{pmatrix}\frac{(2\xi-1)}{2} & \frac{(2\xi+1)}{2} & -2\xi\end{pmatrix}\begin{pmatrix}Q3 \\ Q4 \\ Q5\end{pmatrix} E$$

$$\sigma_{2/5} = \frac{2}{150}\begin{bmatrix}½ & 3/2 & -2\end{bmatrix}\begin{pmatrix}0.02 \\ 0.01 \\ 0\end{pmatrix} 2 \text{ X } 10^5$$

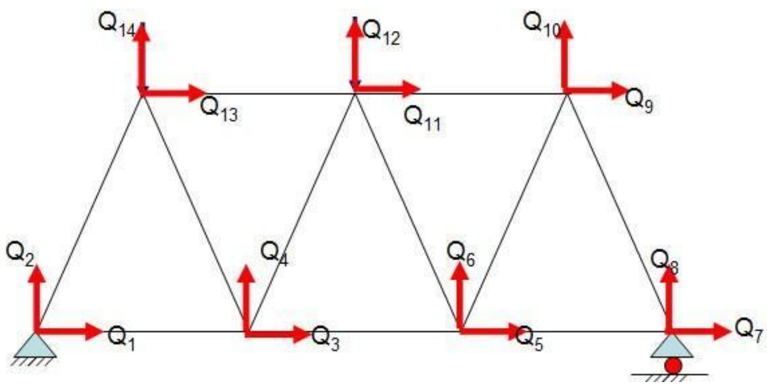
$= 45.45 \text{ N/mm}^2$

ANALYSIS OF TRUSSES

A Truss is a two force members made up of bars that are connected at the ends by joints. Every stress element is in either tension or compression. Trusses can be classified as plane truss and space truss.

- Plane truss is one where the plane of the structure remain in plane even after the application of loads
- While space truss plane will not be in a same plane

Fig shows 2d truss structure and each node has two degrees of freedom. The only difference between bar element and truss element is that in bars both local and global coordinate systems are same where in truss these are different.

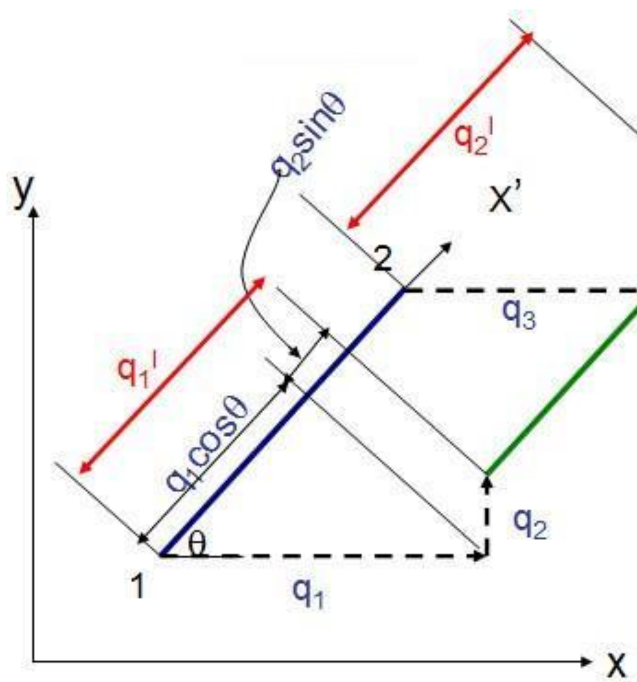


There are always assumptions associated with every finite element analysis. If all the assumptions below are all valid for a given situation, then truss element will yield an exact solution. Some of the assumptions are:

- ➢ Truss element is only a prismatic member ie cross sectional area is uniform along its length
- ➢ It should be a isotropic material
- ➢ Constant load ie load is independent of time
- ➢ Homogenous material

- A load on a truss can only be applied at the joints (nodes)
- Due to the load applied each bar of a truss is either induced with tensile/compressive forces
- The joints in a truss are assumed to be frictionless pin joints
- Self weight of the bars are neglected

Consider one truss element as shown that has nodes 1 and 2 .The coordinate system that passes along the element (x^l axis) is called local coordinate and X-Y system is called as global coordinate system. After the loads applied let the element takes new position say locally node 1 has displaced by an amount q_1^l and node2 has moved by an amount equal to q_2^l.As each node has 2 dof in global coordinate system .let node 1 has displacements q_1 and q_2 along x and y axis respectively similarly q_3 and q_4 at node 2.



Resolving the components q_1, q_2, q_3 and q_4 along the bar we get two equations as

$$q_1^l = q_1 \cos\theta + q_2 \sin\theta$$

$$q_2^l = q_3 \cos\theta + q_4 \sin\theta$$

Or

$$q_1^l = q_1 \ell + q_2 m$$

$$q_2^l = q_3 \ell + q_4 m$$

Writing the same equation into the matrix form

$$\begin{Bmatrix} q_1^l \\ q_2^l \end{Bmatrix} = \begin{pmatrix} \ell & m & 0 & 0 \\ 0 & 0 & \ell & m \end{pmatrix} \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{Bmatrix}$$

$$q^l = L\, q$$

Where L is called transformation matrix that is used for local –global correspondence.

Strain energy for a bar element we have

$$U = ½\, q^T K q$$

For a truss element we can write

$$U = ½\, q^{lT}\, K\, q^l$$

Where $q^l = L\, q$ and $q^{1T} = L^T\, q^T$

Therefore

$$U = \tfrac{1}{2}\, q^{lT}\, K\, q^{l}$$

$$= \tfrac{1}{2}\, L^{T} q^{T} K\, L q$$

$$= \tfrac{1}{2}\, q^{T} (L^{T} K\, L) q$$

$$= \tfrac{1}{2}\, q^{T} K_{T}\, q$$

Where K_T is the stiffness matrix of truss element

$$K_T = L^T K\, L$$

$$L = \begin{pmatrix} \ell & m & 0 & 0 \\ 0 & 0 & \ell & m \end{pmatrix} \qquad L^T = \begin{pmatrix} \ell & 0 \\ m & 0 \\ 0 & \ell \\ 0 & m \end{pmatrix}$$

$$K = \frac{AE}{L} \begin{pmatrix} 1 & -1 \\ -1 & 1 \end{pmatrix}$$

Taking the product of all these matrix we have stiffness matrix for truss element which is given as

$$K_T = \frac{AE}{L} \begin{pmatrix} \ell^2 & \ell m & -\ell^2 & -\ell m \\ \ell m & m^2 & -\ell m & -m^2 \\ -\ell^2 & -\ell m & \ell^2 & \ell m \\ -\ell m & -m^2 & \ell m & m^2 \end{pmatrix}$$

Stress component for truss element

The stress σ in a truss element is given by

$$\sigma = \varepsilon E$$

But strain $\varepsilon = B\, q^{l}$ and $q^{l} = T\, q$

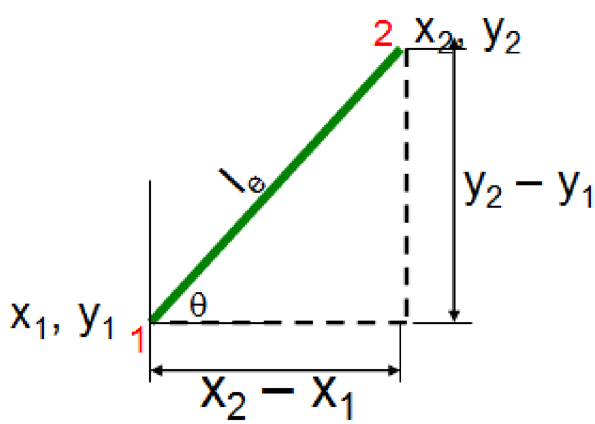
$$\text{where } B = \frac{1}{L}\begin{bmatrix} -1 & 1 \end{bmatrix}$$

Therefore

$$\sigma = \frac{E}{L_e}\begin{pmatrix} -\ell & -m & \ell & m \end{pmatrix}\begin{pmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{pmatrix}$$

How to calculate direction cosines

Consider a element that has node 1 and node 2 inclined by an angle θ as shown .let (x1, y1) be the coordinate of node 1 and (x2,y2) be the coordinates at node 2.



When orientation of an element is know we use this angle to calculate A and m as:

$$A = \cos\theta \qquad m = \cos(90 - \theta) = \sin\theta$$

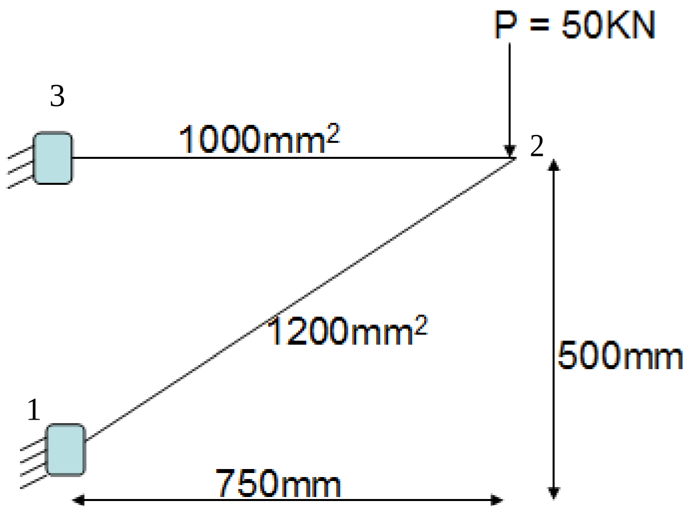
and by using nodal coordinates we can calculate using the relation

$$\ell = \frac{x_2 - x_1}{l_e} \qquad m = \frac{y_2 - y_1}{l_e}$$

We can calculate length of the element as

$$l_e = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

Example 6



Solution: For given structure if node numbering is not given we have to number them which depend on user. Each node has 2 dof say q1 q2 be the displacement at node 1, q3 & q4 be displacement at node 2, q5 &q6 at node 3.

Tabulate the following parameters as shown

Element	θ	L	$\ell = \cos\theta$	m=sin θ
1	33.6	901.3	0.832	0.554
2	0	750	1	0

For element 1 θ can be calculate by using $\tan\theta = 500/700$ ie $\theta = 33.6$, length of the element is

$$l_e = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2}$$

$= 901.3$ mm

Similarly calculate all the parameters for element 2 and tabulate

Calculate stiffness matrix for both the elements

$$K_T = \frac{AE}{L}\begin{bmatrix} \ell^2 & \ell m & -\ell^2 & -\ell m \\ \ell m & m^2 & -\ell m & -m^2 \\ -\ell^2 & -\ell m & \ell^2 & \ell m \\ -\ell m & -m^2 & \ell m & m^2 \end{bmatrix}$$

$$K_1 = 10^5 \begin{array}{c} \begin{array}{cccc} 1 & 2 & 3 & 4 \end{array} \\ \begin{bmatrix} 1.84 & 1.22 & -1.84 & -1.22 \\ 1.22 & 0.816 & -1.22 & -0.816 \\ -1.84 & -1.22 & 1.84 & 1.22 \\ -1.22 & -0.816 & 1.22 & 0.816 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 3 \\ 4 \end{array} \qquad K_2 = 10^5 \begin{array}{c} \begin{array}{cccc} 3 & 4 & 5 & 6 \end{array} \\ \begin{bmatrix} 2.66 & 0 & -2.66 & 0 \\ 0 & 0 & 0 & 0 \\ -2.66 & 0 & 2.66 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ 3 \\ 4 \\ 5 \\ 6 \end{array}$$

Element 1 has displacements q1, q2, q3, q4. Hence numbering scheme for the first stiffness matrix (K1) as 1 2 3 4 similarly for K_2 3 4 5 & 6 as shown above.

Global stiffness matrix: the structure has 3 nodes at each node 3 dof hence size of global stiffness matrix will be 3 X 2 = 6

ie 6 X 6

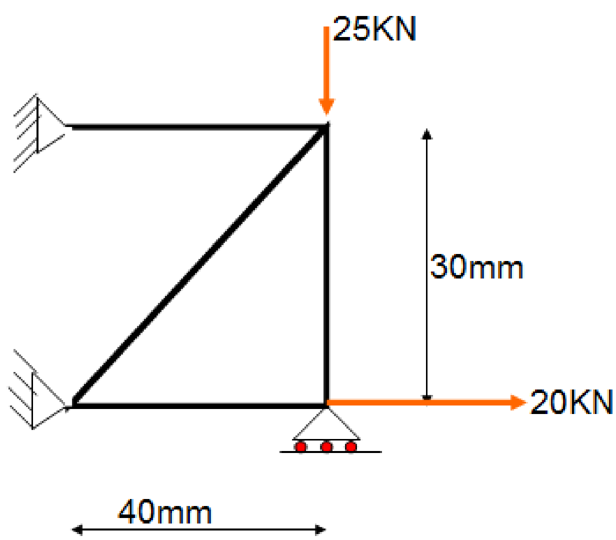
$$K = 10^5 \begin{array}{c} \begin{array}{cccccc} 1 & 2 & 3 & 4 & 5 & 6 \end{array} \\ \begin{bmatrix} 1.84 & 1.22 & -1.84 & -1.22 & 0 & 0 \\ 1.22 & 0.816 & -1.22 & -0.816 & 0 & 0 \\ -1.84 & -1.22 & 4.5 & 1.22 & -2.66 & 0 \\ -1.22 & -0.816 & 1.22 & 0.816 & 0 & 0 \\ 0 & 0 & -2.66 & 0 & 2.66 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{array}$$

From the equation KQ = F we have the following matrix. Since node 1 is fixed q1=q2=0 and also at node 3 q5 = q6 = 0 .At node 2 q3 & q4 are free hence has displacements.
In the load vector applied force is at node 2 ie F4 = 50KN rest other forces zero.

$$10^5 \begin{bmatrix} 1.84 & 1.22 & -1.84 & -1.22 & 0 & 0 \\ 1.22 & 0.816 & -1.22 & -0.816 & 0 & 0 \\ -1.84 & -1.22 & 4.5 & 1.22 & -2.66 & 0 \\ -1.22 & -0.816 & 1.22 & 0.816 & 0 & 0 \\ 0 & 0 & -2.66 & 0 & 2.66 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} Q1 \\ Q2 \\ Q3 \\ Q4 \\ Q5 \\ Q6 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ -50 \times 10^3 \\ 0 \\ 0 \end{Bmatrix}$$

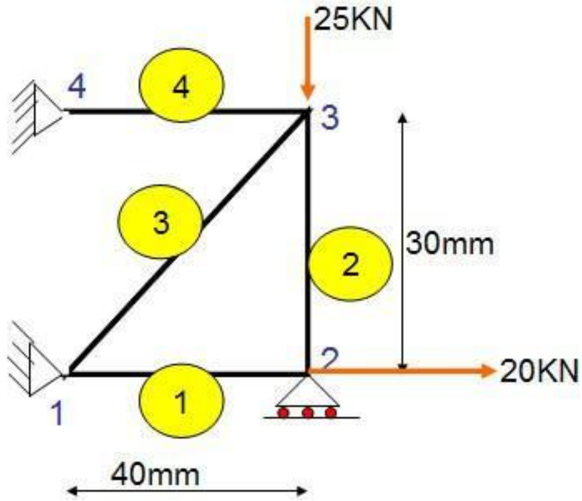
By elimination method the matrix reduces to 2 X 2 and solving we get Q3= 0.28mm and Q4 = -1.03mm. With these displacements we calculate stresses in each element.

Example 7



$E = 29.5 \times 10^6\ N/mm^2 \qquad A = 1 mm^2$

Solution: Node numbering and element numbering is followed for the given structure if not specified, as shown below



Let Q1, Q2 …..Q8 be displacements from node 1 to node 4 and
F1, F2……F8 be load vector from node 1 to node 4.

Tabulate the following parameters

Element	θ	L	$\ell=\cos\theta$	$m=\sin\theta$
1	0	40	1	0
2	90	30	0	1
3	36.8	50	0.8	0.6
4	0	40	1	0

Determine the stiffness matrix for all the elements

$$K_1=10^5\begin{bmatrix} 5 & 0 & -5 & 0 \\ 0 & 0 & 0 & 0 \\ -5 & 0 & 5 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{matrix}1\\2\\3\\4\end{matrix} \quad (\text{columns } 1\ 2\ 3\ 4)$$

$$K_2=10^5\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 6.66 & 0 & -6.66 \\ 0 & 0 & 0 & 0 \\ 0 & -6.66 & 0 & 6.66 \end{bmatrix}\begin{matrix}3\\4\\5\\6\end{matrix} \quad (\text{columns } 3\ 4\ 5\ 6)$$

$$K_3 = 10^5 \begin{array}{c} \\ \\ \\ \\ \end{array}\begin{array}{cccc} 5 & 6 & 1 & 2 \end{array}$$

$$K_3 = 10^5 \begin{pmatrix} 2.56 & 1.92 & -2.56 & -1.92 \\ 1.92 & 1.44 & -1.92 & -1.44 \\ -2.56 & -1.92 & 2.56 & 1.92 \\ -1.92 & -1.44 & 1.92 & 1.44 \end{pmatrix} \begin{matrix} 5 \\ 6 \\ 1 \\ 2 \end{matrix} \qquad K_4 = 10^5 \begin{pmatrix} 5 & 0 & -5 & 0 \\ 0 & 0 & 0 & 0 \\ -5 & 0 & 5 & 0 \\ 0 & 0 & 0 & 0 \end{pmatrix} \begin{matrix} 5 \\ 6 \\ 7 \\ 8 \end{matrix}$$

(Column labels: K_3: 5, 6, 1, 2; K_4: 5, 6, 7, 8)

Global stiffness matrix: the structure has 4 nodes at each node 3 dof hence size of global stiffness matrix will be 4 X 2 = 8

ie 8 X 8

	1	2	3	4	5	6	7	8
1	7.56	1.92	-5	0	-2.56	-1.92	0	0
2	1.92	1.44	0	0	-1.92	-1.44	0	0
3	-5	0	5	0	0	0	0	0
4	0	0	0	6.66	0	-6.66	0	0
5	-2.56	-1.92	0	0	7.56	1.92	-5	0
6	-1.92	-1.44	0	-6.66	1.92	8.11	0	0
7	0	0	0	0	-5	0	5	0
8	0	0	0	0	0	0	0	0

From the equation KQ = F we have the following matrix. Since node 1 is fixed q1=q2=0 and also at node 4 q7 = q8 = 0 .At node 2 because of roller support q3=0 & q4 is free hence has displacements. q5 and q6 also have displacement as they are free to move.

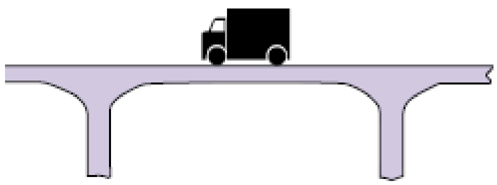
In the load vector applied force is at node 2 ie F3 = 20KN and at node 3 F6 = 25KN, rest other forces zero.

$$
10^5 \begin{pmatrix}
7.56 & 1.92 & -5 & 0 & -2.56 & -1.92 & 0 & 0 \\
1.92 & 1.44 & 0 & 0 & -1.92 & -1.44 & 0 & 0 \\
-5 & 0 & 5 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 6.66 & 0 & -6.66 & 0 & 0 \\
-2.56 & -1.92 & 0 & 0 & 7.56 & 1.92 & -5 & 0 \\
-1.92 & -1.44 & 0 & -6.66 & 1.92 & 8.11 & 0 & 0 \\
0 & 0 & 0 & 0 & -5 & 0 & 5 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{pmatrix}
\begin{pmatrix} Q1 \\ Q2 \\ Q3 \\ Q4 \\ Q5 \\ Q6 \\ Q7 \\ Q8 \end{pmatrix}
=
\begin{pmatrix} 0 \\ 0 \\ 20 \text{ X } 10^3 \\ 0 \\ 0 \\ -25 \text{ X } 10^3 \\ 0 \\ 0 \end{pmatrix}
$$

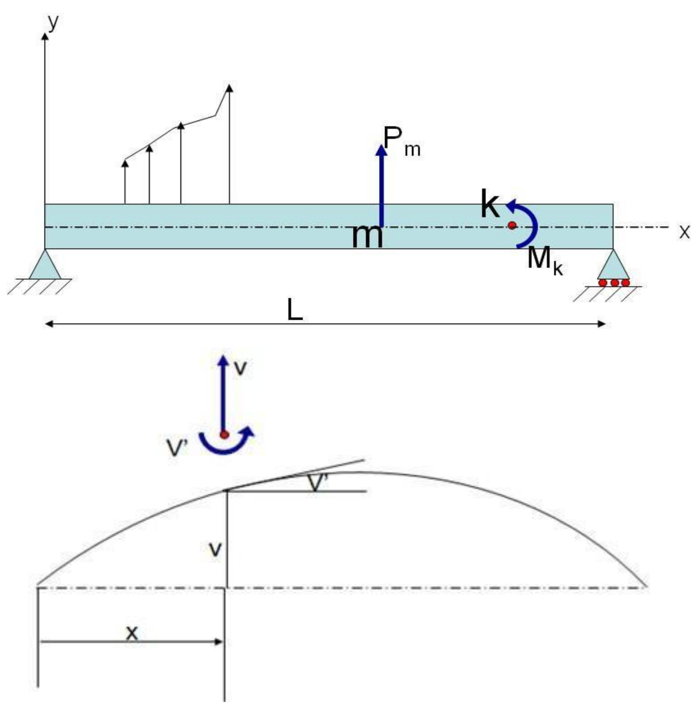
Solving the matrix gives the value of q3, q5 and q6.

Beam element

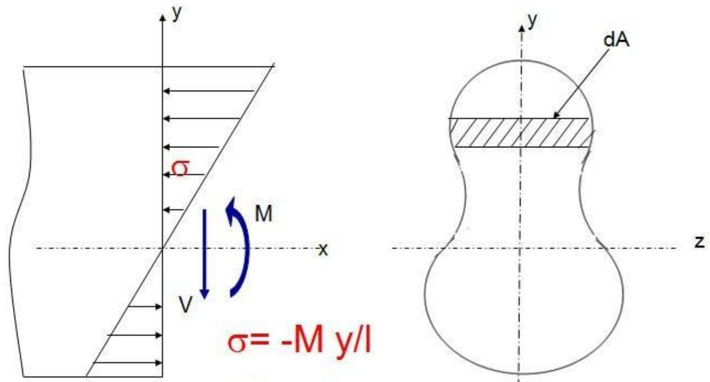
Beam is a structural member which is acted upon by a system of external loads perpendicular to axis which causes bending that is deformation of bar produced by perpendicular load as well as force couples acting in a plane. Beams are the most common type of structural component, particularly in Civil and Mechanical Engineering. A *beam* is a bar-like structural member whose primary function is to support *transverse loading* and carry it to the supports



A truss and a bar undergoes only axial deformation and it is assumed that the entire cross section undergoes the same displacement, but beam on other hand undergoes transverse deflection denoted by v. Fig shows a beam subjected to system of forces and the deformation of the neutral axis



We assume that cross section is doubly symmetric and bending take place in a plane of symmetry. From the strength of materials we observe the distribution of stress as shown.



$$\sigma = -M\,y/I$$

$$d^2v/dx^2 = M/EI$$

Where M is bending moment and I is the moment of inertia.
According to the Euler Bernoulli theory. The entire c/s has the same transverse deflection V as the neutral axis, sections originally perpendicular to neutral axis remain plane even after bending

Deflections are small & we assume that rotation of each section is the same as the slope of the deflection curve at that point (dv/dx). Now we can call beam element as simple line segment representing the neutral axis of the beam. To ensure the continuity of deformation at any point, we have to ensure that V & dv/dx are continuous by taking 2 dof @ each node V & θ(dv/dx). If no slope dof then we have only transverse dof. A prescribed value of moment load can readily taken into account with the rotational dof θ .

Potential energy approach

Strain energy in an element for a length dx is given by

$$= \tfrac{1}{2}\int_A \sigma\,\varepsilon\,dA\,dx$$

$$= \tfrac{1}{2}\int_A \sigma\,\sigma/E\,dA\,dx$$

$$= \tfrac{1}{2}\int_A \sigma^2/E\,dA\,dx$$

But we know $\sigma = M\, y / I$ substituting this in above equation we get.

$$= \frac{1}{2} \int_A \frac{M^2}{EI^2} y^2 dA\, dx$$

$$= \frac{1}{2} \frac{M^2}{EI^2} \left[\int_A y^2 dA \right] dx$$

$$= \frac{1}{2} \frac{M^2}{EI} dx$$

But

$$M = EI\, d^2v/dx^2$$

Therefore strain energy for an element is given by

$$= \frac{1}{2} \int_0^L EI\, (d^2v/dx^2)^2\, dx$$

Now the potential energy for a beam element can be written as

$$\Pi = \frac{1}{2} \int_0^L EI \left(\frac{d^2v}{dx^2} \right)^2 dx - \int_0^L p\, v\, dx - \sum_m P_m V_m - \sum_k M_k V'_k$$

P ---- distribution load per unit length

P_m----- point load @ point m

V_m----- deflection @ point m

M_k----- momentum of couple applied at point k

V'_k----- slope @ point k

Hermite shape functions:

1D linear beam element has two end nodes and at each node 2 dof which are denoted as Q_{2i-1} and Q_{2i} at node i. Here Q_{2i-1} represents transverse deflection where as Q_{2i} is slope or rotation. Consider a beam element has node 1 and 2 having dof as shown.

The shape functions of beam element are called as Hermite shape functions as they contain both nodal value and nodal slope which is satisfied by taking polynomial of cubic order

$$H_i = a_i + b_i\,\xi + c_i\,\xi^2 + d_i\,\xi^3$$

that must satisfy the following conditions

ξ	H_1	H_1'	H_2	H_2'	H_3	H_3'	H_4	H_4'
ξ = -1	1	0	0	1	0	0	0	0
ξ = 1	0	0	0	0	1	0	0	1

Applying these conditions determine values of constants as

$$H_i = a_i + b_i\,\xi + c_i\,\xi^2 + d_i\,\xi^3$$

@ node 1

$$H_1 = 1,\quad H_1' = 0\,,\quad \xi = -1$$

$$1 = a_1 - b_1 + c_1 - d_1 \longrightarrow (1)$$

$$H_1' = \frac{dH_1}{d\xi} = 0 = b_1 - 2c_1 + 3d_1 \longrightarrow (2)$$

$$H_i = a_i + b_i\,\xi + c_i\,\xi^2 + d_i\,\xi^3$$

@ node 2

$$H_1 = 1, \;\; H_1' = 0\,, \;\; \xi = 1$$

$$0 = a_1 + b_1 + c_1 + d_1 \longrightarrow (3)$$

$$H_1' = \frac{dH_1}{d\xi} = 0 = b_1 + 2c_1 + 3d_1 \longrightarrow (4)$$

Solving above 4 equations we have the values of constants

$$a_1 = \tfrac{1}{2}\,,\;\; b_1 = -\tfrac{3}{4}\,,\;\; c_1 = 0,\;\; d_1 = \tfrac{1}{4}$$

Therefore

$$H_1 = \tfrac{1}{4}\,(2 - 3\xi + \xi^3)$$

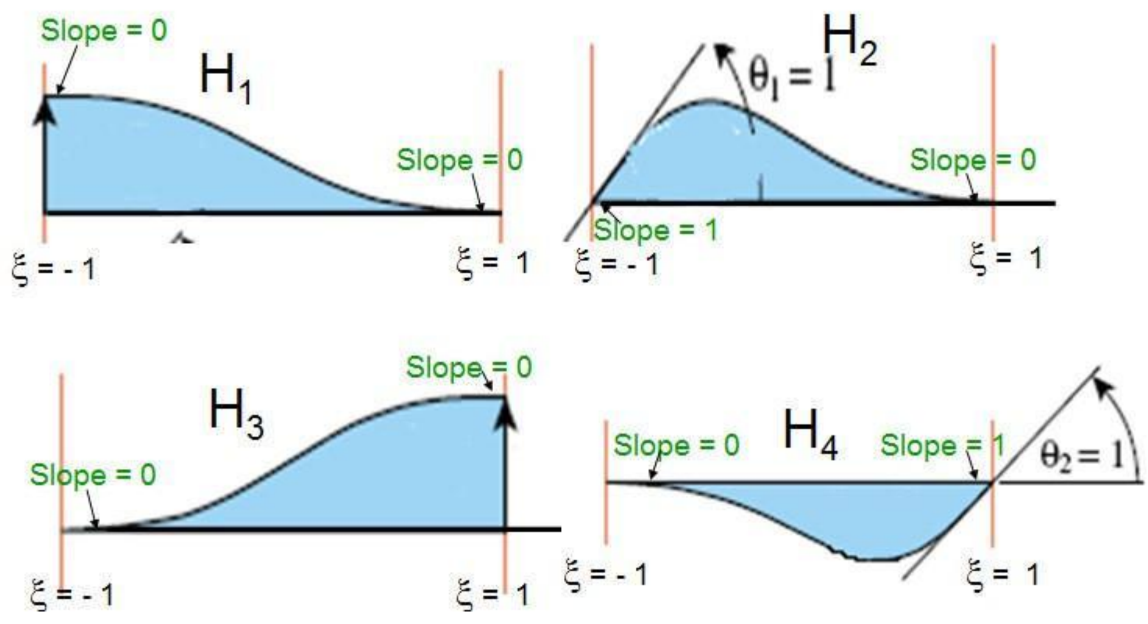
Similarly we can derive

$$H_2 = \tfrac{1}{4}\,(1 - \xi - \xi^2 + \xi^3)$$

$$H_3 = \tfrac{1}{4}\,(2 + 3\xi - \xi^3)$$

$$H_4 = \tfrac{1}{4}\,(-1 - \xi + \xi^2 + \xi^3)$$

Following graph shows the variations of Hermite shape functions



Stiffness matrix:

Once the shape functions are derived we can write the equation of the form

$$V(\xi) = H_1V_1 + H_2\left(\frac{dv}{d\xi}\right)_1 + H_3V_3 + H_4\left(\frac{dv}{d\xi}\right)_2$$

But

$$\frac{dv}{d\xi} = \frac{dv}{dx}\,\frac{dx}{d\xi}$$

$$= \frac{dv}{dx}\,\frac{Le}{2}$$

ie

$$V(\xi) = H_1V_1 + H_2\left(\frac{dv}{dx}\right)\frac{Le}{2} + H_3V_3 + H_4\left(\frac{dv}{dx}\right)\frac{Le}{2}$$

$$V(\xi) = H_1q_1 + H_2q_2\frac{L_e}{2} + H_3q_3 + H_4q_4\frac{L_e}{2}$$

We know $V = H\,q$

where

$$H = \left(H_1 \quad H_2\frac{L_e}{2} \quad H_3 \quad H_4\frac{L_e}{2}\right)$$

Strain energy in the beam element we have

$$= \tfrac{1}{2}\int_0^L EI\,(d^2v/dx^2)^2\,dx$$

$$\frac{d^2v}{dx^2} = \frac{d}{dx}\left(\frac{dv}{dx}\right)$$

$$= \frac{d}{dx}\left(\frac{2}{L_e}\frac{dv}{d\xi}\right)$$

$$= \frac{2}{L_e}\frac{d}{dx}\left(\frac{dv}{d\xi}\right)$$

$$= \frac{2}{L_e}\frac{d}{dx}(m) \qquad \text{Where } m = \frac{dv}{d\xi}$$

$$= \frac{2}{L_e}\left(\frac{2}{L_e}\frac{dm}{d\xi}\right)$$

$$\frac{d^2v}{dx^2} = \frac{4}{L_e^2}\left(\frac{d^2v}{d\xi^2}\right) \qquad \left(\frac{d^2v}{dx^2}\right)^2 = \frac{16}{L_e^4}\left(\frac{d^2v}{d\xi^2}\right)^2$$

$$V = H\,q$$

$$\left(\frac{d^2v}{dx^2}\right)^2 = q^T \frac{16}{L_e^4}\left(\frac{d^2H}{d\xi^2}\right)^T\left(\frac{d^2H}{d\xi^2}\right) q$$

Where

$$\left(\frac{d^2H}{d\xi^2}\right) = \left[\frac{3\xi}{2},\ \left(\frac{-1+3\xi}{2}\right)\frac{l_e}{2},\ \frac{-3\xi}{2},\ \left(\frac{1+3\xi}{2}\right)\frac{l_e}{2}\right]$$

Therefore total strain energy in a beam is

$$= \tfrac{1}{2}\int_e EI\,(d^2v/dx^2)^2\,dx$$

$$= \tfrac{1}{2}\int_e EI\,(d^2v/dx^2)^2\,l_e/2\ d\xi$$

$$= \frac{EI}{2}\frac{L_e}{2}\int_e q^T \frac{16}{L_e^4}\left(\frac{d^2H}{d\xi^2}\right)^T\left(\frac{d^2H}{d\xi^2}\right) q\ d\xi$$

$$= \tfrac{1}{2}\,q^T\left\{\frac{8EI}{L_e^3}\int_{-1}^{+1}\left(\frac{d^2H}{d\xi^2}\right)^T\left(\frac{d^2H}{d\xi^2}\right)\right\} q\,d\xi$$

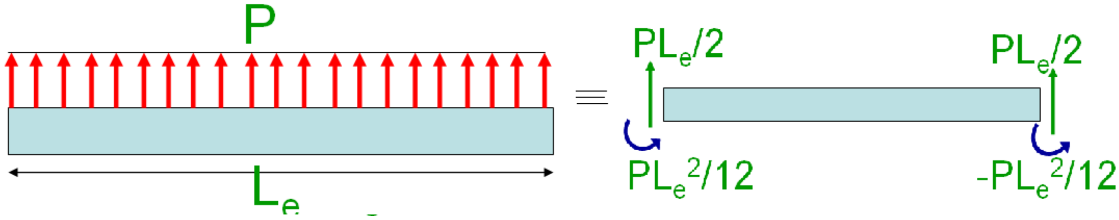
$$= \tfrac{1}{2}\,q^T K\,q$$

Now taking the K component and integrating for limits -1 to +1 we get

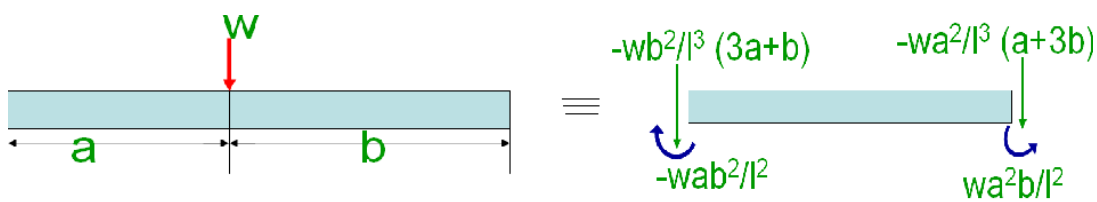
$$K = \frac{EI}{Le^3}\begin{pmatrix} 12 & 6l_e & -12 & 6l_e \\ 6l_e & 4l_e^2 & -6l_e & 2l_e^2 \\ -12 & -6l_e & 12 & -6l_e \\ 6l_e & 2l_e^2 & -6l_e & 4l_e^2 \end{pmatrix}$$

Beam element forces with its equivalent loads

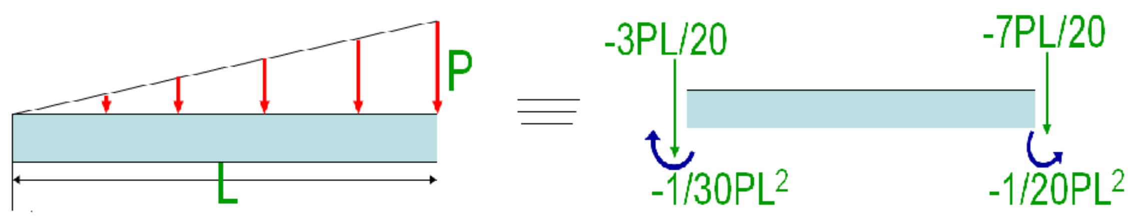
Uniformly distributed load



Point load on the element



Varying load



Bending moment and shear force

We know

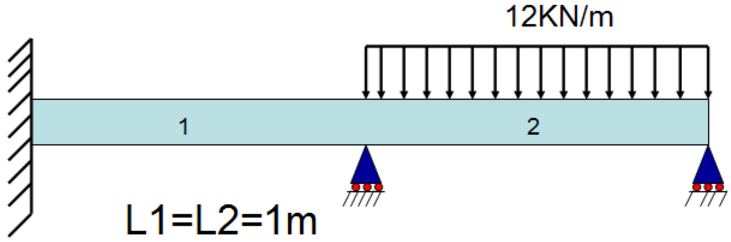
$$M = EI\left(\frac{d^2v}{dx^2}\right) \qquad V = \left(\frac{dM}{dx}\right) \qquad V = Hq$$

Using these relations we have

$$M = \frac{EI}{l_e^2}\left[6\xi q_1 + (3\xi - 1)l_e q_2 - 6\xi q_3 + (3\xi + 1)l_e q_4\right]$$

$$V = \frac{6EI}{l_e^3}\left(2q_1 + l_e q_2 - 2q_3 + l_e q_4\right)$$

Example 8



L1=L2=1m

E = 200GPa

I = 4 X 10^6N/mm^4

Solution:

Let's model the given system as 2 elements 3 nodes finite element model each node having 2 dof. For each element determine stiffness matrix.

$$K_1 = 8 \text{ X } 10^5 \begin{matrix} \begin{matrix} 1 & 2 & 3 & 4 \end{matrix} \\ \begin{pmatrix} 12 & 6 & -12 & 6 \\ 6 & 4 & -6 & 2 \\ -12 & -6 & 12 & -6 \\ 6 & 4 & -6 & 4 \end{pmatrix} \end{matrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \qquad K_2 = 8 \text{ X } 10^5 \begin{matrix} \begin{matrix} 3 & 4 & 5 & 6 \end{matrix} \\ \begin{pmatrix} 12 & 6 & -12 & 6 \\ 6 & 4 & -6 & 2 \\ -12 & -6 & 12 & -6 \\ 6 & 4 & -6 & 4 \end{pmatrix} \end{matrix} \begin{matrix} 3 \\ 4 \\ 5 \\ 6 \end{matrix}$$

Global stiffness matrix

$$K = 8 \text{ X } 10^5 \begin{matrix} \begin{matrix} 1 & 2 & 3 & 4 & 5 & 6 \end{matrix} \\ \begin{pmatrix} 12 & 6 & -12 & 6 & 0 & 0 \\ 6 & 4 & -6 & 2 & 0 & 0 \\ -12 & -6 & 24 & 0 & -12 & 6 \\ 6 & 2 & 0 & 8 & -6 & 2 \\ 0 & 0 & -12 & -6 & 12 & -6 \\ 0 & 0 & 6 & 2 & -6 & 4 \end{pmatrix} \end{matrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{matrix}$$

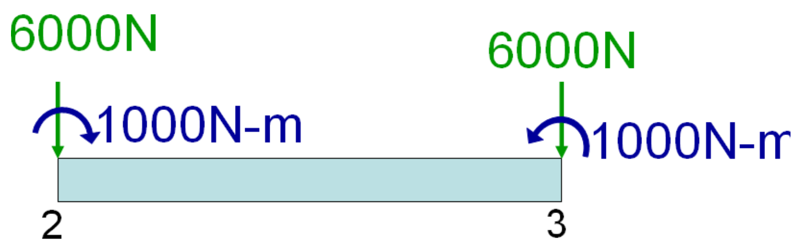
Load vector because of UDL

Element 1 do not contain any UDL hence all the force term for element 1 will be zero.

ie

$$F_1 = \begin{pmatrix} F1 \\ F2 \\ F3 \\ F4 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

For element 2 that has UDL its equivalent load and moment are represented as



ie

$$F_2 = \begin{pmatrix} F3 \\ F4 \\ F5 \\ F6 \end{pmatrix} = \begin{pmatrix} -6000 \\ -1000 \\ -6000 \\ 1000 \end{pmatrix}$$

Global load vector:

$$F = \begin{pmatrix} F1 \\ F2 \\ F3 \\ F4 \\ F5 \\ F6 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ -6000 \\ -1000 \\ -6000 \\ 1000 \end{pmatrix}$$

From KQ=F we write

$$8 \times 10^5 \begin{pmatrix} 12 & 6 & -12 & 6 & 0 & 0 \\ 6 & 4 & -6 & 2 & 0 & 0 \\ -12 & -6 & 24 & 0 & -12 & 6 \\ 6 & 2 & 0 & 8 & -6 & 2 \\ 0 & 0 & -12 & -6 & 12 & -6 \\ 0 & 0 & 6 & 2 & -6 & 4 \end{pmatrix} \begin{pmatrix} Q1 \\ Q2 \\ Q3 \\ Q4 \\ Q5 \\ Q6 \end{pmatrix} = \begin{pmatrix} 0 \\ 0 \\ 6000 \\ -1000 \\ -6000 \\ 1000 \end{pmatrix}$$

At node 1 since its fixed both q1=q2=0

node 2 because of roller q3=0

node 3 again roller ie q5= 0

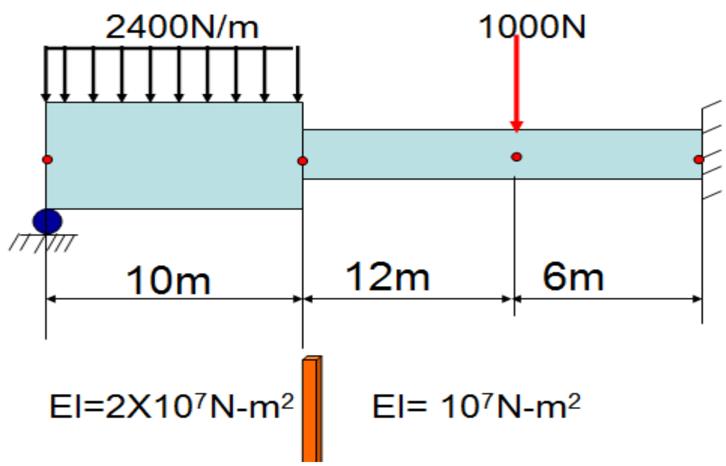
By elimination method the matrix reduces to 2 X 2 solving this we have Q4= -2.679×10^{-4}mm and Q6 = 4.464×10^{-4}mm

To determine the deflection at the middle of element 2 we can write the displacement function as

$$V(\xi) = H_1 q_3 + H_2 q_4 \frac{L_e}{2} + H_3 q_5 + H_4 q_6 \frac{L_e}{2}$$

= -0.089mm

Example 9



Solution: Let's model the given system as 3 elements 4 nodes finite element model each node having 2 dof. For each element determine stiffness matrix. Q1, Q2……Q8 be nodal displacements for the entire system and F1……F8 be nodal forces.

$$K_1 = \frac{2X\,10^7}{10^3}\begin{bmatrix} 12 & 60 & -12 & 60 \\ 60 & 400 & -60 & 200 \\ -12 & -60 & 12 & -60 \\ 60 & 200 & -60 & 400 \end{bmatrix} \quad \text{(rows/columns: 1, 2, 3, 4)}$$

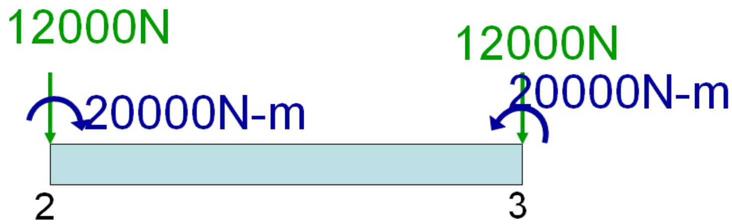
$$K_2 = \frac{10^7}{12^3}\begin{bmatrix} 12 & 72 & -12 & 72 \\ 72 & 576 & -72 & 288 \\ -12 & -72 & 12 & -72 \\ 72 & 288 & -72 & 576 \end{bmatrix} \quad \text{(rows/columns: 3, 4, 5, 6)}$$

$$K_3 = \frac{10^7}{6^3}\begin{bmatrix} 12 & 36 & -12 & 36 \\ 36 & 14 & -36 & 72 \\ -12 & -36 & 12 & -36 \\ 36 & 72 & -36 & 144 \end{bmatrix} \quad \text{(rows/columns: 5, 6, 7, 8)}$$

Global stiffness matrix:

$$K = \begin{bmatrix} & & \\ & 8 \times 8 & \\ & & \end{bmatrix} \quad \text{(rows/columns: 1, 2, 3, 4, 5, 6, 7, 8)}$$

Load vector because of UDL:
For element 1 that is subjected to UDL we have load vector as



ie

$$F_1 = \begin{Bmatrix} F1 \\ F2 \\ F3 \\ F4 \end{Bmatrix} = \begin{Bmatrix} -12000 \\ -20000 \\ -12000 \\ 20000 \end{Bmatrix}$$

Element 2 and 3 does not contain UDL hence

$$F_2 = \begin{Bmatrix} F3 \\ F4 \\ F5 \\ F6 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \qquad F_3 = \begin{Bmatrix} F5 \\ F6 \\ F7 \\ F8 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

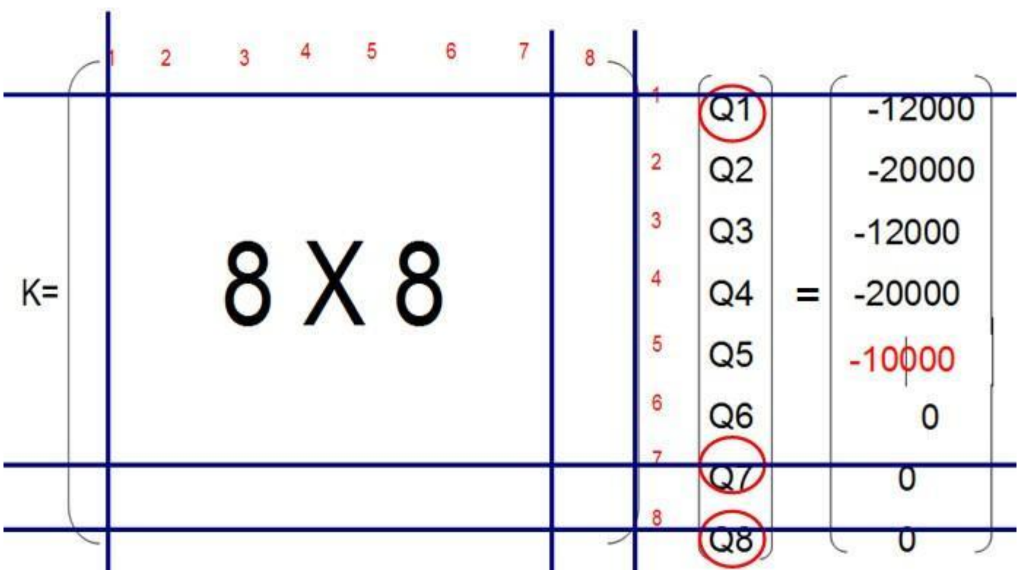
Global load vector:

$$F = \begin{Bmatrix} F1 \\ F2 \\ F3 \\ F4 \\ F5 \\ F6 \\ F7 \\ F8 \end{Bmatrix} = \begin{Bmatrix} -12000 \\ -20000 \\ -12000 \\ -20000 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

And also we have external point load applied at node 3, it gets added to F5 term with negative sign since it is acting downwards. Now F becomes,

$$F=\begin{pmatrix} F1 \\ F2 \\ F3 \\ F4 \\ F5 \\ F6 \\ F7 \\ F8 \end{pmatrix} = \begin{pmatrix} -12000 \\ -20000 \\ -12000 \\ -20000 \\ 0 \quad -10000 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

From KQ=F



At node 1 because of roller support q1=0

Node 4 since fixed q7=q8=0

After applying elimination and solving the matrix we determine the values of q2, q3, q4, q5 and q6.

UNIT-III

Two Dimensional Analysis

Many engineering structures and mechanical components are subjected to loading in two directions. Shafts, gears, couplings, mechanical joints, plates, bearings, are few examples. Analysis of many three dimensional systems reduces to two dimensional, based on whether the loading is plane stress or plane strain type. Triangular elements or Quadrilateral elements are used in the analysis of such components and systems. The various load vectors, displacement vectors, stress vectors and strain vectors used in the analysis are as written below,

the displacement vector **u** = $[u, v]^T$,
u is the displacement along x direction, v is the displacement along y direction,

the body force vector **f** = $[f_x , f_y]^T$
f_x , is the component of body force along x direction, f_y is the component of body force along y direction

the traction force vector **T** = $[T_x , T_y]^T$
T_x , is the component of body force along x direction, T_y is the component of body force along y direction

Two dimensional stress strain equations

From theory of elasticity for a two dimensional body subjected to general loading the equations of equilibrium are given by

$[6o_x /6x] + [6ı_{yx} /6 y] + F_x = 0$

$[6ı_{xy} /6x] + [6o_y /6y] + F_y = 0$

Also $ı_{xy} = ı_{yx}$

The strain displacement relations are given by

$s_x = 6u /6x$, $s_y = 6v /6y$, $y_{xy} = 6u /6y + 6v /6x$

$s = [6u /6x, 6v /6y, (6u /6y + 6v /6x)]^T$

The stress strain relationship for plane stress and plane strain conditions are given by the matrices shown in the next page. o_x o_y $ı_{xy}$ s_x s_y y_{xy} are usual stress strain components, v is the poisons ratio. E is young's modulus. Please note the differences in [D] matrix .

Two dimensional elements

Triangular elements and **Quadrilateral elements** are called two dimensional elements. A simple triangular element has straight edges and corner nodes. This is also a linear element. It can have constant thickness or variable thickness.

The stress strain relationship for plane stress loading is given by

o_x			1	V	0		s_z
o_y	=	E / (1-v ^{2})	V	1	0	*	s_y
$ı_{xy}$			0	0	1-v / 2		y_{yz}

[**O**] = [**D**] [S]

The stress strain relationship for plane strain loading is give by

o_x			1-v	V	0		s_z
o_y	=	E / (1+v)(1-2v)	V	1-v	0	*	s_y
$ı_{xy}$			0	0	½ -v		y_{yz}

[**O**] = [**D**] [S]

The element having mid side nodes along with corner nodes is a higher order element. Element having curved sides is also a higher order element.

A simple quadrilateral element has straight edges and corner nodes. This is also a linear element. It can have constant thickness or variable thickness. The quadrilateral having mid side nodes along with corner nodes is a higher order element. Element having curved sides is also a higher order element.

The given two dimensional component is divided in to number of triangular elements or quadrilateral elements. If the component has curved boundaries certain small region at the boundary is left

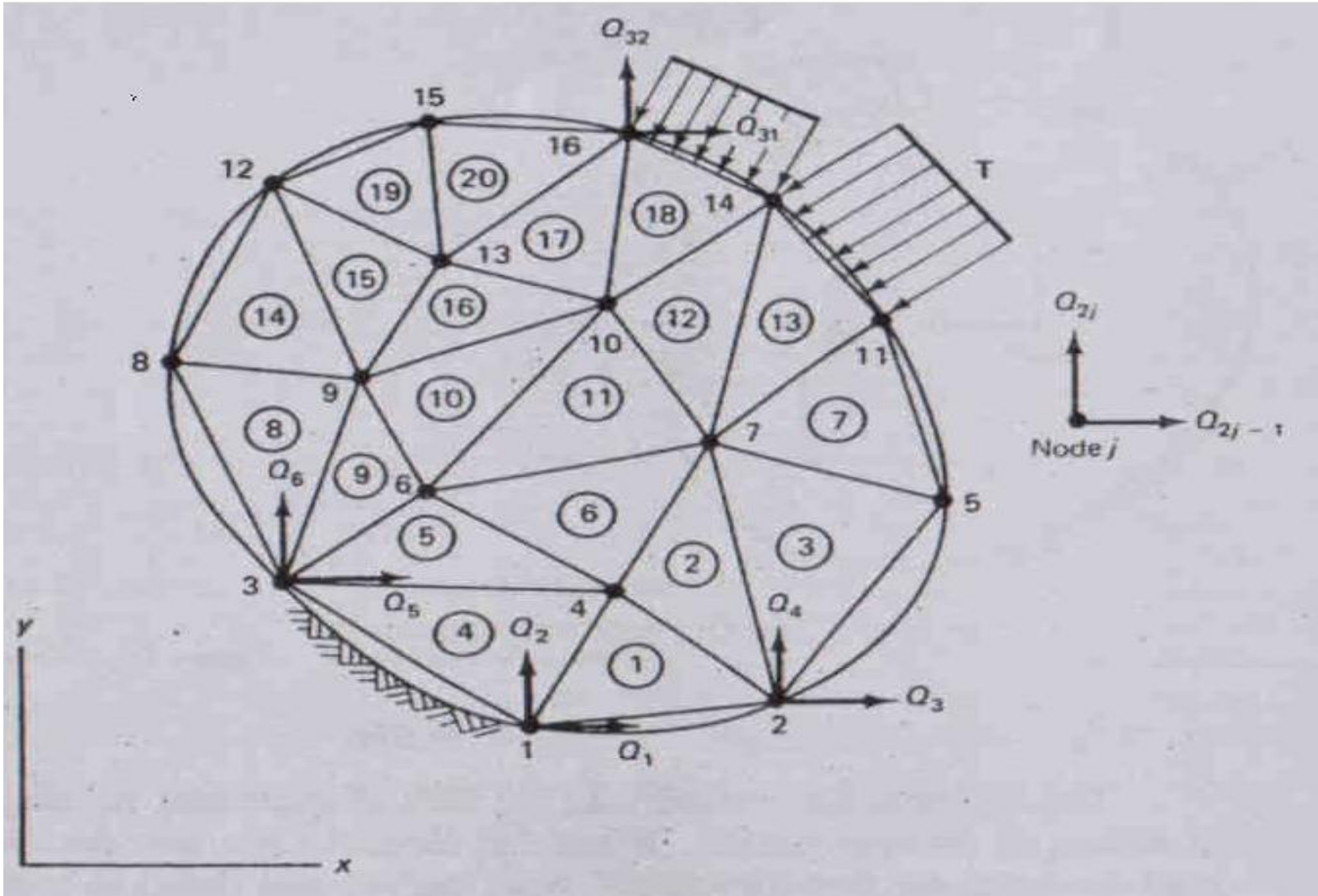
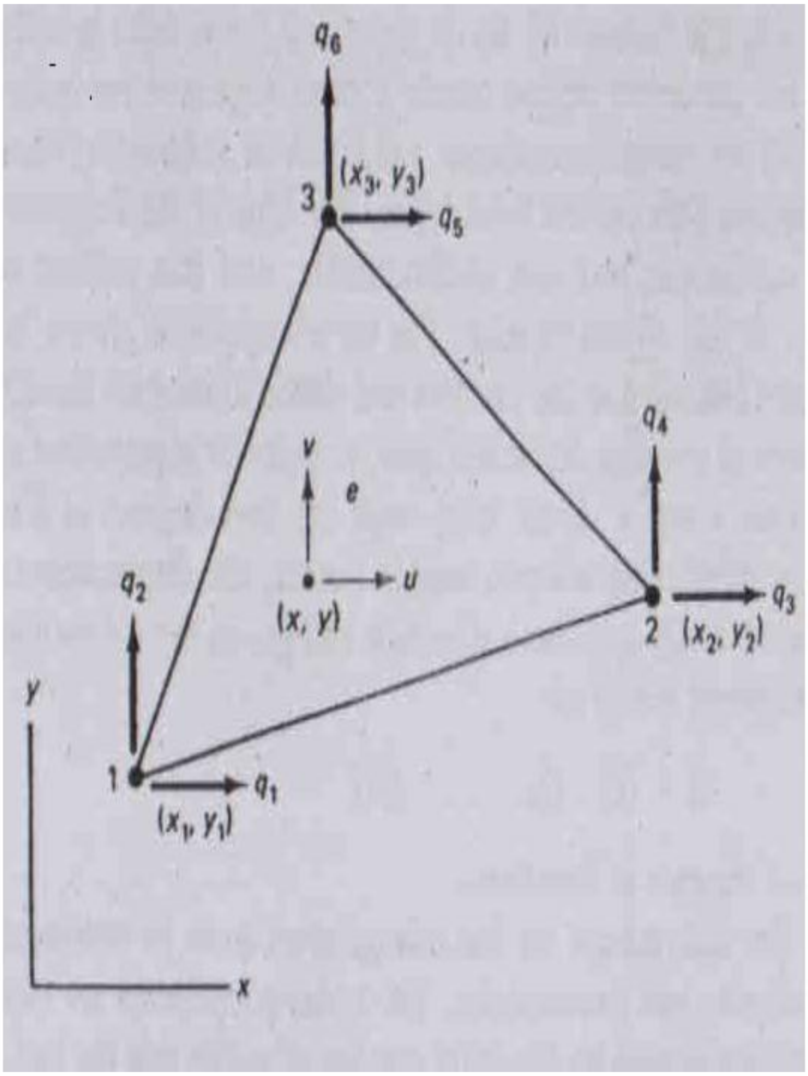
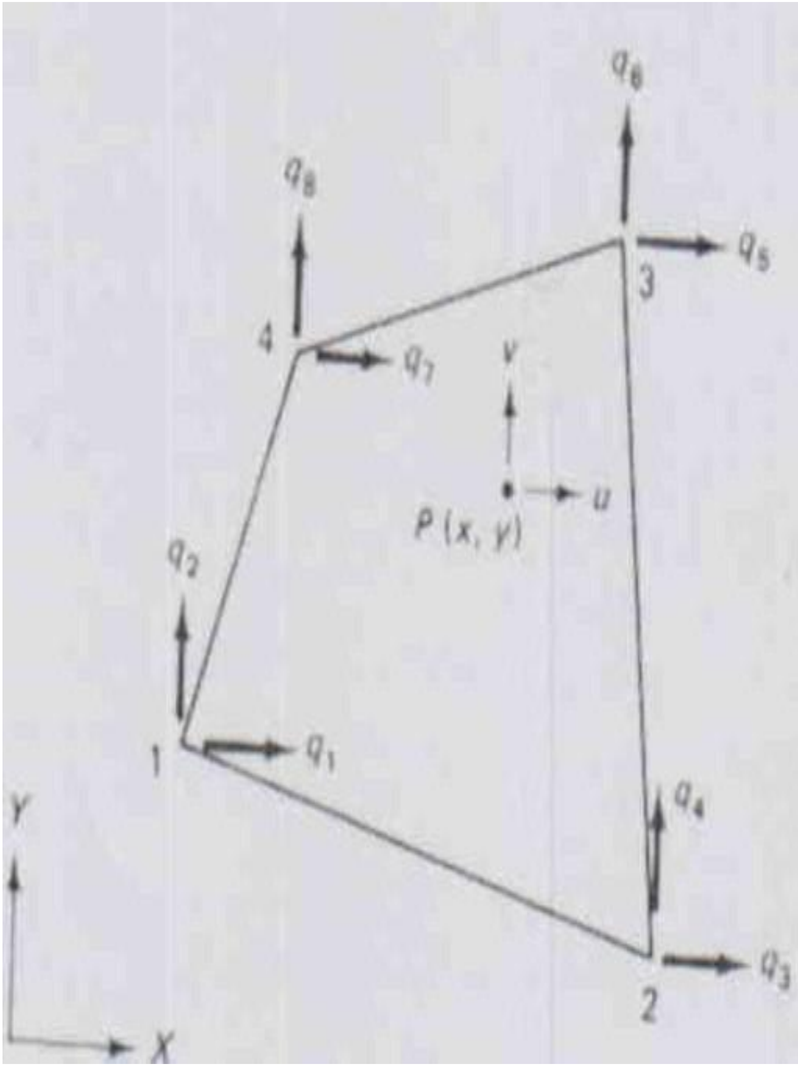


Figure 5.2 Finite element discretization.



Constant Strain Triangle



Quadrilateral

Constant Strain Triangle

It is a triangular element having three straight sides joined at three corners. and imagined to have a node at each corner. Thus it has three nodes, and each node is permitted to displace in the two directions, along x and y of the Cartesian coordinate system. The loads are applied at nodes. Direction of load will also be along x direction and y direction, +ve or –ve etc. Each node is said to have two degrees of freedom. The nodal displacement vector for each element is given by,

$\mathbf{q} = [q_1, q_2, q_3, q_4, q_5, q_6]$

q_1, q_3, q_5 are nodal displacements along x direction of node1, node2 and node3 simply called horizontal displacement components.

q_2, q_4, q_6 are nodal displacements along y direction of node1, node2 and node3 simply called vertical displacement components. q_{2j-1} is the displacement component in x direction and q_{2j} is the displacement component in y direction.

Similarly the nodal load vector has to be considered for each element.

Poi

nt loads will be acting at various nodes along x and y

…………………………

(x_1, y_1), (x_2, y_2), (x_3, y_3) are cartesian coordinates.of node 1 node 2 and node 3.

In the discretized model of the continuum the node numbers are progressive, like 1,2,3,4,5,6,7,8………etc and the corresponding displacements are Q_1, Q_2, Q_3,

Q_4, Q_5 Q_6, Q_7, Q_8, Q_9, Q_{10}….. Q_{16}, two displacement components at each node.

Q_{2j-1} is the displacement component in x direction and Q_{2j} is the displacement component in y direction. Let j = 10, ie

10^{th} node, $Q_{2j-1} = Q_{19}$ $Q_{2j} = Q_{20}$

The element connectivity table shown establishes correspondence of local and global node numbers and the corresponding degrees of freedom. Also the (x_1, y_1), (x_2, y_2) and (x_3, y_3) have the global correspondence established through the table.

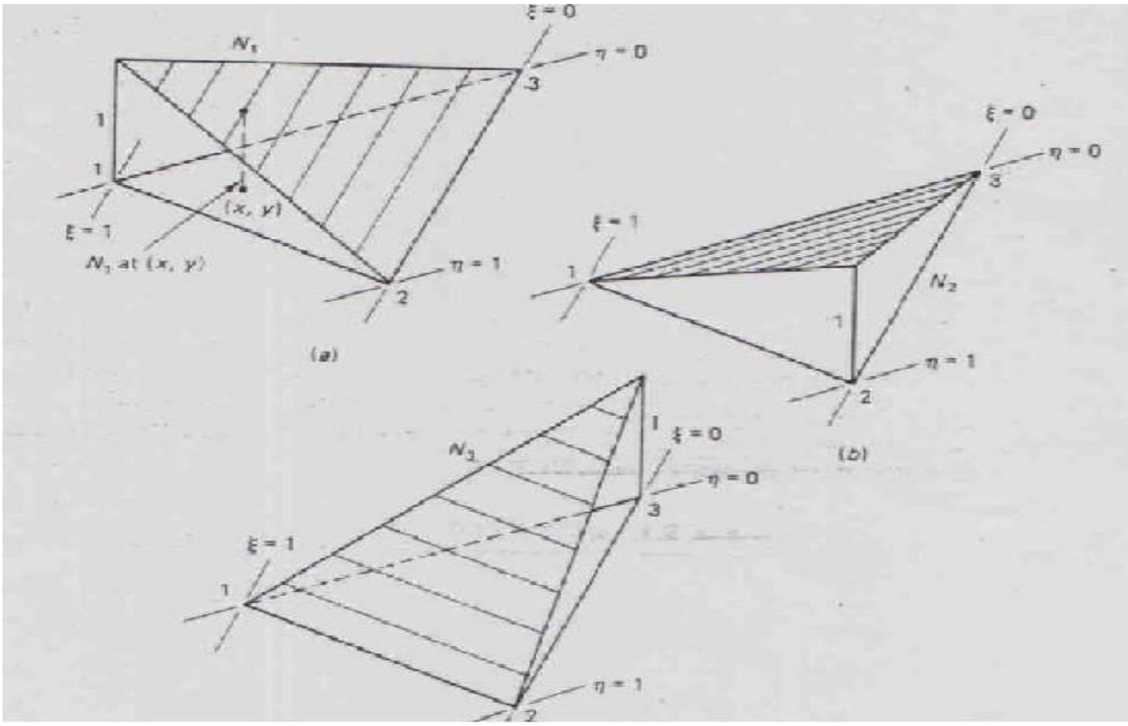
Element Connectivity Table Showing Local – Global Node Numbers				
Element Number	Local Nodes Numbers 1	2	3	
1	1	2	4	Corres-
2	4	2	7	
3				-ponding-
..	..	..	..	Global-
11	6	7	10	Node-
..	..	..	..	
20	13	16	15	Numbers

Nodal Shape Functions: under the action of the given load the nodes are assumed to deform linearly. element has to deform elastically and the deformation has to become zero as soon as the loads are zero. It is required to define the magnitude of deformation

and nature of deformation for the element Shape functions or Interpolation functions are used to model the magnitude of displacement and nature of displacement.

The Triangular element has three nodes. Three shape functions N1 , N2 , N3 are used at nodes 1,2 and 3 to define the displacements. Any linear combination of these shape functions also represents a plane surface.
N1 = Ç , N2 = y , N3 = 1 – Ç – y (1.8)

The value of N1 is unity at node 1 and linearly reduces to 0 at node 2 and 3. It defines a plane surface as shown in the shaded fig. N2 and N3 are represented by similar surfaces having values of unity at nodes 2 and 3 respectively and dropping to 0 at the opposite edges. In particular N1 + N2 + N3 represents a plane at a height of 1 at nodes 1 , 2 and 3 The plane is thus parallel to triangle 1 2 3.



Shape Functions N_1 , N_2 , N_3

For every N1 , N2 and N3 , N1 + N2 + N3 = 1 N1 N2 andN3 are therefore not linearly independent.

N1 = Ç N2 = y N3 = 1 – Ç – y, where Ç and y are natural coordinates The displacements inside the element are given by,

u = N1 q1 + N2 q3 + N3 q5

v = N1 q2 + N2 q4 + N3 q6 writing these in the matrix form

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} * \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \end{Bmatrix}$$

$[u] = [N][q]$

Iso Paramatric Formulation :

The shape functions N1, N2, N3 are also used to define the geometry of the element apart from variations of displacement. This is called Iso-Parametric formulation

- u = N1 q1 + N2 q3 + N3 q5

v = N1 q2 + N2 q4 + N3 q6 , defining variation of displacement.

- x = N1 x1 + N2 x2 + N3 x3
- y = N1 y1 + N2 y2 + N3 y3 , defining geometry.

Potential Energy :

Total Potential Energy of an Elastic body subjected to general loading is given by

n = Elastic Strain Energy + Work Potential

$n = ½ \int o^T s\ dv - \int u^T f\ dv - \int u^T T\ ds - \Sigma u^T i\ Pi$

For the 2- D body under consideration P.E. is given by

$v = ½ \int s^T \mathbf{D}\ s\ \mathbf{te\ dA} - \int \mathbf{u^T\ f\ t\ dA} - \int \mathbf{u^T\ T\ t\ dl} - Z\mathbf{u^T i\ Pi}$

This expression is utilised in deriving the elemental properties such as Element stiffness matrix **[K] ,** load vetors **f^e , T^e ,** etc .

Derivation of Strain Displacement Equation and Stiffness Matrix for CST (derivation of [B] and [K]) :

Consider the equations

u = N1 q1 + N2 q3 + N3 q5v = N1 q2 + N2 q4 + N3 q6

x = N1 x1 + N2 x2 + N3 x3y = N1 y1 + N2 y2 + N3 y3 Eq (1)

We Know that u and v are functions of x and y and they in turn are functions of Ç and y .

u = u (x (Ç ,y) , y (Ç ,y)) v = v (x (Ç ,y) , y (Ç ,y))

taking partial derivatives for u , using chain rule, we have equation (A) given by

$$\frac{\partial u}{\partial \xi} = \frac{\partial u}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial u}{\partial y}\frac{\partial y}{\partial \xi}$$

$$\frac{\partial u}{\partial \eta} = \frac{\partial u}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial u}{\partial y}\frac{\partial y}{\partial \eta}$$

Eq (A)

Similarly, taking partial derivatives for v using chain rule, we have equation (B) given by

$$\frac{\partial v}{\partial \xi} = \frac{\partial v}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial v}{\partial y}\frac{\partial y}{\partial \xi}$$

$$\frac{\partial v}{\partial \eta} = \frac{\partial v}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial v}{\partial y}\frac{\partial y}{\partial \eta}$$

Eq (B)

now consider equation (A), writing it in matrix form

$$\begin{bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \end{bmatrix}$$

$$\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}$$ Is called **JACOBIAN [J]**

Jacobian is used in determining the strain components, now we can get

$$\begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \end{Bmatrix} = [\,J\,]^{-1} \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{Bmatrix}$$

In the Left vector 6u /6x = s_x , is the strain component along x-dirction.

Similarly writing equation (B) in matrix form and considering [J] we get ,

$$\begin{Bmatrix} \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial y} \end{Bmatrix} = [\,J\,]^{-1} \begin{Bmatrix} \frac{\partial v}{\partial \xi} \\ \frac{\partial v}{\partial \eta} \end{Bmatrix}$$

In the left vector 6v /6y = s_y , is the strain component along y-direction..

6u/6x = s_x , 6v/6y = s_y , y_{xy} = 6u/6y + 6v/6x

We have to determine [J] , $[J]^{-1}$ which is same for both the equations.

First we will take up the determination 6u/6x = s_x and 6u/6y using J and J^{-1} , Consider the equations

u = N1 q1 + N2 q3 + N3 q5 v = N1 q2 + N2 q4 + N3 q6

Substituting for N1 , N2 and N3 , in the above equations we get

u = Ç q1 + y q3 + (1 – Ç – y) q5 = (q1 - q5) Ç + (q3 - q5) y + q5

= q_{15} Ç + q_{35} y + q_5

6u /6Ç = q15 6u /6y = q 35

v = Ç q2 + y q4 + (1 – Ç – y) q6 = (q2 - q6) Ç + (q4 - q6) y + q6

= q_{26} Ç + q_{46} y + q_6

6v /6Ç = q 26 6v /6y = q 46

Consider x = N1 x1 + N2 x2 +

N3 x3 y = N1 y1 + N2

y2 + N3 y3

Substituting for N1 , N2 and N3 , in the above equations we get

x = Ç x1 + y x2 + (1 – Ç – y) x3

x = (x1 - x3) Ç + (x2 - x3) y + x3 = x_{13} Ç + x_{23} y + x3

6x /6Ç = x13 6x /6y = q 23

y = Ç y1 + y y2 + (1 – Ç – y) y3

y = (y1 - y3) Ç + (y2 - y3) y + y3 = y_{13} Ç + y_{23} y + y3

6y /6Ç = y13 6y /6y = y23

To determine [J] , [J] $^{-1}$

6u /6Ç = q15 6u /6y = q 35 6v /6Ç = q 26 6v /6y = q 46

6x /6Ç = x13 6x /6y = y236y /6Ç = y13 6y /6y = y23

[J] = 6x /6Ç 6y /6Ç [J] = x13 , y13 x1 - x 3 , y1 - y 3

6x /6y 6y /6y x23 , y23 x2 - x 3 , y2 - y3

To determine [J] $^{-1}$: find out co

factors [J] co-factors of x ij = (-1)

$^{i+j}$ | |

co-factors [co] = (y2 - y3), -(x2 - x 3) y23 , x32
-(y1 - y3), (x1 - x 3) y31, x13

Adj [J] = $[co]^T$ = y23 y31
x32 x13

$[J]^{-1}$ = Adj [J] / | J |

$[J]^{-1}$ = (1/ |J|) y23 y31
x32 x13

Also we have

6u /6Ç = q15 = q1- q5 6u /6y = q 35 = q3 –q5

6u /6x = $[J]^{-1}$ 6u /6Ç
6u /6y 6u /6y

6u /6x = (1/ |J|) y23 y31 q1- q5
6u /6y x32 x13 q3 –q5

6u /6x = (1/ |J|) y23 q1- q5 + y31 q3 –q5
6u /6y x32 q1- q5 + x13 q3 –q5

6u /6x = (1/ |J|) y23 q1- y23 q5 + y31 q3 – y31q5
6u /6y x32 q1- x32 q5 + x13 q3 – x13q5

6u /6x = (1/ |J|) y23q1 + y31 q3 - y23 q5– y31q5
6u /6y x32 q1 + x13 q3 - x32 q5– x13q5

6u /6x = (1/ |J|) y23q1 + y31 q3 - q5 (y2 - y3+ y3 -y1)
6u /6y x32 q1 + x13 q3 - q5 (x3 - x2+x1 - x3)

6u /6x = (1/ |J|) y23q1 + y31 q3 - q5 (y2 -y1)
6u /6y x32 q1 + x13 q3 - q5 (- x2+x1)

6u /6x = (1/ |J|) y23q1 + y31 q3 + q5 (y1- y2)
6u /6y x32 q1 + x13 q3 + q5 (x2 -x1)

6u /6x = (1/ |J|) y23q1 + y31 q3 + y12 q5
6u /6y x32 q1 + x13 q3 + x21 q5

Writing the R.H.S of above equation in Matrix form

6u /6x = 1/ |J| y23 0 y31 0
y12 0 q1 6u /6y x32 0
x13 0 x21 0 q2 q3
q4q5
q6…… eq (6)

Similarly Considering equation (B) we get

$$\begin{Bmatrix} \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial y} \end{Bmatrix} = [\,\mathbf{J}\,]^{-1} \begin{Bmatrix} \frac{\partial v}{\partial \xi} \\ \frac{\partial v}{\partial \eta} \end{Bmatrix}$$

[J] = 6x /6Ç 6y /6Ç = x13 , y13 x1 - x 3 , y1 - y 3
6x /6y 6y /6y x23 , y23 x2 - x 3 , y2 - y3

[J] -1 = 1/ |J| y23 y31
x32 x13

consider v = N1 q2 + N2 q4 + N3 q6 v = Ç q2 + y q4

+ (1 Ç– y) q6
v = (q2 - q6) Ç + (q4 - q6) y + q6
= q26 Ç + q46 y + q6

6v /6Ç = q26
6v /6y = q46
6v /6x = [J] $^{-1}$ 6v
/6Ç 6v /6y 6v
/6y

6v /6x = (1/ |J|) y23 y31 q2- q6
6v /6y x32 x13 q4 –q6

6v /6x = (1/ |J|) y23 (q2- q6) + y31 (q4 –q6)
6v /6y x32 (q2- q6) + x13 (q4 –q6)

6v /6x = (1/ |J|) y23 q2- y23 q6 + y31 q4 – y31q6
6v /6y x32 q2- x32q6 + x13 q4 – x13q6

6v /6x = (1/ |J|) y23q2 + y31 q4 - y23q6– y31q6
6v /6y x32 q2 + x13 q4 - x32q6– x13q6

6v /6x = (1/ |J|) y23q2 + y31q4 - q6(y2 - y3+ y3 -y1)
6v /6y x32 q2 + x13 q4 - q6(x3 -

x2+x1 - x3) canceling y3 and x3 , we get

6v /6x = (1/ |J|) y23q2 + y31q4 - q6(y2 -y1)

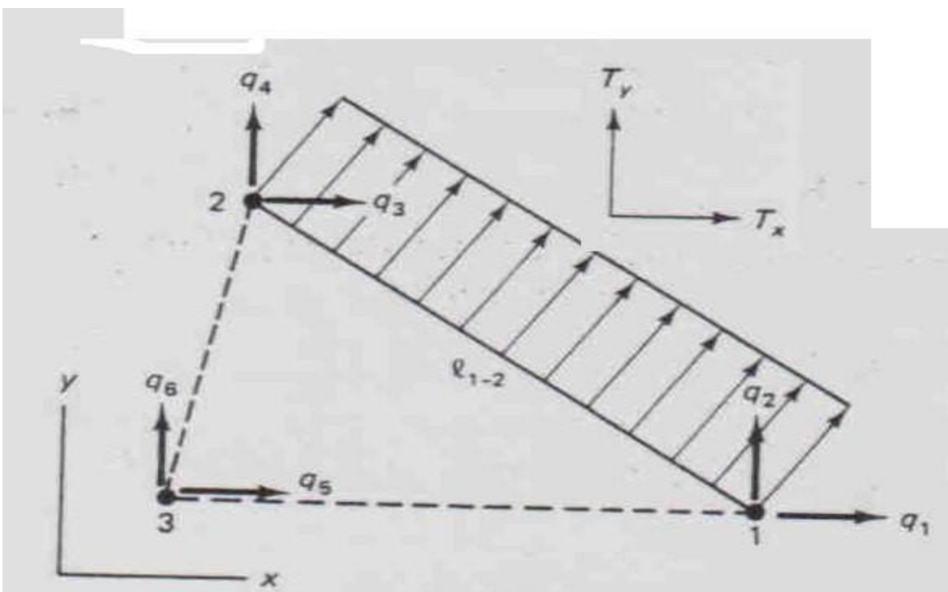
6v /6y x32 q2 + x13 q4 - q6(- x2+x1)

6v /6x = (1/ |J|) y23q2 + y31q4 + q6(y1 + y2)
6v /6y x32 q2 + x13 q4 + q6(x2+x1)

6v /6x = (1/ |J|) y23q2 + y31q4 + y12 q6
6v /6y x32 q2 + x13 q4 +

x21q Writing in matrix form

6v /6x = 1/ |J| 0 y23 0
y31 0 y12 q1 6v /6y
0 x32 0 x13 0 x21
q2 q3
q4 q5 q6



Triangular element with traction force on edge 1-2

Let u and v are the displacements and Tx , Ty are the components of traction forces

W.p. due to traction force = ∫ (u Tx+
v Ty) t dl u = N1 q1 + N2 q3 v =
N1 q2 + N2 q4
only one edge connecting two nodes is considered, let l_{1-2} is the edge.

= ∫ [(N1 q1 + N2 q3) Tx + (N1 q2 + N2 q4) Ty] te dl

$= \int (te\ Tx\ N1\ q1 + te\ Tx\ N2\ q3) + (te\ Ty\ N1\ q2 + te\ Ty\ N2\ q4)\ dl$

$= (q1\ te\ Tx \int N1\ dl + q3\ te\ Tx \int N2\ dl) + (q2\ te\ Ty \int N1\ dl + q4\ te$

$Ty \int N2\ dl)$ Arranging them Node wise

$= q1\ (te\ Tx \int N1\ dl) + q2\ (te\ Ty \int N1\ dl) + q3(te\ Tx \int N2\ dl) + q4\ (te\ Ty \int N2\ dl)$

N3 is zero along the edge l_{1-2} , N1 and N2 are similar to the shape functions of 1-D bar element.

Where N1 = (1 – Ç) / 2 and N2 = (1 + Ç) / 2

$\int N1\ dl = \int (1 – Ç) / 2\ (le/2)\ dÇ = (le/2) \int (1 – Ç)/2\ dÇ$ (the integration is between the limits – 1 to 1)

$\int (1 – Ç)/2\ dÇ) = ½ [\int dÇ - \int Ç\ dÇ] = ½ [Ç - Ç^2 /2]$ l - limit = (-1) u - limit = (1)

[½ [1-(-1)] – ½ [(12/ 2) – (-12 /2)]] = [1 - 0] = 1

$\int N1\ dl = (le/2) = l_{1-2} / 2$ =led /2

Stress calculations :

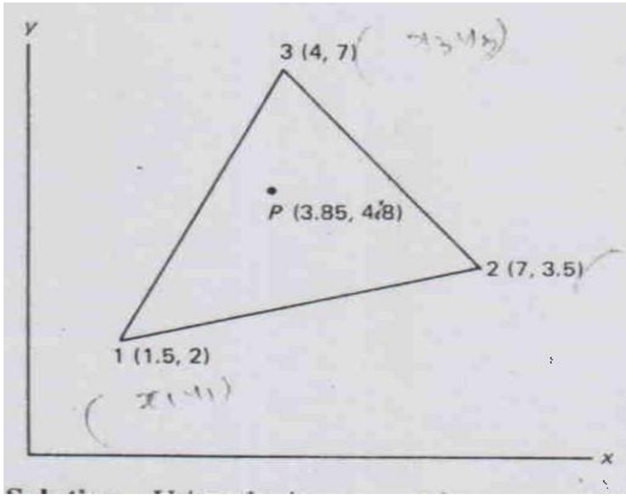
Strains are constant over CST , hence Stresses are also constant over an element.

{o} = [D] [B]{q}

Element connectivity table should be used to extract elemental displacement vector form the Global Displacement vector. Principal stresses and strains are calculated separately using Mohr's circle relations.

Numerical Examples

Evaluate the shape functions N1, N2 and N3 at the interior point P for the triangular element shown in Fig:



Solution : given point P (3.85,4.81) :

the coordinates of the nodes are . node 1 (x1, y1) = (1.5 , 2.0)

node 2 (x2, y2) = (7.0 , 3.5)   node 3 (x3, y3) = (4.0 , 7.0

Consider x = N1 x1 + N2 x2 + N3 x3   y = N1 y1 + N2 y2 + N3 y3 Substituting for x1, y1, x2, y2 , and noting P (3.85,4.81) etc we have 3.85 = 1.5 N1 + 7.0 N2 + 4.0 N3

4.80 = 2.0 N1 + 3.5 N2 + 7.0 N3

But N1 = Ç, N2 = y , N3 = (1 – Ç – y) 3.85 = 1.5 Ç + 7.0 y + 4.0 (1 – Ç – y)

4.80 = 2.0 Ç + 3.5 y + 7.0 (1 – Ç – y),

simplifying and re arranging we get

2.5 Ç – 3 y = 0.15 :   5Ç + 3.5 y = 2.2

2.5 Ç – 3 y = 0.15 : …………..* 2

5Ç + 3.5 y = 2.2 : ………….. *1 and subtract

5 Ç – 6 y = 0. 3 :

--5Ç + --3.5 y = --2.2

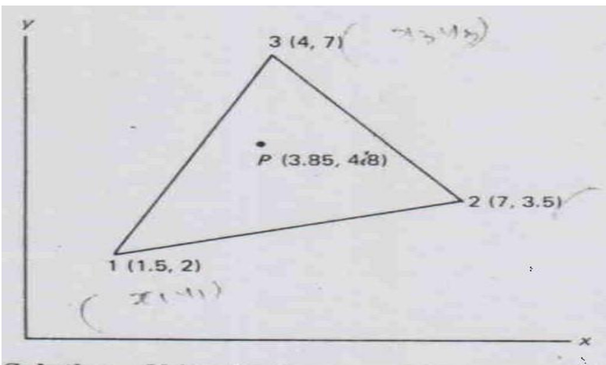
- 9.5 y = -1.9 < y = 0.2

substitute this value in the equation 2.5 Ç – 3 y = 0.15

= 2.5 Ç – 3(0.2) = 0.15 2.5 Ç = 0.15 – 0.6 = 0.75 Ç = 0.75 / 2.5 = 0.3

Thus Ç = 0.3 y = 0.2 is the required Answer

2.0 Determine The Jacobian of transformation for the triangular element shown in Fig: (x1, y1) = (1.5 , 2.0) (x2, y2) = (7.0 , 3.5) (x3, y3) = (4.0 , 7.0)



J	=	x13	y13
		x23	y23

J	=	x1 - x 3 1.5 – 4.0 = - 2.5	y1 - y 3 2.0-7.0 = -5.0
		x2 - x 3 7.0-4.0= 3.0	y2 - y3 3.5-7.0 = -3.5

= (-2.5) (-3.5) – (3)(-5) = **23.75 Ans**

(Note : J = 2*A where A is the area of the triangle)

3.0 Determine The Jacobian of transformation considering the nodes 1 2 3 in clock wise order for the previous problem (take node 3 as node 2).

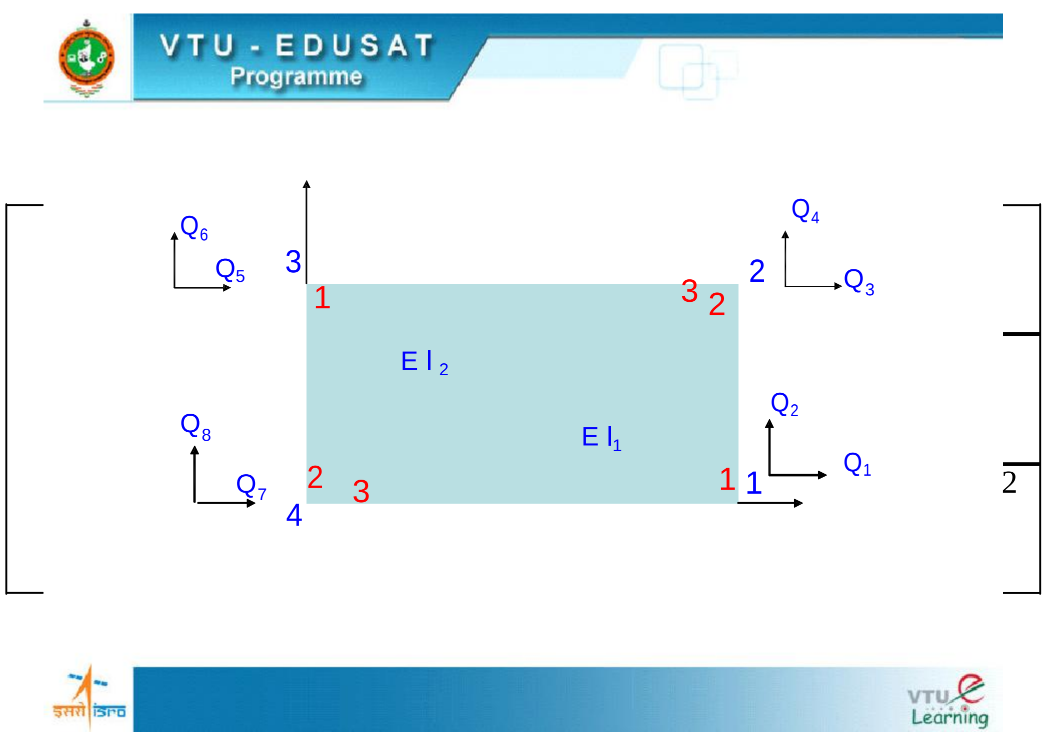
Solution : the value of J becomes negetive

J	=	x1 - x 3 1.5 – 7.0 = - 5.5	y1 - y 3 2.0-3.5 = -1.5
		x2 - x 3 4.0-7.0 = -3.0	y2 - y3 7.0-3.5 = 3.5

J = (-5.5) (3.5) – (-3)(-1.5) = -19.5 – 4.5 = **- 23.75**

(J = 2*A). where A is the area of the triangle

4.0 Find [B]¹ , [B]² for the elements shown in fig below using the local node numbers shown at the corners. Length of rectangle 3 in, breadth = 2 in



Ele ?	1	2	3 Lo
1	1	2	4 Gl
2	3	4	2 Gl

Solution :

Consider left lower corner of the rectangle as the origin

For element (1) (x1,y1) = (3 , 0) (x2,y2) = (3 , 2) (x3,y3) = (0 , 0)

For element (2) (x1,y1) = (0 , 2) (x2,y2) = (0 , 0) (x3,y3) = (3 , 2)

To determine [B] matrix for element 1 :

(x1,y1) = (3 ,0) (x2,y2) = (3, 2) (x3,y3) = (0 , 0)

To determine the | J |

(x1,y1) = (3 ,0) (x2,y2) = (3, 2) (x3,y3) = (0 , 0)

x1 – x3 3 – 0	y1 – y3 0 - 0	3	0	6 – 0 = 6
x2- x3 3 – 0	y2 – y3 2 - 0	3	2	

| J | = 6

1/ |J| = 1 / 6	y2 -y 3 2-0	0	y3 - y1 0 - 0	0	y1- y 2 0 - 2	0
	0	x3 -x 2 0 - 3	0	x1 - x3 3 – 0	0	x2 -x1 3 - 3
	x3 -x 2 - 3	y2 -y 3 2	x1 - x3 3	y3 - y1 0	x2 -x1 0	y1- y 2 - 2

To determine [B] matrix for element 2 :

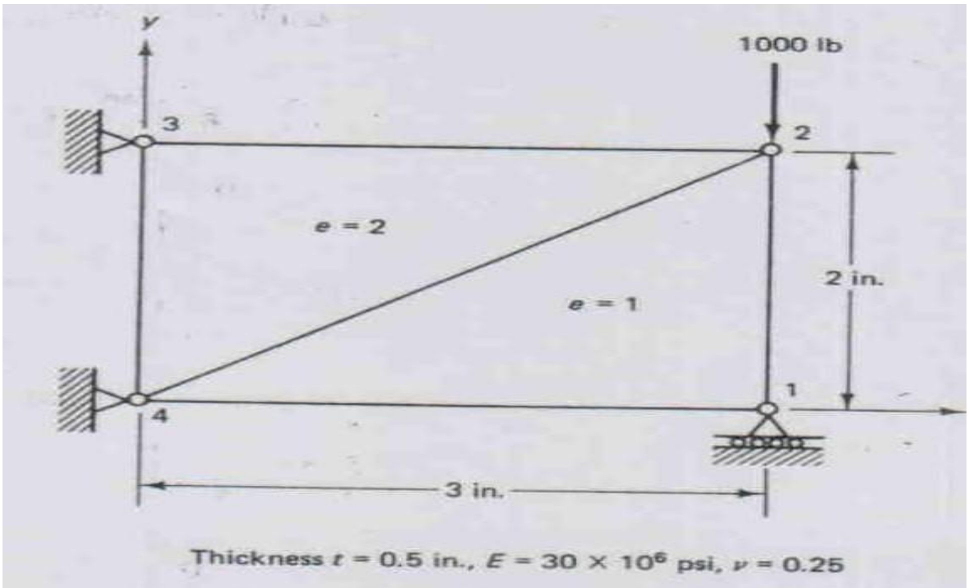
(x1,y1) = (0 , 2) (x2,y2) = (0 , 0) (x3,y3) = (3 , 2)

1 / 6	2-0	0	0 - 0	0	0 - 2	0
	0	- 3	0	3	0	0
	|- 3	2	3	0	0	- 2
	[B] 1 for element 1					

1/ | J |	y2 -y 3 0 - 2	0	y3 - y1 2 - 2	0	y1- y 2 2 - 0	0
	0	x3 -x 2 3 - 0	0	x1 - x3 0 - 3	0	x2 -x1 0 - 0
	x3 -x 2 3	y2 -y 3 -2	x1 - x3 -3	y3 - y1 0	x2 -x1 0	y1- y 2 2

1/ 6	– 2	0	0	0	2	0
	0	3	0	- 3	0	0
	3	-2	-3	0	0	2
	[B] 2 for element 2					

5.0 For the 2-d plate shown, determine the displacements at nodes 1 and 2 and the Element Stresses. Use plane stress. condition . Thickness of the plate t is 0.5 in, take the value of E = 30*106 psi, neglect the effect of body force.



Solution : Knowing the value of E and v observe the

following to get [D] $(30\times10^6) / (1-0.25^2) = 32000000 =$

3.2×10^7

$(1-0.25) / 2 = 0.75 / 2 = 0.375$ $3.2 \times 10^7 \times 0.375 = 1.2 \times 10^7$

$3.2 \times 10^7 \times 0.25 = 0.8\times10^7$, with these note the [D] in the next page

$[B]^1$ has already been determined in the previous problem, let us multiply [D] & $[B]^1$ – for element 1

For plane stress condition [D] is given by

D	= =	E / (1-v 2) [3.2x10 7]	1 [1]	V [0.25]	0 [0]
			V [0.25]	1 [1]	0 [0]
			0 [0]	0 [0]	1-v / 2 [1-0.25] / 2

=

[3.2x10 7]	[0.8x 10 7]	0
[0.8x10 7]	[3.2x10 7]	0
0	0	[1.2x10 7]

Taking out 10^7 common from the elements of [D] and multiplying with B^1 as shown below

(3.2*2) /6 = 1.067   0.8 *(-3) /6 = -0.4   (-2 *3.2) / 6 = -1.067

(0.8 * 2) / 6 = 0.267   (3.2 * (-3) / 6 = -1.6 1.2 * (-3) / 6 = -0.6

(1.2 * 2) / 6 = 0.4   There are only few multiplication to do, operations repeat

10 7 [D]	3.2	0.8	0
	0.8	3.2	0
	0	0	1.2

1/6 $[B]^1$	2	0	0	0	-2	0
	0	-3	0	3	0	0
	-3	2	3	0	0	-2

=

10^7	1.067	-0.4	0	0.4	-1.067	0
	0.267	-1.6	0	1.6	-0.267	0
	-0.6	0.4	0.6	0	0	-0.4
		This is DB^1				

$[B]^2$ matrix has already been determined let us multiply [D] & $[B]^2$ – for element 2

10^7	3.2	0.8	0
	0.8	3.2	0
[D]	0	0	1.2

1/6	-2	0	0	0	2	0
$[B]^2$	0	3	0	-3	0	0
	3	- 2	-3	0	0	2

=

10^7	-1.067	0.4	0	-0.4	1.067	0
	-0.267	1.6	0	-1.6	0.267	0
	0.6	-0.4	-0.6	0	0	0.4
		This is DB^2				

Observe DB^1 and DB^2 , the elements are same except for +ve or –ve sign.

To calculate stiffness matrices k^1 and K^2 :

$k^1 = t_e A_e \; B^{1T} [D] [B]^1$ $\quad k^2 = te \; Ae \; B^{2T} [D] [B]^2$

First look at the following simple

calculations , t e = 0.5 in

A e = ½ b * h = ½ * 3 * 2 = 3 in 2 t e A e = 0.5 * 3 = 1.5 in 3

(te Ae) / 6 = 1.5 / 6 = 0.25 (1/6 is of $[B]^{1T}$)

2 * 0.25 = **0.5** : -3 * 0.25 = **-0.75** -2 * 0.25 = **-0.5** : 3 * 0.25 = **0.75** etc

0.25	2	0	-3
	0	-3	2
	0	0	3
	0	3	0
	-2	0	0
	0	0	-2

0.5	0	-0.75
0	-0.75	0.5
0	0	0.75
0	0.75	0
-0.5	0	0
0	0	-0.5
This is $[A_e\ t_e / 6]\ [B]^{1T}$		

$k^1 = t_e\ A_e\ B^{1T}\ [D]\ [B]^1 =$

0.5	0	-0.75
0	-0.75	0.5
0	0	0.75
0	0.75	0
-0.5	0	0
0	0	-0.5
This is $[A_e\ t_e / 6]\ [B]^{1T}$		

10^7	1.067	-0.4	0	0.4	-1.067	0
	0.267	-1.6	0	1.6	-0.267	0
	-0.6	0.4	0.6	0	0	-0.4
		This is DB^1				

(0.5 * 1.067) + (-0.75) *(-0.6) = 0.5335 + 0.45 = 0.9835

0.5*(-0.4) + (-0.75*0.4) = - 0.20 – 0.30 = -0.50

-0.75 * 0.6 = -0.45 ; -0.75 *(-1.6)+0.5* 0.4 = 1.4

-0.75 *(1.6) = -1.2 0.5*0.4 = 0.20

0.5 (-1.067) = -0.5335 = -0.533

-0.75 * (-0.4) = 0.3

10^7

Q1	Q2	Q3	Q4	Q7	Q8	
0.983	-0.5	-0.45	0.2	-0.533	0.3	Q1
-0.5	1.4	0.3	-1.2	0.2	0.2	Q2
-0.45	0.3	0.45	0	0	-0.3	Q3
0.2	- 1.2	0	1.2	-0.2	0	Q4
-0.533	0.2	0	-0.2	0.533	0	Q7
0.3	-0.2	-0.3	0	0	0.2	Q8
	$k^1 = t_e A_e \; B^{1T} [D] [B]^1$					

Global degrees of freedom associated with element 1 are Q1 ,Q2 ,Q3 ,Q4 ,Q7 ,Q8 see fig To facilitate assembly it should be written in order Q1 ,Q2 ,Q3 ,Q4 ,Q_5 ,Q_6 ,Q7 ,Q8 . It will be shown after determining k^2

To determine k^2 :

$k^2 = t_e A_e B^{2\,T} [D] [B]^2$ =

0.5	0	-0.75
0	-0.75	0.5
0	0	0.75
0	0.75	0
-0.5	0	0
0	0	-0.5
This is $[A_e t_e / 6] [B]^{2T}$		

10^7	-1.067	0.4	0	-0.4	1.067	0
	-0.267	1.6	0	-1.6	0.267	0
	0.6	-0.4	-0.6	0	0	0.4
		This is DB^2				

(0.5 * 1.067) + (-0.75) *(-0.6)
= 0.5335 + 0.45 = 0.9835 =0.983

0.5*(-0.4) + (-0.75*0.4)
= - 0.20 – 0.30 = -0.50

-0.75 * 0.6 = -0.45 ;

-0.75 *(-1.6)+0.5* 0.4 = 1.4

-0.75 *(1.6) = -1.2 ; 0.5*0.4 = 0.20

0.5 (-1.067) = -0.5335 = -0.533

-0.75 * (-0.4) = 0.3

Q5	Q6	Q7	Q8	Q3	Q4	
0.983	-0.5	-0.45	0.2	-0.533	0.3	Q_5
-0.5	1.4	0.3	-1.2	0.2	0.2	Q_6
-0.45	0.3	0.45	0	0	-0.3	Q_7
0.2	- 1.2	0	1.2	-0.2	0	Q_8
-0.533	0.2	0	-0.2	0.533	0	Q_3
0.3	-0.2	-0.3	0	0	0.2	Q_4
	$K^2 = t_e A_e \quad B^{2T} [D] [B]^2$					

Global degrees of freedom associated with element 2 are Q5 ,Q6 ,Q7 ,Q8 ,Q3 ,Q4 see fig To facilitate assembly it should be written in order Q1 ,Q2 ,Q3 ,Q4 ,Q_5 ,Q_6 ,Q7 ,Q8 . It will be shown below .

10^7

K_1 modified								
Q1	Q2	Q3	Q4	Q5	Q6	Q7	Q8	
0.983	-0.5	-0.45	0.2	0.0	0.0	-0.533	0.3	Q1
0.5	1.4	0.3	-1.2	0.0	0.0	0.2	-0.2	Q2
-0.45	0.3	0.45	0.0	0.0	0.0	0.0	-0.3	Q3
0.2	-1.2	0.0	1.2	0.0	0.0	-0.2	0.0	Q4
0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Q5
0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Q6
0.533	0.2	0.0	0.2	0.0	0.0	0.533	0.0	Q7
0.3	0.2	-0.3	0.0	0.0	0.0	0.0	0.2	Q8

+

10^7

K_2 modified								
Q1	Q2	Q3	Q4	Q5	Q6	Q7	Q8	
0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Q1
0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0	Q2
0.0	0.0	0.533	0.0	0.533	0.2	0.0	-0.2	Q3
0.0	0.0	0.0	0.2	0.3	0.20	-0.3	0.0	Q4
0.0	0.0	-0.533	0.3	0.983	-0.5	-0.45	0.2	Q5
0.0	0.0	0.2	-0.2	-0.5	1.4	0.3	1.2	Q6
0.0	0.0	0.0	-0.3	-0.45	0.3	0.45	0.0	Q7
0.0	0.0	-0.2	0.0	0.2	-1.2	0.0	1.2	Q8

=

10^7

K $_1$ modified + K $_2$ modified								
Q1	Q2	Q3	Q4	Q5	Q6	Q7	Q8	
0.983	-0.5	- 0.45	0.2	0.0	0.0	-0.533	0.3	Q1
0.5	1.4	0.3	-1.2	0.0	0.0	0.2	-0.2	Q2
-0.45	0.3	0.983	0.0	0.533	0.2	0.0	-0.5	Q3
0.2	-1.2	0.0	1.4	0.3	0.2	-0.5	0.0+	Q4
0.0	0.0	-0.533	0.3	0.983	–0.5	-0.45	+0.2	Q5
0.0	0.0	0.2	- 0.2	- 0.5	1.4	0.3	1.2	Q6
0.533	0.2	0.0	-0.1	–0.45	0.3	0.983	0.0	Q7
0.3	0.2	-0.5	0.0	0.2	- 1.2	0.0	+1.4	Q8

OVERALL EQUATION TO BE SOLVED **[K] [Q] = [F]**											
Q1	Q2	Q3	Q4	Q5	Q6	Q7	Q8				
0.983	-0.5	- 0.45	0.2	0.0	0.0	-0.533	0.3		Q1		0
0.5	1.4	0.3	-1.2	0.0	0.0	0.2	-0.2		Q2		0
-0.45	0.3	0.983	0.0	0.533	0.2	0.0	-0.5		Q3		0
0.2	-1.2	0.0	1.4	0.3	0.2	-0.5	0.0+	*	Q4	=	-1000
0.0	0.0	-0.533	0.3	0.983	–0.5	-0.45	+0.2		Q5		0
0.0	0.0	0.2	- 0.2	- 0.5	1.4	0.3	1.2		Q6		0
0.533	0.2	0.0	-0.1	–0.45	0.3	0.983	0.0		Q7		0
0.3	0.2	-0.5	0.0	0.2	- 1.2	0.0	+1.4		Q8		0

The boundary conditions
are : Node 1 has roller
support Q2 = 0
Node 3 is fixed Q7 = 0 , Q8 = 0
Node 4 is fixed Q5 = 0, Q6

= 0 The dof Q2 Q5 Q6 Q7

Q8 = 0

Therefore in the assembled matrix by the method of elimination Rows 2, 5, 6, 7, 8 and Columns 2, 5, 6, 7, 8 will cancel

10^7

[K] assembled Boundary condition applied								
Q1	Q2	Q3	Q4	Q5	Q6	Q7	Q8	
0.983		**- 0.45**	**0.2**					Q1
								Q2
-0.45		**0.983**	**0.0**					Q3
0.2		**0.0**	**1.4**					Q4
								Q5
								Q6
								Q7
								Q8

	TO B	SOL	ED	
	Q_1	Q_3	Q_4	
10^7	0.983	-0.45	0.2	Q_1
	-0.45	0.983	0.0	Q_3
	0.2	0.0	1.4	Q_4

10^7 [0.983 Q1 – 0.45 Q3 + 0.2 Q4] = 0
10^7 [-0.45 Q1 + 0.983 Q3] = 0
10^7 [0.2 Q1 + 1.4 Q4] = -1000
Take 10^3 inside the bracket

10^4 [983 Q1 – 450 Q3 + 200 Q4] = 0
10^4 [-450 Q1 + 983 Q3] = 0
10^4 [200 Q1 + 1400 Q4] = -1000

Now divide all eqs by 10^4
[983 Q1 – 450 Q3 + 200 Q4] = 0 ……………eq. 1
[-450 Q1 + 983 Q3] = 0 ………………eq. 2
[200 Q1 + 1400 Q4] = - 0.1 ………………..eq. 3

rewriting eq. 1,

983 Q1 = 450 Q3 - 200 Q4

Q1 = 0.457 Q3 -0.203 Q4 ; substituting this

Q1 in eq. 2 we get, -450 (0.457 Q3 -0.203

Q4) + 983 Q3 = 0

= 777.35 Q3 + 90 Q4 = 0 ………. eq. 4

Similarly substituting for Q1 in eq. 3
200 (0.457 Q3 -0.203 Q4) + 1400 Q4 = - 0.1,
simplifying we get , 91.4 Q3 + 1359.4 Q4 = -
0.1………. eq. 5

now consider eq.4 and eq.5

777.35 Q3 + 90.0 Q4 = 0.0 …….x 91.4

91.4 Q3 + 1359.4 Q4 = -0.1 …….x 777.35

71049.79 Q3 + 8226 Q4 = 0.0

71049.79 Q3 + 1056729.59 Q4 = -77.735, now subtract

- 1048503.59 Q4 = 77.735

Q4 = -7.414 x 10 -5 in substitute this in eq. 4

777.35 Q3 + 90 (-7.414 x 10-5) = 0

Q3 = 8.5848x 10-6 = 0.854 x 10-5 in

Q3 = 0.854 x 10-5 in

Substituting for Q3 in the equation

-450 Q1 + 983 Q3 = 0

-450 Q1 + 983 (0.854 x 10-5) = 0 , we get

Q1 = 1.866 x 10-5 in

Answer from the text book is

Q1 = 1.913 * 10-5 inQ3 = 0.875 * 10-5 in Q4 = -7.436 * 10-5 in ……..ok

Element – 1		
10^5	Q1	q1 = 1.913 in
	Q2	q2 = 0.0 in
	Q3	q3= 0.875 in
	Q4	q4= -7.436 in
	Q7	q5 = 0.0 in
	Q8	q6 = 0.0 in

=

Element – 2		
10^5	Q5	q1 = 0.0 in
	Q6	q2 = 0.0 in
	Q7	q3= 0.0 in
	Q8	q4= 0.0 in
	Q3	q5 = 0.875 in
	Q4	q6 = -7.436 in

To calculate stresses in the elements :

Stresses acting on element 1 $\sigma_1 = [D]\,[B]^1 \times [q^1]$

10^7	1.067	-0.4	0	0.4	-1.067	0
	0.267	-1.6	0	1.6	-0.267	0
	-0.6	0.4	0.6	0	0	-0.4
		This is DB^1				

10^5	1.913
	0.0
	0.875
	-7.436
	0.0
	0.0

93.3 psi	σ
-1138.7 psi	σ_y
62.3 psi	τ_{xy}

[(1.067 x 1.913) – (0.4 x 7.436)] (10^2) = [2.041171-2.9744] (10^2) = -93.3

[(0.267 x 1.913) – (1.6 x 7.436)] (10^2) = [0.510771-11.8976] (10^2) -1138.7

[(-0.6 x 1.913) + (0.6 x .875)] (10^2) = [-1.1478 +0.525] (10^2) = - 62.3

Stresses acting on element 2 ***$o_1 = [D] [B]^2 x [q^2]$***

10^7	-1.067	0.4	0	-0.4	1.067	0
	-0.267	1.6	0	-1.6	0.267	0
	0.6	-0.4	-0.6	0	0	0.4
		This is DB[1]				

10^5	0
	0.0
	0.0
	0.0
	0.875
	-7.436

=

93.4 psi	o
23.4 psi	o_y
-297.4 psi	$\imath_{xy}$

[1.067 x 0.875] (10^2) = (0.933625) (10^2) = 93.4

[0.267 x 0.875)] (10^2) = (0.233625) (10^2) = 23.4

[0.4 x -7.436] (10^2) = (-2.9744) (10^2) = - 297.4

o 1 = [o_x = -93.3, o_y = -1138.7, $\imath_{xy}$ - 62.3 $]^T$ psi

o 2 = [o_x = 93.4, o_y = 23.4, $\imath_{xy}$ = - 297.]

$_T$ psi Acting at the centroid of the elements

UNIT-IV

TWO-DIMENSIONAL ANALYSIS USING QUADRILATERAL ELEMENTS

Introduction

Many engineering structures and mechanical components are subjected to loading in two directions. Shafts, gears, couplings, mechanical joints, plates, bearings, are few examples. Analysis of many three dimensional systems reduces to two dimensional, based on whether the loading is plane stress or plane strain type. Iso-parametric Quadrilateral elements are widely used in the analysis of such components and systems. For Iso-parametric quadrilateral elements the derivation of shape function is simple and the stiffness matrix is generated using numerical Integration . The various load vectors, displacement vectors, stress vectors and strain vectors used in the analysis are as written below,

the displacement vector **u** = $[u, v]^T$,

u is the displacement along x direction, v is the displacement along y direction,

the body force vector **f** = $[f_x , f_y]^T$

f_x , is the component of body force along x direction, f_y is the component of body force along y direction

the traction force vector **T** = $[T_x , T_y]^T$

T_x , is the component of body force along x direction, T_y is the component of body force along y direction

Two dimensional stress strain equations

From theory of elasticity for a two dimensional body subjected to general loading the equations of equilibrium are given by

$$[6o_x /6x] + [6ı_{yx} /6 y] + F_x = 0$$

$$[6ı_{xy} /6x] + [6o_y /6y] + F_y = 0$$

Also $ı_{xy} = ı_{yx}$

The strain displacement relations are given by

s_x = 6u /6x, s_y = 6v /6y, y_{xy} = 6u /6y + 6v /6x

$s = [6u /6x,\ 6v /6y,\ (6u /6y + 6v /6x)]^T$

The stress strain relationship for plane stress and plane strain conditions are given by the matrices shown in the next page. o_x o_y $ı_{xy}$ s_x s_y y_{xy} are usual stress strain components, v is the poisons ratio. E is young's modulus. Please note the differences in [D] matrix .

The stress strain relationship for plane stress loading is given by

o_x			1	V	0		s_z
o_y	=	$E / (1-v^2)$	V	1	0	*	s_y
$\imath_{xy}$			0	0	1-v / 2		y_{yz}

[**O**] = [**D**] [S]

The stress strain relationship for plane strain loading is give by

o_x			1-v	V	0		s_z
o_y	=	E / (1+v)(1-2v)	V	1-v	0	*	s_y
$\imath_{xy}$			0	0	½ -v		y_{yz}

[**O**] = [**D**] [S]

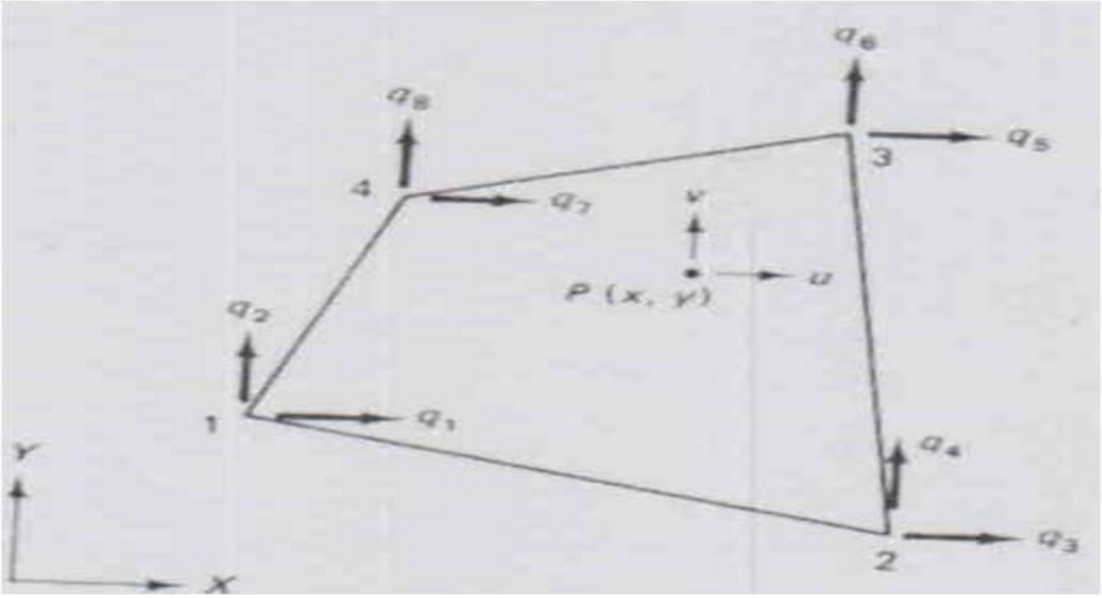
The element having mid side nodes along with corner nodes is a higher order element. Element having curved sides is also a higher order element.

A simple quadrilateral element has straight edges and corner nodes. This is also a linear element. It can have constant thickness or variable thickness. The quadrilateral having mid side nodes along with corner nodes is a higher order element. Element having curved sides is also a higher order element.

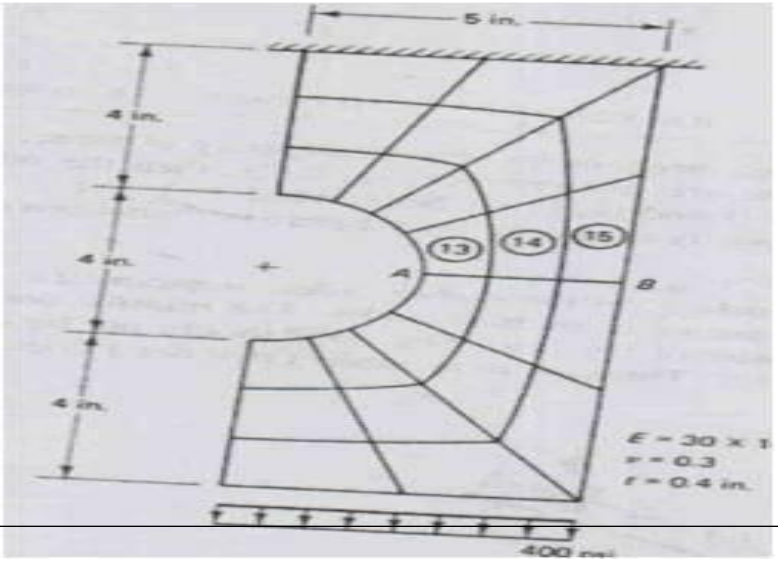
The given two dimensional component is divided in to number of quadrilateral elements. If the component has curved boundaries certain small region at the boundary is left uncovered by the elements. This leads to some error in the solution.

Quadrilateral

It is a quadrilateral element having four straight sides joined at four corners. and imagined to have a node at each corner. Thus it has four nodes, and each node is permitted to displace in the two directions, along x and y of the Cartesian coordinate system. The loads are applied at nodes. Direction of load will also be along x direction and y direction, +ve or –ve etc. Each node is said to have two degrees of freedom. The nodal displacement vector for each element is given by,



Four Node Quadrilateral



A body discretized using quadrilaterals

$\mathbf{q} = [q_1, q_2, q_3, q_4, q_5, q_6, q_7, q_8]$

q_1, q_3, q_5, q_7 are nodal displacements along x direction of node1, node2 and node3 node4 simply called horizontal displacement components.

q_2, q_4, q_6, q_8 are nodal displacements along y direction of node1, node2 and node3 node 4 , simply called vertical displacement components. q_{2j-1} is the displacement component in x direction and q_{2j} is the displacement component in y direction.

Similarly the nodal load vector has to be considered for each element. Point loads will be acting at various nodes along x and y …………………………

(x_1, y_1), (x_2, y_2), (x_3, y_3), (x_4, y_4) are cartesian coordinates.of node 1 node 2 node 3. and node 4

In the discretized model of the continuum the node numbers are progressive, like 1,2,3,4,5,6,7,8……….etc and the corresponding displacements are Q_1, Q_2, Q_3, Q_4, Q_5 Q_6, Q_7, Q_8, Q_9, Q_{10}….. Q_{16}, two displacement components at each node.

Q_{2j-1} is the displacement component in x direction and Q_{2j} is the displacement component in y direction. Let j = 10, ie 10th node, $Q_{2j-1} = Q_{19}$ $Q_{2j} = Q_{20}$

The element connectivity table shown establishes correspondence of local and global node numbers and the corresponding degrees of freedom. Also the (x_1, y_1), (x_2, y_2) and (x_3, y_3), (x_4, y_4) have the global correspondence established through the table.

Element Connectivity Table Showing Local – Global Node Numbers

Element Number	Local Nodes Numbers 1	2	3	4
1	1	2	3	4
2	3	4	5	6
3	5	6	7	8

..		..	..	..
11	12	19	14	21
..		..	..	..
20				

Nodal Shape Functions: under the action of the given load the nodes are assumed to deform linearly. element has to deform elastically and the deformation has to become zero as soon as the loads are zero. It is required to define the magnitude of deformation and nature of deformation

for the element. Shape functions or Interpolation functions are used to model the magnitude of displacement and nature of displacement.

A Quadrilateral has Four Nodes, each node having Two Degrees of Freedom (Displacements)
Displacement along x direction and y direction

[q] = [q1 , q 2 , q 3 , q4 , q 5 , q6 ,q7 ,q8] ^{T} , This is nodal displacement vector q1 q3 q5 q7 displacements along x direction of node1, node2 , node3 and node 4 q2 q4 q6 q8 displacements along y direction of node1, node2 , node3 and node 4

Nodal coordinates are (x1 , y1), (x2 , y2) , (x3 , y3) (x4 , y4). The displacement of an interior point P (x,y) is given by u = [u(x,y), v(x,y)]^T

The local nodes are numbered 1,2,3 and 4 in counter clockwise fashion. The loads are applied at nodes (+ ve or – ve)

The Master Quadrilateral is defined in Ç , y coordinate system. It is a square having four nodes each node having two dof . Four Lagrange shape functions N1 , N2 , N3 and N4 are used to model the displacement. Ni is unity at node i and zero at other nodes

N1 = 1 at node 1, 0 at nodes 2,3,4 , ……………………….eq(1)
this means N1 = 0 along edges Ç = (+1) and y = (+1) So by intuition N1 has

to be of the form N1 = c (1 – Ç) (1 – y) …………eq(2)
where c is a constant

N1 = 1, at Ç = (-1) and y = (-1) ie at Node 1,
therefo
re 1 = c (2) (2) giving c = 1/4 ,
……………eq(3) thus N1 = ¼ (1 – Ç) (1
– y), …………eq(4)

similarly other shape functions are also written
N1 = ¼ (1 – Ç) (1 – y), N2 = ¼ (1 + Ç) (1 – y), N3 = ¼ (1 +Ç) (1 + y),
N4 = ¼ (1 – Ç) (1+ y),

…………………………..

…..eq(5) Ni = ¼ (1– Ç Çi) (1 – yyi) (Çi , yi) are

coordinates of node i

At node 1 Ç = (-1) , y= (-1) N1 = ¼ (1 – Ç) (1 – y),
At node 2 Ç = (+1) , y = (-1) N2 = ¼ (1 + Ç
) (1 – y), At node 3 Ç = (+1) , y = (+1) N3
= ¼ (1 + Ç) (1+ y), At node 4 Ç = (1) , y
= (-1) N4 = ¼ (1 - Ç) (1 + y),

Iso Paramatric Formulation :

u = N1 q1 + N2 q3 + N3 q5 + N4 q7 v = N1 q2 + N2 q4 + N3 q6 +
N4 q8 x = N1 x1 + N2 x2 + N3 x3 + N4 x4 y = N1 y1 + N2 y2
+ N3 y3 + N4 y4

The same shape functions are used to define the displacement and geometry of the element.
This is called Iso-Parametric formulation.

$$\begin{bmatrix} u \\ v \end{bmatrix} = \begin{bmatrix} N1 & 0 & N2 & 0 & N3 & 0 & N4 & 0 \\ 0 & N1 & 0 & N2 & 0 & N3 & 0 & N4 \end{bmatrix} \begin{bmatrix} q1 \\ q2 \\ q3 \\ q4 \\ q5 \\ q6 \\ q7 \\ q8 \end{bmatrix}$$

$$\begin{bmatrix} x \\ y \end{bmatrix} = \begin{bmatrix} N1 & 0 & N2 & 0 & N3 & 0 & N4 & 0 \\ 0 & N1 & 0 & N2 & 0 & N3 & 0 & N4 \end{bmatrix} \begin{bmatrix} x1 \\ y1 \\ x2 \\ y2 \\ x3 \\ y3 \\ x4 \\ Y4 \end{bmatrix}$$

Potential Energy :

Total Potential Energy of an Elastic body subjected to general loading is given by

n = Elastic Strain Energy + Work Potential

$n = ½ \int o^T s \, dv - \int u^T f \, dv - \int u^T T \, ds - \Sigma u^T i \, Pi$

For the 2- D body under consideration P.E. is given by

$v = ½ \int s^T \mathbf{D} \, s \, \mathbf{te \, dA} - \int \mathbf{u^T f \, t \, dA} - \int \mathbf{u^T T \, t \, dl} - Z\mathbf{u^T i \, Pi}$

This expression is utilised in deriving the elemental properties such as Element stiffness matrix

[K] , load vetors $\mathbf{f^e}$ **,** $\mathbf{T^e}$ **,** etc.

Derivation of Strain Displacement Equation and Stiffness Matrix for (derivation of [B] and [K]) :

$s_x = 6u / 6x \quad s_y = 6v / 6y \quad y_{xy} = 6u / 6y + 6v / 6x$

u = u (x (Ç ,y) , y (Ç ,y)) v = v (x (Ç ,y) , y (Ç ,y))

To get expressions for different strain components, derivations which are almost similar has to be repeated twice. That is what we did in the case for CST.

Instead we consider f = f [x (Ç ,y) , y (Ç ,y)] as a general implicit function, derive Jacobean, 6f /6x, 6f /6y, 6f /6Ç , 6f /6y etc and use them changing suitably as functions for u or v etc. This way the derivations become simple.

$$\frac{\partial f}{\partial \xi} = \frac{\partial f}{\partial x}\frac{\partial x}{\partial \xi} + \frac{\partial f}{\partial y}\frac{\partial y}{\partial \xi} \qquad \frac{\partial f}{\partial \eta} = \frac{\partial f}{\partial x}\frac{\partial x}{\partial \eta} + \frac{\partial f}{\partial y}\frac{\partial y}{\partial \eta}$$

$$\begin{Bmatrix} \frac{\partial f}{\partial \xi} \\ \frac{\partial f}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \begin{Bmatrix} \frac{\partial f}{\partial x} \\ \frac{\partial f}{\partial y} \end{Bmatrix}$$

$$\begin{Bmatrix} \frac{\partial f}{\partial \xi} \\ \frac{\partial f}{\partial \eta} \end{Bmatrix} = \mathbf{J} \begin{Bmatrix} \frac{\partial f}{\partial x} \\ \frac{\partial f}{\partial y} \end{Bmatrix}$$

where [J] = is called Jacobian

$$\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}$$

$$\frac{\partial f}{\partial x} \qquad\qquad \frac{\partial f}{\partial \xi}$$

$$= [\,\mathbf{J}\,]^{-1}$$

$$\frac{\partial f}{} \qquad\qquad \frac{\partial f}{}$$

∂y $\partial \eta$

Let us determine [J] , $[J]^{-1}$, 6f /6Ç , 6f /6y , 6x /6Ç , 6x /6y , 6y /6Ç , 6y /6y etc consider f = f [x (Ç ,y) , y (Ç ,y)]

x = N1 x1 + N2 x2 + N3 x3 + N4 x4 y = N1 y1 + N2 y2 + N3 y3 + N4 y4

x = ¼ (1 – Ç) (1 – y) x1 + ¼ (1 + Ç) (1 – y) x2
+ ¼ (1 +Ç) (1 + y) x3 + ¼ (1 – Ç) (1+ y)x4

6x / 6Ç = ¼ [- (1 – y) x1 + (1 – y) x2 + (1 + y) x3 – (1+ y)x4]
6x / 6y = ¼[- (1 – Ç) x1 - (1 + Ç) x2 + (1 + Ç) x3+ (1- Ç)x4]

y = ¼ (1 – Ç) (1 – y) y1 + ¼ (1 + Ç) (1 – y) y2
+ ¼ (1 +Ç) (1 + y) y3 + ¼ (1 – Ç) (1+ y) y4

6y / 6Ç = ¼ [- (1 – y) y1 + (1 – y) y2 + (1 + y) y3 – (1+ y)y4]
6y / 6y = ¼ [- (1 – Ç) y1 - (1 + Ç) y2 + (1 + Ç) y3+ (1- Ç)y4]

Writing the elements of Matrix J

$$J = \begin{bmatrix} J_{11} & J_{12} \\ J_{21} & J_{22} \end{bmatrix}$$

J_{11} = ¼ [- (1 – y) x1 + (1 – y) x2 + (1 + y) x3 – (1+ y)x4]

J_{12} = ¼ [- (1 – y) y1 + (1 – y) y2 + (1 + y) y3 – (1+ y)y4]

J_{21} = ¼ [- (1 – Ç) x1 - (1 + Ç) x2 + (1 + Ç) x3+ (1- Ç)x4]

J_{22} = ¼ [- (1 – Ç) y1 - (1 + Ç) y2 + (1 + Ç) y3+ (1- Ç)y4]

We have $\mathbf{[J]^{-1} = 1/ |J| * [co]^{T}}$

$$\mathbf{J} = \begin{bmatrix} J_{11} & J_{12} \\ J_{21} & J_{22} \end{bmatrix}$$

$$[\mathbf{J}]^{-1} = 1/|J| \begin{bmatrix} J_{22} & -J_{12} \\ -J_{21} & J_{11} \end{bmatrix}$$

$$\begin{Bmatrix} \dfrac{\partial f}{\partial x} \\[2ex] \dfrac{\partial f}{\partial y} \end{Bmatrix} = 1/|J| \begin{bmatrix} J_{22} & -J_{12} \\ -J_{21} & J_{11} \end{bmatrix} * \begin{Bmatrix} \dfrac{\partial f}{\partial \xi} \\[2ex] \dfrac{\partial f}{\partial \eta} \end{Bmatrix}$$

s_x = 6u /6x = 1/ |J| [(J_{22} 6u /6 Ç) + (- J_{12} 6u /6 y)] Ç ,y))

6u /6y = 1/ |J| [(-J_{21} 6u /6 Ç) + (J_{11} 6u /6 y)]

6v /6x = 1/ |J| [(J_{22} 6v /6 Ç) + (- J_{12} 6v /6 y)]

s_y = 6v /6y = 1/ |J| [(-J_{21} 6v /6 Ç) + (J_{11}

6v /6 y)] This equation is utilized in

deriving [B] [k] etc

Changingf to u and v we get following matrices ………

$$\begin{Bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial u}{\partial y} \end{Bmatrix} = 1/|J| \begin{bmatrix} J_{22} & -J_{12} \\ -J_{21} & J_{11} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial f}{\partial \xi} \\ \dfrac{\partial f}{\partial \eta} \end{Bmatrix}$$

$$\begin{Bmatrix} \dfrac{\partial v}{\partial x} \\ \dfrac{\partial v}{\partial y} \end{Bmatrix} = 1/|J| \begin{bmatrix} J_{22} & -J_{12} \\ -J_{21} & J_{11} \end{bmatrix} \begin{Bmatrix} \dfrac{\partial f}{\partial \xi} \\ \dfrac{\partial f}{\partial \eta} \end{Bmatrix}$$

Combining the matrices we get expression for strain components

$$\begin{Bmatrix} s_x \\ s_y \\ y_{xy} \end{Bmatrix} \begin{Bmatrix} 6u/6x \\ 6v/6y \\ 6u/6y + 6v/6x \end{Bmatrix} = 1/|J| \begin{bmatrix} J_{22} & -J_{12} & 0 & 0 \\ 0 & 0 & -J_{21} & J_{11} \\ -J_{21} & J_{11} & J_{22} & -J_{12} \end{bmatrix} \begin{Bmatrix} 6u/6\text{Ç} \\ 6u/6y \\ 6v/6\text{Ç} \\ 6v/6y \end{Bmatrix}$$

$$[\, s_{\text{global coordinates}} \,] = [\, A \,]\, [\, s_{\text{local coordinates}} \,]$$

$$\mathbf{[\, s_{gco} \,] = [\, A \,]\ [\, s_{lco} \,]}$$

$$[\, s \,] = [\, A \,]\ [\ G\ ]\, [\, q \,]$$

$$[\, s \,] = [\ \mathbf{B}\ ]\ [\, \mathbf{q} \,]$$

Now let us differentiate u and v w.r.t Ç and y to get strain components in local coordinate system

u = N1 q1 + N2 q3 + N3 q5 + N4 q7
= [¼ (1 – Ç) (1 – y)]q1 + [¼ (1 + Ç) (1 – y)]q3
+ [¼ (1 +Ç) (1 + y)]q5 + [¼ (1 – Ç) (1+ y)]q7

6u /6 Ç =¼ [(–1) (1 – y)]q1 + [(1) (1 – y)]q3 + [(1) (1 + y)]q5 + [(–1) (1+ y)]q7

= ¼ [-(1- y)q1 +(1 – y)q3 +(1 + y)q5 +(-(1+ y)q7)]

6u / 6y =¼ [(–1) (1 – Ç)]q1 + [(-1) (1 + Ç)]q3 + [(1) (1 + Ç)]q5 + [(1) (1- Ç)]q7

= ¼ [-(1- Ç)q1 + (-(1+ Ç)q3) +(1 + Ç)q5 + (1- Ç)q7]

v = N1 q2 + N2 q4 + N3 q6 + N4 q8
= [¼ (1 – Ç) (1 – y)]q2 + [¼ (1 + Ç) (1 – y)]q4
+ [¼ (1 +Ç) (1 + y)]q6 + [¼ (1 – Ç) (1+ y)]q8

6 v /6 Ç = ¼ [-(1- y) q2 +(1 – y) q4 +(1 + y) q6 + (- (1+ y) q8]

6 v / 6y = ¼ [-(1- Ç) q2 +(-(1+ Ç) q4 +(1 + Ç) q6 +(1- Ç) q8]

6u /6 Ç = ¼ [-(1- y)q1 +(1 – y)q3 +(1 + y)q5 +(-(1+ y)q7)]

6u / 6y = ¼ [-(1- Ç)q1 + (-(1+ Ç)q3) +(1 + Ç)q5 + (1- Ç)q7]

Substituting these in the equation [s $_{g\ co}$] = [A] [s $_{l\ co}$]
We get

6u/6Ç	=	1/4	-(1-y)	0	(1–y)	0	(1+y)	0	-(1+y)	0
6u/6y			-(1-Ç)	0	-(1+Ç)	0	(1+Ç)	0	(1-Ç)	0
6v/6Ç			0	-(1-y)	0	(1–y)	0	(1+y)	0	-(1+y)
6v/6y			0	-(1-Ç)	0	-(1+Ç)	0	(1+Ç)	0	(1-Ç)

q_1
q_2
q_3
q_4
q_5
q_6
q_7
q_8

[s] = [B] [q] = [3 x 1]

[s] = [A] [¼ …...] [q …] ; [3x1] = [3x4]
[4x8] [8x1] [s] = [A] [G] [q …]
= [3x8] [8x1]

[B] = [A] [G]

The terms of [B] and |J| are involved functions of Ç & y . The strain in the element is expressed in terms of nodal displacement.

o = D B q where D is 3x3 matrix

Elemental strain energy is given by ½ ∫ o^Ts dv
o = D B q where D is 3x3 matrix

U = Σ t_e ∫e ½ o^Ts dA = ½ te ∫e [D B q]^T B q dA

= ½ q^T [t_e $\int_e$ B^T D B dA] q

U = Σ ½ q^T [t_e ∫ ∫ B^TD B det J dÇ dy] q

= Σ ½ q^T [k^e] q

where k^e = t_e ∫ ∫ B^TDB det J dÇ dy is the element stiffness matrix

B and det J are involved functions of Ç & y, and so the integration has to be performed numerically. The element stiffness matrix is (8 x 8)

The Body force vector $\int_v u^T f\, dv$ **:**

U = Nq
f = [f_x, f_y]^T is constant within each element

$\int_v \mathbf{u^T f\, dv}$ = Σ q^T f_e

$f_e = t_e [\int \int N^T \det d\text{Ç} \, dy] \{ f_x , f_y \}^T$

the body force has to be evaluated by Numerical Integration

Traction Force Vector

Traction force vectors are assumed to act on the edges of the quadrilateral. Let T = [Tx , Ty] act on edge 2-3 , along which Ç = 1. For this edge the shape function becomes . N1 = N4 = 0 , N2 = (1- y) / 2 N3 = (1 + y) / 2 , they are linear functions along the edges, similar to 1-d bar element . From the expression of P.E. eq. the traction force is given by ,

$$\int u^T T \, t \, dl = \int [N q]^T T \, dl = \int [N^T q^T T \, le/2 \, dy$$

$$= q^T [le/2 \int N^T d y]^T \quad = q^T [le/2 \int N^T d y]^T$$

$$T_e = (t_e \, l_{2\text{-}3} / 2) [0 , 0 , T_x , T_y \, T_x , T_y , 0 , 0]^T$$

Numerical Integration And Gauss Quadrature Formula

The solution of many Engineering Problems involve evaluating one or more INTEGRALS. The value of integrals can be evaluated by conventional methods only for simple and continuous functions. In many occasions the integration is to be carried out where the value of integrand is known at discrete points and within an interval. Generally the evaluation of definite integrals by conventional method is tedious, difficult and some time impossible. Numerical methods are generally used as an alternate to conventional method.

A function f (x) is assumed to be continuous in an interval (x_A , x_C). A polynomial is used to approximate the function in this interval (made to pass through certain set of points). The area under the polynomial and the x-axis will clearly, either exceed the actual area for $x_A \le x \le x_B$ or less than the actual area for $x_B \le x \le x_C$ (area between f(x) and x-axis is the actual area). See Fig (a). Therefore the error associated with the integral for. $x = x_A$ to $x = x_C$ is reduced. Higher the order of the polynomial lesser will be the error. Trapezoidal Rule, Simpson's 1/3rd Rule, Simpson's 3/8th Rule, Newtons - Cotes formula etc are basic numerical methods of integration. These methods require equally spaced sampling points (pivotals).

Consider an arbitrary function f(x). The area bound by f(x) and the x-axis for the interval x_A to x_B is given by (see fig (b)).

- $I = \int_{xa}^{xb} f(x)\, dx$ ……….eq. 1

$$\text{let } I = \hat{I} = \int_{xa}^{xb} f(x)\, dx = w_1 f(x_1) + w_2 f(x_2) + w_3 f(x_3) + \dots w_i f(x_i)$$

$$= \Sigma\, w_i f(x_i) \text{ …………eq. 1a}$$

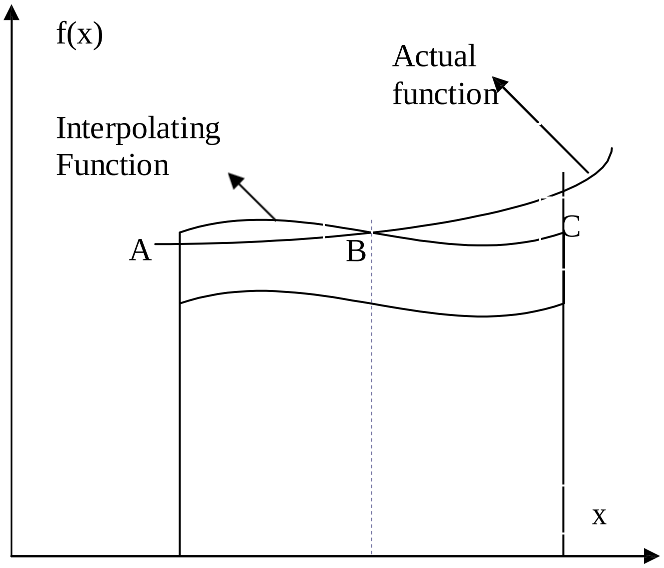


Fig (a)

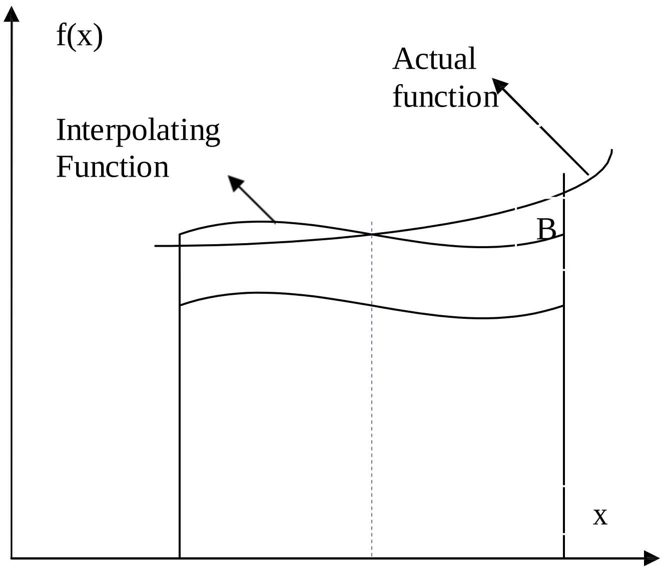


Fig (b)

w_1, w_2, w_3, etc are called weights or weight functions x1, x2, x3 are called gauss points or sampling points

both w_i and x_i are unknowns, They are determined using Legendre polynomials, hence the equation is also called Gauss-Legendre-quadrature formula. In this the value of n sampling points can be used to fit (2n-1) degree variation. The Gauss points are selected such that a polynomial of (2n-1) degree is integrated exactly by employing n gauss points. In other words the error in the approximates are zero if the (2n+2) th derivative of the integrand vanishes.

The Gauss points are selected such that a polynomial of (2n-1) degree is integrated exactly by employing n gauss points. In other words the error in the approximates are zero if the (2n+2)th derivative of the integrand vanishes.

Numerical Integration

one Dimensional Analysis -- One Point Formula :

let $I = \hat{I} = {}_{xa}\int^{xb} f(x)\,dx = w_1 f(x_1) + w_2 f(x_2) + w_3 f(x_3) + \ldots w_i f(x_i)$ let $I = \hat{I} = {}_{xa}\int^{xb} f(x)\,dx = w_1 f(x_1)$ ....eq 1

for single point approximation w1 and x1 are unknowns.
since there are two unknowns, the integral must hold for f(x) = 1 and f(x) = x
for number of gauss points n = 1 and the order of the polynomial is (2n-1) = 1, it is linear.

Consider $\hat{I} = {}_{xa}\int^{xb} f(x)\,dx = w_1 f(x_1)$ ....eq 2

for f(x) = 1 $\;{}_{xa}\int^{xb} f(x)\,dx = xB - xA = w(1) = w1$ ....eq 3a for f(x) = x $\;{}_{xa}\int^{xb} f(x)\,dx = (xB^2 - xA^2) / 2 = w(x1) = w1x1$ ....eq 3b

solving these two eqs.

$xB - xA = w1 \quad (xB^2 - xA^2) / 2 = w1x1$

we get w1 = xB – xA

$x1 = (xB + xA) / 2$

……….eq 3c

Now Î = w1 f(x1) = (xB – xA) f [(xB + xA)/2] ………………eq 4

the integral is evaluated without regard to functional value at x = xA or x = xB, it can be got by knowing the functional value at a point representing the average of xA and xB

Numerical Integration -One Dimensional Analysis
One Point Formula Ç- co-ordinate system

In finite element method the elemental characteristics are in Ç coordinate system in case of one dimensional analysis. Ç varies from -1 to 1.

We have to transform the eqs. from x-axis to Ç-axis by linearly relating x to Ç.
Let x = a0+a1 Ç …………eq 5

the constants a0 and a1 are determined by using new limits of integration.
x = xA at Ç = -1   x = xB at Ç = 1 substituting these in to eq 5 and solving we get a0 = (xB + xA) / 2   and a1 = (xB – xA) / 2

Substituting these in eq 5 , x = [(xB + xA) / 2] + [(xB – xA) / 2] Ç ……eq 6 Differentiating this w,r,t Ç we get , dx = [(xB – xA) / 2] dÇ …….eq 7,
using eq 6 and 7
eq 1 $I = \int_{xa}^{xb} f(x)\,dx = w_1 f(x_1)$ can now be written as

$I = \int_{-1}^{+1}$ [(xB + xA) / 2] + f [(xB + xA)/2 + (xB – xA)/2]

dÇ…………..eq. 8 $I = \int_{-1}^{+1}$ f **(**Ç) dÇ ……eq. 9 , now from eq.

3c

w1 = xB – xA = 1-(-1) = 2 x1= (xB + xA)/2 = (1 – 1)/2 = 0

w1 = 2 Ç = 0 This is the transformation

The exact Integral and Gauss quadrature formula that involve single term can be related as $I = \int_{-1}^{+1} f(Ç)\, dÇ = \hat{I} = w1\, f(Ç1)$ ……eq 10

If the curve happens to be a straight line, the integral can be evaluated to sample f(0) at the middle point when Ç = 0, and multiply by the length of the interval as,

$I = \int_{-1}^{+1} f(0)\, dÇ = f(0)\,[1+1] = 2\, f(0)$

……….eq 11 ie w1 = 2 Ç1 = 0

There are two parameters w1 ,Ç1 , , we consider the formula represented by eq. 10 to be exact when f (Ç) is a polynomial of order (2n-1) =1 ie linear.

$f(Ç) = a0 + a1Ç$ …………eq.12 therefore

$I = \int_{-1}^{1} f(a0 + a1Ç)\, dÇ = 2a0$ ………..eq.13

we also have $\hat{I} = w1\, f(Ç1) = w1(a0 + a1\, Ç1)$ eq14

Error $\mathbf{e = I - \hat{I}}$ $\mathbf{e} = \int_{-1}^{1} f(Ç)\, dÇ - w1\, f(Ç1) = 2a0 - w1(a0+a1\, Ç1)$

$\mathbf{e} = a0(2 - w1) - a1\, w1Ç1$ ....eq 15

the error will be minimum if 6e /6a0 = 6e /6a1 = 0 … eq 16

6e /6a0 = 2-w1 = 0 w1 = 2

6e /6a1 = -w1Ç1 = 0 Ç1 = 0 therefore

$I = w1\, f(Ç1) = 2\, f(0)$……..eq 17

These are same as we

got earlier.

Two Point Formula Ç- co-ordinate system :

Consider Gauss-Legendre quadrature formula with sampling Gauss points n = 2

$I = \int_{-1}^{1} f(Ç)\, dÇ = w1\, f(Ç1) + w2\, f(Ç2) = \hat{I}$ ……eq 18

We have four parameters to select, therefore I will be exact when 1f (Ç) is a polynomial of order
3. (cubic polynomial. 2n-1 = 3)

1f (Ç) = a0+ a1 Ç+ a2 Ç2 + a3 Ç3 ……………..eq 19

I = $\int_{-1}^{1}$ (a0+ a1 Ç+ a2 Ç2 + a3 Ç3) dÇ = 2a0+ (2/3) a2

……………eq 20 Now consider equation 18 as
Î = w1 f **(**Ç1) + w2 f **(**Ç2)
= w1 (a0+ a1 Ç1+ a2 Ç12 + a3 Ç13) + w2 (a0+ a1 Ç2+ a2 Ç22 + a3 Ç23)

= a0 **(**w1+w2) + a1(w1 Ç1+ w2 Ç2) + a2(w1 Ç12 + w2 Ç22) + a3 (w1 Ç13 +

w2 Ç23)……. eq. 21 e = I – Î
e = [2a0+ (2/3) a2] – [a0 **(**w1+w2) + a1(w1 Ç1+ w2 Ç2)
+ a2(w1 Ç12 + w2 Ç22) + a3 (w1 Ç13 + w2 Ç23)]

the error will be zero if
6e /6a0 = 6e /6a1 = 6e /6a2 = 6e /6a3= 0

These yields w1+ w2 = 2 w1 Ç1+ w2 Ç2 = 0
w1 Ç12 + w2 Ç22 = 2/3 w1 Ç13 + w2 Ç23 = 0
These eqs. have the unique solution,
w1 = w2 = 2 Ç1 = - 1/√3 , Ç2 = 1/√3

Substituting these values in to
I = $\int_{-1}^{1}$f **(**Ç) dÇ = w1 f **(**Ç1) + w2 f **(**Ç2) we will get approximate answer.

Gauss-Legendre quadrature formula with sampling Gauss points n = 3

$\int_{-1}^{1}$f (Ç) dÇ = w1 f (Ç1) + w2 f (Ç2) + w3 f (Ç3) = Î ….eq 22
We have six parameters to select, therefore I will be exact when f (Ç) is a polynomial of order 5. (5th degree polynomial. 2n-1 = 5) .

f (Ç) = a0+ a1 Ç+ a2 Ç2 + a3 Ç3 + a4 Ç4 + a5 Ç5 ……..eq 23

I = $\int_{-1}^{1}$ (a0+ a1 Ç+ a2 Ç2 + a3 Ç3 + a4 Ç4 + a5 Ç5) dÇ

I = 2a0+ (2/3) a2 + (2/5) a4

……………………………eq 24 f(Ç1) = a0+ a1

Ç1+ a2 Ç12 + a3 Ç13 + a4 Ç14 + a5 Ç15

f(Ç2) = a0+ a1 Ç2+ a2 Ç22 + a3 Ç23 + a4 Ç24

+ a5 Ç25

f(Ç3) = a0+ a1 Ç3+ a2 Ç32 + a3 Ç33 + a4 Ç34 + a5 Ç35

Substituting these in to (eq.22)

Î = $_{-1}\int^{1}$ f **(**Ç) dÇ = w1 f **(**Ç1) + w2 f **(**Ç2) + w3 f **(**Ç3) we get Î = w1(a0+ a1 Ç1+ a2 Ç12 + a3 Ç13 + a4 Ç14 + a5 Ç15)

+ w2 (a0+ a1 Ç2+ a2 Ç22 + a3 Ç23 + a4 Ç24 + a5 Ç25)

+ w3 (a0+ a1 Ç3+ a2 Ç32 + a3 Ç33 + a4 Ç34 + a5 Ç35)

Simplifying the eqn. we get

Î = a0 **(**w1+w2+w3) + a1(w1 Ç1+ w2 Ç2 + w3 Ç3)
+ a2(w1 Ç12 + w2 Ç22 + w3 Ç32) + a3 (w1 Ç13 + w2 Ç23 + w3 Ç33)
+ a4(w1 Ç14+ w2 Ç24 + w3 Ç34) + a5 (w1 Ç15 + w2 Ç25 + w3 Ç35)

……………….eq. 25

e = I – Î

e = [2a0+ (2/3) a2 +(2/5) a4]
– [a0 **(**w1+w2+w3) + a1(w1 Ç1+ w2 Ç2 + w3 Ç3)
+ a2(w1 Ç12 + w2 Ç22 + w3 Ç32) + a3 (w1 Ç13 + w2 Ç23 + w3 Ç33)
+ a4(w1 Ç14+ w2 Ç24 + w3 Ç34) + a5 (w1 Ç15 + w2 Ç25 + w3 Ç35)]

…………….eq. 26

the error will be zero if
6e /6a0 = 6e /6a1 = 6e /6a2 = 6e /6a3= 6e /6a4 = 6e /6a5 = 0

These yields w1+ w2+ w3 = 2 w1 Ç1+ w2 Ç2 + w3 Ç3 = 0
w1 Ç12 + w2 Ç22 + w3 Ç32 = 2/3 w1 Ç13 + w2 Ç23 + w3 Ç33 = 0 w1 Ç14 + w2 Ç24 + w3 Ç34 = 2/5 w1 Ç15 + w2 Ç25 + w3 Ç35 = 0

………...eq.27

These eqs. have the unique solution,

Ç1 = - √0.6= -0.774596692 Ç2 = 0.00000000

Ç3 = √0.6 = 0.774596692

w1 = w3 = 5/9 = 0.555555555

w2 = 8/9 = 0.888888888

Substituting these values in to $I = \int_{-1}^{1} f(Ç)\, dÇ = w1\, f(Ç1) + w2\, f(Ç2) + w3\, f(Ç3)$

we will get approximate answer

Two and Three Dimensional Analysis, Ç, y - co-ordinate system :

Quadrilateral Plane elements and Hexahedral solid elements :

In these cases we apply the one dimensional integration formulas successively in each direction. Similar to the analytical evaluation of double or triple integral, successively the innermost integral is evaluated by keeping the variables corresponding to other integrals constant.

For Quadrilateral Plane region the Gauss Quadrature formula is given by

$$I = \int_{-1}^{1}\int_{-1}^{1} f(Ç,y)\, dÇ\, dy = \hat{I} = \Sigma^{n}_{i=1} \Sigma^{n}_{j=1} wi\, wj\, f(Çi,yj) \quad ……eq\ 28$$

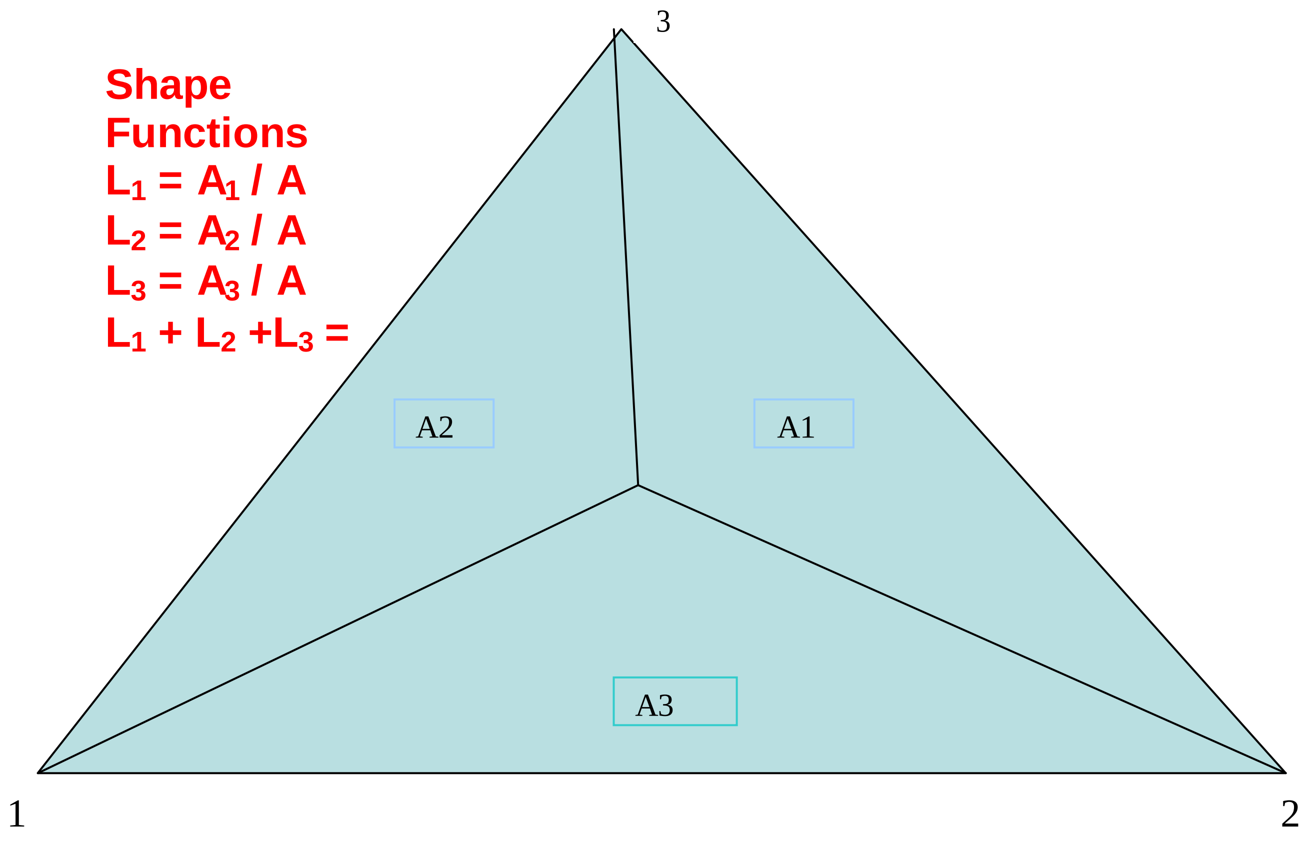
For Hexahedral Solid region the Gauss Quadrature formula is given by

$$I = \int_{-1}^{1}\int_{-1}^{1}\int_{-1}^{1} f(Ç,y,Ç)\, dÇ\, dy\, dÇ = \Sigma^{n}_{i=1} \Sigma^{n}_{j=1} \Sigma^{n}_{k=1} w_i w_j w_k f(Ç_i, y_j, Ç_k) \quad \text{......eq 29}$$

The method discussed so for cannot be applied for Triangular and Tetrahedral solid regions.

In case of Triangular plane and Tetrahedral solid regions the limits integration involve variables. The Integrals are expressed using Area coordinates instead of natural coordinates and integration is carried out

We know the area co-ordinates of CST, consider an arbitrary point p(x,y) in the triangle. Join all the corners of the triangle to the point. The area of the triangle is divided in to three parts A1 A2 A3 . A is the over all area



If A is the total area of the triangle then . L1 = A1 / A , L2 = A2 / A , L3 = A3 / A are called the area co-ordinates of point p(x,y),

L1 = 1 at node 1 and 0 at node 2 and 3

L2 = 1 at node 2 and 0 at node 1 and 3

L3 = 1 at node 3 and 0 at node 2 and 1

They are shape functions in terms of area co-ordinates.

$I = {}_{A}\int f(L1, L2, L3)\, dA$

$I = {}_{0}\int^{1} {}_{0}\int^{1-L1} f(L_1, L_2)\, dL_1\, dL_2$

$= \hat{I} = {}_{i=1}\Sigma^{n} wi\, f(L^{i}_{1}, L^{i}_{2}, L^{i}_{3})$ ($L1 = L_1$ etc)

This is the Gauss Quadrature formula , i is the location of Gauss points. Tables are available which give the Gauss points and weights for linear , qradratic and cubic triangular planes

Also In the case of Tetrahedral solid regions the limits of integration involve variables. The Integrals are expressed using Volume coordinates instead of natural coordinates and integration is carried out. The Volume co-ordinates of a Tetrahedron is similar to area coordinates of CST and can be explained as follows. Consider an arbitrary point p (x,y) inside the tetrahedron, Join all the corners of the tetrahedron to this point. The volume of the triangle is divided in to four parts v1 v2 v3 v4.

If v is the total volume of the tetrahedron then , L1 = v1 / v , L2 = v2 / v, L3 = v3 / v, L4 = v4 / v , are called the volume co-ordinates of point p(x,y) . v1 is the volume p234, v2 is the volume of p134 , v3 is the volume p124 , v4 is the volume p123. The Integral of the tetrahedral solid is given by

The Integral of the tetrahedral solid is given by

$I = {}_{v}\int f(L1, L2, L3, L4)\, dv$

$= {}_{0}\int^{1} {}_{0}\int^{1-L1} {}_{0}\int^{1-L1-L2} f(L1, L2, L3)\, dL1\, dL2\, dL3$

$\hat{I} = \Sigma^{n}_{i=1} w\, i\, f(L^{i}_{1}, L^{i}_{2}, L^{i}_{3}, L^{i}_{4})$

Tables are available that gives the Gauss points and weights for the linear, quadratic and tetrahedral solids.

Prob1 : Evaluate the following using one point and two point Gauss quadrature

$I = {}_{-1}\int^{1} [\, 3\, e^{Ç} + Ç^{2} + 1/(Ç+2)\,]\, dÇ$

One point formula : for n = 1 we have w1 = 2 and Ç1 = 0

I = $_{-1}\int^{1}$ f (Ç) dÇ = w1 f (Ç1) f (Ç1) = f (0) = [3 e^{0} + 0 ^{2} + 1/ (0+2)] = 3.5

I = w1 f (Ç1) = 2 x 3.5 = 7

Two point formula : for n = 2 we have w1 = w2 = 1 and Ç1 = - 0.57735

Ç2 = + 0.57735

I = $_{-1}\int^{1}$ f (Ç) dÇ = w1 f (Ç1) +

w2 f (Ç2) f (Ç1) = f (- 0.57735)

= [3 $e^{-0.57735}$ + $(-0.57735)^{2}$ + 1/ (- 0.57735 +2)] =2.720

f (Ç2) = f (+0.57735)

= [3 $e^{0.57735}$ + $(0.57735)^{2}$ + 1/ (0.57735 +2)] = 6.065

I = w1 f (Ç1) + w2 f (Ç2) = 1(2.720) + 1(6.065) = 8.785 Ans the exact

solution is 8.815 Note : For better accuracy minimum six decimal digits

should be used in weight functions and
sampling points

In the above discussion the sampling points and weight functions Çi , wi are considered only for natural interval from (-1 to 1). However to make the calculations general the sampling points and weight functions for any interval from (a to b) are given by Çi**!** , wi**!** where

Çi**!** = [(a+b)/2 + ((b-a)/2)Çi] wi**!** = ((b-a)/2) wi

Prob2 : Evaluate using two point Gauss quadrature, I = I

= $_{0}\int^{3}$ ($2^{Ç}$ - Ç) dÇ Solution : For the natural interval (-1 to 1)

for two point gauss quadrature

n = 2 w1 = w2 = 1 and Ç1 = - 0.57735 Ç2 = + 0.57735

For the interval (0 to 3) using the formulae

Ç 1 **!** = [(a+b)/2 + ((b-a)/2)Ç 1] Ç 2 **!** = [(a+b)/2 + ((b-a)/2)Ç 2]

w 1 **!** = ((b-a)/2) w 1 w 2 **!** = ((b-a)/2) w 2 we have

Ç1**!** = [(3+0)/2 + ((3-0)/2)(-0.57735)] = 0.633975

Ç2**!** = [(3+0)/2 + ((3-0)/2)(0.57735)] = 2.36602

w1**!** = ((3-0)/2) (1) = 3/2 w2**!** = ((3-0)/2) (1) = 3/2

f (Ç) = ($2^{Ç}$ - Ç) using Ç1**!** **,** Ç2**!** In place of Ç

f (Ç1**!**) = (20.633975 – 0.633975) = 0.9178

f (Ç2**!**) = (22.36602 – 2.36602) = 2.789

Î = w1! f (Ç1!) + w2! f (Ç2!) = (3/2) (0.9178) + (3/2) (2.789) = 5.56

Prob3 : Using two point Gaussian quadrature formula evaluate the following integral

$$I = \int_{-1}^{1}\int_{-1}^{1} f(Ç,y)\, dÇ\, dy = Î = \Sigma^{n}_{i=1} \Sigma^{n}_{j=1} w_i w_j f(Ç_i, y_j) \quad \ldots\ldots eq\ 28$$

$$I = \int_{-1}^{1}\int_{-1}^{1} [Ç^2 + 2Çy + y^2]\, dÇ\, dy$$

Solution :The above integral can be expressed in general form as

$$I = \int_{-1}^{1}\int_{-1}^{1} f(Ç,y)\, dÇ\, dy = Î = \Sigma^{n}_{i=1} \Sigma^{n}_{j=1} w_i w_j f(Ç_i, y_j)$$

n = 2 in both Ç and y direction. Expanding the above equation

Î = w1[w1 f (Ç1,y1) + w2 f (Ç1,y2)] + w2[w1 f (Ç2,y1) + w2 f (Ç2,y2)]
= w12 f (Ç1,y1) + w1w2 f (Ç1,y2) + w2w1 f (Ç2,y1) + w22 f (Ç2,y2)]

•

Î = w1[w1 (Ç12 + 2Ç1y1 +y12) + w2 (Ç12 + 2Ç1y2 +y22)]
+ w2[w1 (Ç22 + 2Ç2y1 +y12) + w2 (Ç22 + 2Ç2y2 +y22)]

From table we get the values of Gauss points and weights for two point Gauss quadrature formula as Ç1 = y1 = - 1/√3 Ç2 = y2 = 1/√3 w1 = w2 = 1, substituting these in the above equation we get,

Î = 1[1 ((- 1/√3)2 + 2(- 1/√3) (- 1/√3) +(- 1/√3)2)
+ 1 ((- 1/√3)2 + 2(- 1/√3) (1/√3) +(1/√3) 2)]
+ 1[1((1/√3) 2 + 2 (1/√3) (- 1/√3) + (- 1/√3)2)
+ 1 ((1/√3) 2 + 2(1/√3)(1/√3) +(1/√3) 2)]

on simplification gives Î = 8/3

Prob 4 : evaluate the integral $\int_{-1}^{1} (2 + x + x^2)dx$

Solution : we need at least two point integration rule since the integrand contain a quadratic term. We will use both one point and two point guass quadrature and show that two point result matches with the exact solution.

For one point rule we know x1 = 0 w1 = 2

Consider $\hat{I} = \int_{-1}^{1} f(x)\,dx = w1 f(x1)$ ………eq 2

$= 2(2+0+0) = 4$

For two point rule we know $x1 = 1/\sqrt{3}$ $x2 = -1/\sqrt{3}$ $w1 = w2 = 1$

$I = \int_{-1}^{1} f(x)\,dx = w1\, f(x1) + w2\, f(x2)$

$= 1\,(2 + 1/\sqrt{3} + (1/\sqrt{3})2) + 1\, f(2 - 1/\sqrt{3} + (1/\sqrt{3})2)$

= 4.6667 The exact solution is 4.6667

Prob 5 : evaluate the integral $\int_{-1}^{1}$ **cos** (nx/2) dx

Solution :**For one point rule we know x1 = 0 w1 = 2**

$\hat{I} = \int_{-1}^{1} f(x)\,dx = w1 f(x1) = 2\,[\,\cos(n\,0/2)\,] = 2$

For two point rule we know $x1 = 1/\sqrt{3}$ $x2 = -1/\sqrt{3}$ $w1 = w2 = 1$

$I = \int_{-1}^{1} f(x)\,dx = w1\, f(x1) + w2\, f(x2)$

$= 1 \cos[(n/2)(1/\sqrt{3})] + 1 \cos[(n/2)(-1/\sqrt{3})]$

$= 2\cos(n/2\sqrt{3})$ = **1.232381**

For three point rule we know $x1 = \sqrt{0.6}$ $x2 = 0$ $x3 = -\sqrt{0.6}$

$w1 = 5/9$ $w2 = 8/9$ $w3 = 5/9$

$I = \int_{-1}^{1} f(x)\,dx = w1\, f(x1) + w2\, f(x2) + w3\, f(x3)$

$= 5/9 \cos(n/2)(\sqrt{0.6}) + 8/9 \cos(n/2)(0) + 5/9 \cos(n/2)(-\sqrt{0.6})$

= 1.274123754

- **For Four point rule** we know
-

x1 = 0.8611363 x2 = 0.3399810x3 = -0.3399810 x4 = -0.8611363

w1 = 0.347854845 w2 = 0.652145155 w3 =

0.652145155 w4 = 0.347854845

$I = \int_{-1}^{1} f(x)\,dx = w1\,f(x1) + w2\,f(x2) + w3\,f(x3) + w4\,f(x4)$

= 0.347854845 **cos** (n/2)(0.8611363) + 0.652145155 **cos** (n/2)(0.3399810)
+ 0.652145155 **cos** (n/2)(- 0.3399810) + 0.347854845 **cos** (n/2)(- 0.8611363)

= 1.273229508

The actual answer is = 1.273239544

Determination of Stiffness Matrix [K] for quadrilateral element :

Elemental strain energy is given by $\frac{1}{2}\int o^T s\,dv$

o = D B q where D is 3x3 matrix

$U = \Sigma\, t_e \int e\, \frac{1}{2}\, o^T s\, dA = \frac{1}{2}\, te \int e\, [D\,B\,q]^T\, B\,q\,dA$

$= \frac{1}{2}\, q^T\, [t_e \int_e B^T\, D\, B\, dA]\, q$

$U = \Sigma\, \frac{1}{2}\, q^T\, [\, t_e \int\int B^T D\, B\, \det J\, dÇ\, dy\,]\, q$

$= \Sigma\, \frac{1}{2}\, q^T\, [\, k^e\,]\, q$

where $k^e = t_e \int\int B^T DB \det J\, dÇ\, dy$ is the element stiffness matrix

B and det J are involved functions of Ç & y, and so the integration has to be performed numerically. The element stiffness matrix is (8 x 8)

Integration has to be carried out to determine only upper triangular elements , Let 0 is the ij th element of the integrand above

O(Ç,y) = te ($B^T DB$ det J) i j Using 2 x 2 rule we get (4 point integration)

K i j = w12 **O**(Ç1,y1) + w1w2 **O**(Ç1,y2) + w2w1 **O**(Ç2,y1) + w22 **O**(Ç2,y2)

w1 = w2 = 1.0 Ç1 = y1 = - 0.57735 Ç2 = y2 = 0.57735

(look in to prob 3 solved using 2 x 2 rule),

$K\,i\,j = \Sigma\, w_{IP}\, 0_{IP}$ $_{IP\ \ 1\ to\ 4}$ IP is the integration point

where 0_{IP} is the value of 0 and w_{IP} is the weight factor at integration point IP. [$B^T DB$ det J] results in 8x8 matrix, containing 64 terms which are symmetric about principal diagonal. Each term is a function of Ç & y, on integration give kij ‘s Numerical integration is carried out considering the gauss points 1,2,3,4 as indicated above.

k11 = w12 011 (Ç1,y1) + w1w2 011 (Ç1,y2) + w2w1 011 (Ç2,y1) + w22 011 (Ç2,y2) k12= w12 012 (Ç1,y1) + w1w2 012 (Ç1,y2) + w2w1 012 (Ç2,y1) + w22 012 (Ç2,y2) k13 = w12 **O**13 (Ç1,y1) + w1w2 **O**13 (Ç1,y2) + w2w1 **O**13(Ç2,y1) + w22 **O**13 (Ç2,y2)

………..

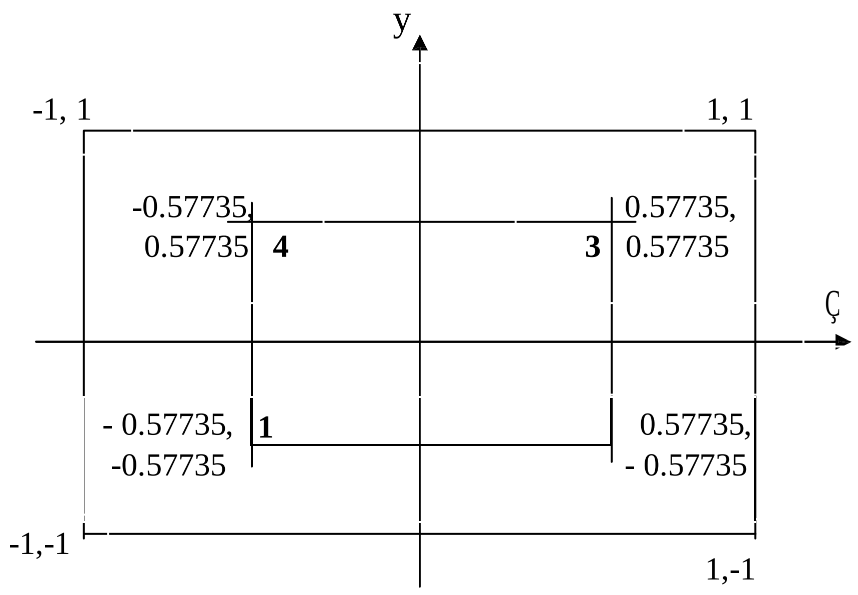
………..

………..

k88 = w12 **O**88 (Ç1,y1) + w1w2 **O**88 (Ç1,y2) + w2w1 **O**88(Ç2,y1) + w22 **O**88 (Ç2,y2)

Ç , y

Thus No General k matrix is developed in case of quadrilateral. Elements of k depends on gauss points. Some time elements of stiffness matrix [k] are determined at mid point of the quadrilateral where Ç = y = 0. This is a simplified procedure and results in relatively lesser data to handle.



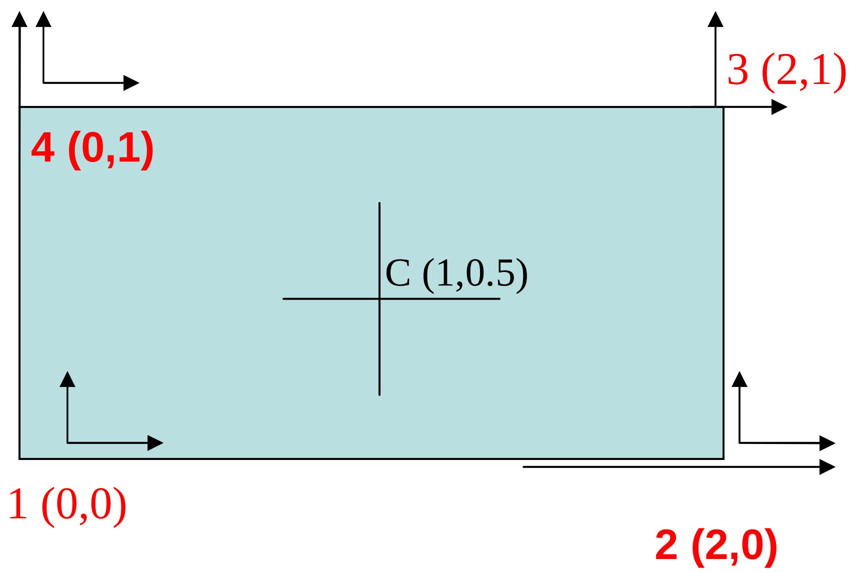
Stress Calculations in Quadrilateral element : o = DBq

o = DBq is not constant within the element. They are functions of Ç , y and consequently vary within the element. In practice the stresses evaluated at Gauss points, which are also the points used for numerical evaluation of k_e, where they are found be accurate. For a quadrilateral with 2 x 2 integration this gives four sets of stress values. For generating less data one may evaluate stresses at one point per element, say at Ç = y = 0 . Many Computer schemes use this approach

Problems on Quadrilateral elements

Consider the rectangular element shown in fig. Assume plane stress conditions, E = 30 x 106 psi, v = 0.3, and q = [0, 0, 0.002, 0.003, 0.006, 0.0032, 0, 0]T in. Evaluate J, B and o at Ç = y = 0

Solution : at node 1 ,2,3, 4 displacement components are q_1 q_2 q_3 q_4 q_5 q_6 q_7 q_8 in usual direction .



(x1,y1) = (0 , 0) (x2,y2) = (2 , 0) (x3,y3) = (2 , 1)
(x4,y4) = (0 , 1) (xc ,yc) = (1, 0.5)
Gauss points Ç = y = 0 q = [0, 0, 0.002, 0.003, 0.006, 0.0032, 0, 0 $]^T$ in.

first let us determine J

J11 = ¼ [- (1 – y) x1 + (1 – y) x2 + (1 + y) x3 – (1+ y) x4]
J12 = ¼ [- (1 – y) y1 + (1 – y) y2 + (1 + y) y3 – (1+ y) y4]
J21 = ¼ [- (1 – Ç) x1 - (1 + Ç) x2 + (1 + Ç) x3+ (1- Ç) x4]
J22 = ¼ [- (1 – Ç) y1 - (1 + Ç) y2 + (1 + Ç) y3+
(1- Ç) y4] x1,y1 = 0 , 0 x2,y2 = 2 , 0 x3,y3 = 2 , 1
x4,y4 = 0 , 1 Ç = y = 0

J11 = ¼ [- 0 + 2(1 – y) + 2(1 + y) – 0] = ¼ [2(1 – y) + 2(1 + y)] = 1 J11 = 1

J12 = ¼ [- 0 + 0 + (1 + y) – (1+ y)] = ¼ [(1

+ y) – (1+ y)] = 0 J12 = 0 J21 = ¼ [- 0 - 2(1 + Ç) +

2(1 + Ç) + 0] = ¼ [- 2(1 + Ç) + 2(1+ Ç) = 0

J21 = 0 J22 = ¼ [- 0 - 0 + (1 + Ç) + (1- Ç)] = ¼ [(1 +

Ç) + (1- Ç)] = ½ J22 = ½

J_{11} = 1 J_{12} = 0 J_{21} = 0 J_{22} = ½

[J] is a constant Matrix			
Also |J| = ½			
J	=	1	0
		0	½

Also |J| = ½

A	=	1/ | J |	J_{22}	$-J_{12}$	0	0
			0	0	$-J_{21}$	J_{11}
			$-J_{12}$	J_{11}	J_{22}	$-J_{12}$

1/ (½) = 2	½	0	0	0
	0	0	0	1
	0	1	½	0

[A]			
1	0	0	0
0	0	0	2
0	2	1	0

This is [G] substitute Ç = y = 0								
	-(1-y**)**	**0**	**(1-**y**)**	**0**	**(1**+y**)**	**0**	**-(1**+y**)**	**0**
	-(1- Ç**)**	**0**	**-(1**+Ç**)**	**0**	**(1**+Ç**)**	**0**	**(1-**Ç**)**	**0**
1/4	**0**	**-(1-**y**)**	**0**	**(1-**y**)**	**0**	**(1**+y**)**	**0**	**-(1**+y**)**
	0	**-(1-** Ç**)**	**0**	**-(1**+Ç**)**	**0**	**(1**+Ç**)**	**0**	**(1-**Ç**)**

mbnc

This is [G] substituting Ç = y = 0							
-1/4	0	1/4	0	1/4	0	-1/4	0
-1/4	0	-1/4	0	1/4	0	1/4	0
0	-1/4	0	1/4	0	1/4	0	-1/4
0	-1/4	0	-1/4	0	1/4	0	1/4

The stresses at Ç = y = 0 are now given by $o^{0} = D B^{0} q$

[D]			
30 x 106 / (1-.09) =32.96x106	1	0.3	0
	0.03	1	0
	0	0	0.35

[B] ^{0} = [A] [G]							
-1/4	0	1/4	0	1/4	0	-1/4	0
0	-1/2	0	-1/2	0	1/2	0	1/2
-1/2	-1/4	-1/2	1/4	1/2	1/4	1/2	-1/4

[q]
0
0
0.002
0.003
0.006
0.0032
0
0

$[D][B]^0 = 10^6$

$30 \times 10^6 / (1-0.09) = 32.96 \times 10^6$ 32.96x0.3 = 9.890 32.96x0.03 = 0.9888

32.96x0.35 = 11.53 32.96x0.25 = 8.24 32.96x0.5 = 16.48

9.890 x 0.5 = 4.945 0.988 x 0.25 = 0.2472 11.53 x 0.5 = 5.765

11.53 x 0.25 = 5.765

-8.24	-4.945	8.24	-4.945	8.24	4.945	-8.24	4.945
-0.2472	-16.48	0.2472	-16.48	0.2472	16.48	-0.2472	16.48
-5.765	-2.883	-5.765	2.883	5.765	2.883	5.765	-2.883

The stresses at Ç = y = 0 are now given by $o^0 = D B^0 q$

[D] [B0] = 10 ^{6} x							
-8.24	-4.945	8.24	-4.945	8.24	4.945	-8.24	4.945
-0.2472	-16.48	0.2472	-16.48	0.2472	16.48	-0.2472	16.48
-5.765	-2.883	-5.765	2.883	5.765	2.883	5.765	-2.883

[q]
0
0
0.002
0.003
0.006
0.0032
0
0

10^6 [8.24*0.002 - 4.945 * 0.003 + 8.24*0.006+ 4.945* 0.0032]

= $0.066909*10^6$ = 66909 psi

10^6 [0.2472*0.002 – 16.48 * 0.003 + 0.2472*0.006 + 16.48*0.0032]

= 23080 psi (5273.6 psi)

10 6 [-5.765*0.002 +2.883* 0.003 + 5.765*0.006 + 2.883* 0.0032]

= 0.040905 * 106 = 40905

[o ^{0}] = [66909 , 23080, 40905] ^{T} psi

Higher Order Elements :

The four Node quadrilateral studied so for have shape functions containing the terms 1, Ç & y etc which are linear terms. Elements having shape functions containing $Ç^2$ y, $Ç^2$ y^2 and Ç y^2 etc are called Higher order elements. They have middle nodes along with corner nodes or other normal nodes. They provide greater accuracy in analysis

Determination of [k] for higher order elements follow the routine steps :

u = Nq s = B q $k^e = t_e \int\int B^T DB$ det J dÇ dy k^e is evaluated at gauss points etc.

Nine Node Quadrilateral , Eight Node Quadrilateral , Six Node Triangle are the Higher order elements used in 2-d analysis. The shape functions are derived using Lagrange shape function formula

(Ç- Ç0)(Ç- Ç1)…… (Ç- Çp-1)(Ç- Çp+1)….. (Ç- Çn)

------------- (Çp- Ç0)(Çp - Ç1)…… (Çp- Ç p-1)(Çp- Çp+1)….. (Çp- Çn)

The shape functions are also determined using Serendipity approach, assuming a polynomial of suitable order (depending on degrees of freedom), determining the values of constants using boundary conditions and other mathematical constraints specific to certain analysis and geometry, etc,.

Nine node quadrilateral :

The Element is a Quadrilateral consisting of Four Corner Nodes and Four Middle Nodes and a Node at the center of the element total Nine Nodes. Shape functions can be defined in local coordinates using serendipity approach. We use a master quadrilateral to define N's. consider Ç- axis alone with local nodes 1,2,3 with Ç = -1, 0, 1. , L1 , L2 L3 are generic shape functions with usual definition L1 (Ç) = 1 at node 1 and 0 at other two nodes etc, Consider L1 = 0 at Ç = 0 and at Ç = +1 , hence it should be of the form L1 = c Ç (1- Ç) Since L1 =1 at Ç = -1 we get c = - ½ , thereforeL1 (Ç) = - Ç (1- Ç) / 2

Using similar argument L2 (Ç) = (1+ Ç) (1- Ç) L3 (Ç) = Ç (1+ Ç) / 2

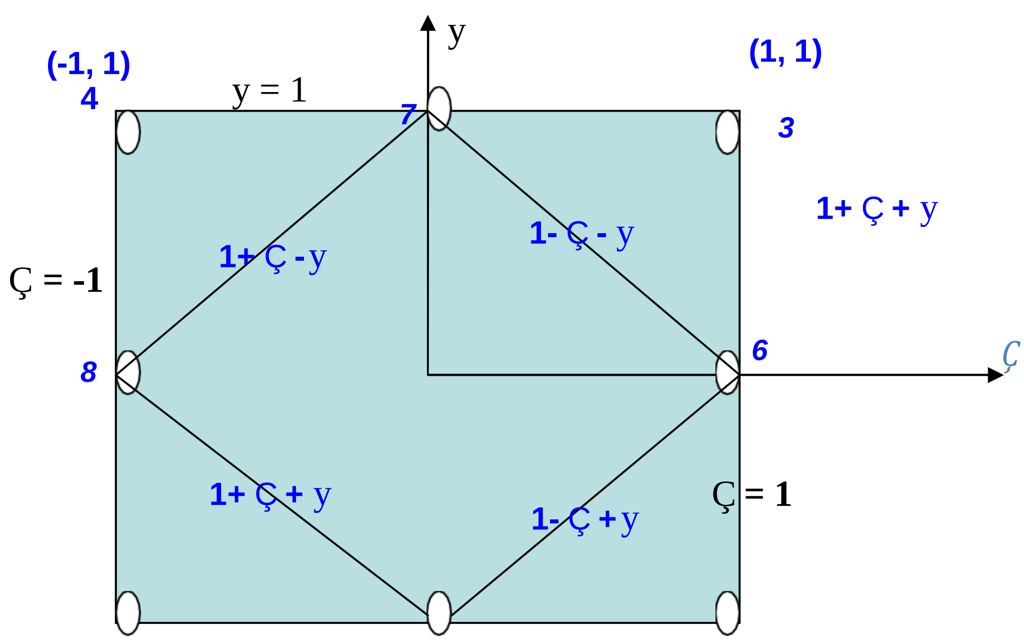
Similarly along y axis we have, L1 (y) = - y (1- y) / 2 L2(y) = (1- y) (1+y) L3 (y) = [y (1+y) / 2]

In the master quadrilateral element every node has the coordinates Ç = -1, 0 or +1 y = -1, 0 or +1 , thus the following product rule give the shape functions as ,

N1 = L1 (Ç) L1 (y) N5 = L2 (Ç) L1 (y) N2 = L3 (Ç) L1 (y)
N8 = L1 (Ç) L2(y) N9 = L2 (Ç) L2 (y) N6 = L3 (Ç) L2 (y)
N4 = L1 (Ç) L3 (y) N7 = L2 (Ç) L3 (y) N3 = L3 (Ç) L3 (y)

Higher order terms in N leads to higher order interpolation of displacement as given by u = Nq Higher order terms can also be used to define geometry. This leads to quadrilateral having curved edges if required. [x] = [N] [x] [y] = [N] [y] . Any how sub parametric formulation can also be adopted using nine node shape functions to interpolate displacement and four node shape function to define geometry.

Shape functions of a Eight node quadrilateral :



1
(-1,-1)

y =-1

5

2

(1,-1)

Eight node master quadrilateral

The Element is a Quadrilateral consisting of Four Corner Nodes and Four Middle Nodes, total Eight Nodes. Shape functions can be defined in local coordinates using serendipity approach.
We use a master quadrilateral to define N's.

Ni = 1 at node I and 0 all other nodes. Thus N1 has to vanish along lines Ç = +1 & y = +1 Thus N1 is of the form N1 = c(1- Ç) (1- y) (1+ Ç+ y).

At node 1 N1 = 1, Ç = y = -1 1 = c(1+1) (1+1)(1-1-1) = - 4c , thus c = -1/4
Therefore N1 = - ¼ [(1+ Ç) (1- y) (1+ Ç+ y)] , similarly N2 , N3 , N4 are determined .

N1 = - [(1+ Ç) (1- y) (1+ Ç+ y)] / 4, N2 = - [(1+ Ç) (1- y) (1- Ç+ y)] / 4
N3 = - [(1+ Ç) (1+ y) (1-Ç- y)] / 4, N4 = - [(1- Ç) (1+ y) (1+Ç- y)] / 4

N5 N6 , N7 , N8 are determine at mid points

N5 vanishes along the edges Ç = +1, y = +1 , Ç = -1, hence it has to be of the form N5 = c(1- Ç) (1- y) (1+ Ç) , = c(1- $Ç^2$) (1- y)
we have the condition N5 = 1 at node 5
or N5 = 1 at Ç = 0, y= -1 1 = c(1- $Ç^2$) (1- y) = c (1) (2) c = ½

Thus N5 = ½ [(1- $Ç^2$) (1- y)] , similarly remaining can

be determined. N5 = [(1- Ç2) (1- y)] / 2 N6 = [(1+

Ç) (1- y2)] / 2
N7 = [(1- Ç2) (1+ y)] / 2 N8 = [(1- Ç) (1- y2)] / 2

Shape functions of a Six node Triangle : Ç = 1 – Ç – y.

The Element is a triangle consisting of Three Corner Nodes and Three Middle Nodes, total Six Nodes. Shape functions can be defined in local coordinates using serendipity approach.. We use a master Triangle to define N's.
Ni = 1 at node 1 and 0 all other nodes etc . N1 = Ç (2Ç – 1)

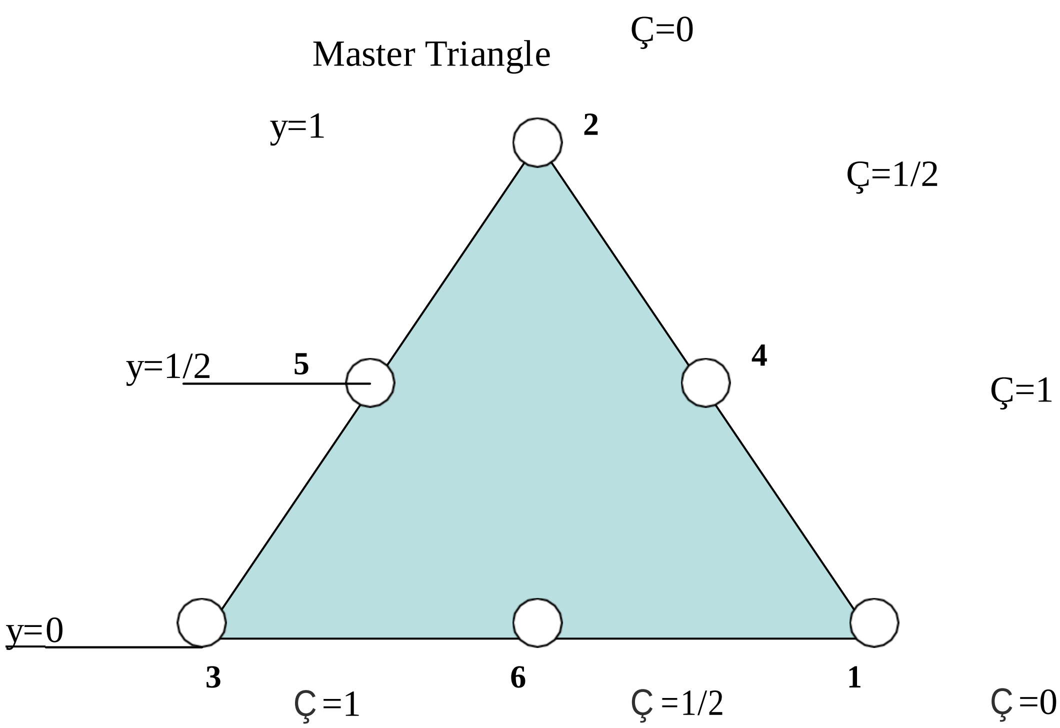
N2 = y (2 y– 1) N3 = Ç (2 Ç– 1) N4 = 4Ç y
N5 = 4 Ç y N6 = 4Ç Ç

Since terms $Ç^2$, y^2 are also present the triangle is also called quadratic triangle.

Iso parametric representation is u = N q , x = Σ Ni xi y = Σ Ni yi

[k] has to be got by numerical integration ke = te ∫ ∫ B^TDB det J dÇ dy

One point rule at the centroid with w1= ½ and the gauss points Ç1 = y1 = Ç1 = 1/3 is used Other choices of wi and Gauss points are available in the table.



Shape functions of Iso parametric Linear Bar Element :

Element characteristics of iso parametric elements are derived using natural coordinate system Ç defined by element geometry and not by the element orientation in the global-coordinate system. That is, axial coordinate is attached to the bar and remains directed along the axial length of the bar, regardless of how the bar is oriented in space.

Consider a two node, linear bar element having two degrees of freedom, axial deformations Ui and Uj at nodes i and j, associated with the global x-coordinate as shown in figure

Consider the displacements field u(Ç) to the nodal displacement Ui and Uj using a linear polynomial   u(Ç) = a1 + a2 Ç , where Ç is natural coordinates and vary from -1 to +1, a1 and a2 are generalized coordinates and can be determined from the following nodal conditions.

At, Ç = -1, u(-1) = Ui   and   Ç = +1; u(1) = Uj
By substituting above conditions into equation ,we obtain Ui = a1 + a2 (-1)   Ui = a1 - a2
Uj = a1 + a2 (1)   Uj = a1 + a2

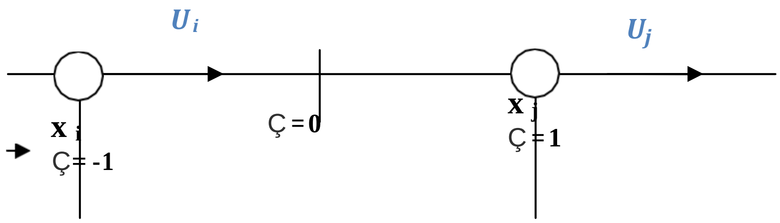
adding both we get Ui + Uj = 2a1 therefore a1 = (Ui + Uj)/2

Subtracting we get Ui -Uj = - 2a2 therefore a2 = - (Ui - Uj)/2 a2 = (Uj - Ui)/2

u(Ç) = (Ui + Uj)/2 + [(Uj - Ui)/2] (Ç)
= (Ui - Ui Ç) /2 + (Uj + Uj Ç)/2
= Ui (1- Ç) /2 + Uj (1+ Ç)/2
= [(1- Ç)/ 2]*ui + [(1+Ç)/ 2]*uj or   [(1- Ç)/ 2]*q1+ [(1+Ç)/ 2]*q2

Thus u(Ç) = Ni (Ç)*ui+Nj (Ç)*uj or = N1q1 + N2q2 Ni (Ç) = N1 = (1- Ç)/ 2
Nj (Ç) = N2 = (1+Ç)/ 2
Are called the shape functions

At node 1 Ç = -1 N1 = 1 At node 2 Ç = 1 N1 = 0
At node 2 Ç = 1 N2 = 1 At node 1 Ç = -1 N2 = 0

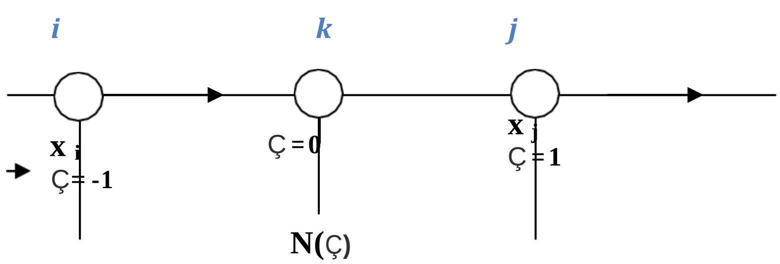


Consider a three noded bar element as shown in figure below. Let i and j be the end nodes and k be the middle node. The element is defined in natural coordinate system. The shape functions can be derived either by using the displacement polynomial of order two or the Lagrange shape function formula.
Let Ni, Nj and Nk be the shape functions of nodes i , j and k respectively. Let Çi, Çj and Çk be the nodal coordinates defined in the natural coordinate system figure below Using the Lagrange shape function formula for one-dimensional element we obtain the shape function Ni of node i as

$$Ni(Ç) = [(Ç- Çk)(Ç-Çj)] / [(Çi -Çk)(Çi -Çj)]$$

Introducing Çi = -1, Çk = 0, Çj = +1 into above expression, we obtain

$Ni(Ç) = [(Ç- 0)(Ç-1)] / [(-1 -0)(-1 -1)] =$
$Ç(Ç-1) / 2$ $Ni(Ç) = [Ç(Ç-1) / 2]$



Similarly we obtain the shape functions Nk and Nj of nodes k and

j respectively as $Nk(Ç) = [(Ç- Çi)(Ç-Çj)] / [(Çk –Çi)(Çk –Çj)] =$

$(1- Ç^2)$

$= [(Ç+1)(Ç-1)] / [(1)(-1)] = [(1- Ç)(1+Ç)] /$

$[(1)(1)] = (1- Ç^2)$ $Nk(Ç) = (1- Ç^2)$

$Nj(Ç) = [(Ç- Çi)(Ç-Çk)] / [(Çj –Çi)(Çj –Çk)] = [Ç(Ç+1) / 2]$

$[(Ç- (-1)(Ç-0)] / [(1 – (-1)) (1 –0)] =$

$[Ç(Ç+1) / 2]$ $Nj(Ç) = [Ç(Ç+1) / 2]$

$N1 (Ç) = Ç(Ç-1) / 2$ $N2 (Ç) = [Ç(Ç+1) / 2]$ $N3 (Ç) = (1- Ç^2)$

Isoparametric Linear Triangular Element :

For an actual or generalized physical element, in a physical space, natural coordinate axes need not be orthogonal or parallel to the global coordinate axes. The natural coordinates are attached to the element and maintain their position with respect to it regardless of the element orientation in global coordinates. Also an element's physical size and shape have no effect on the numerical values of reference coordinates at which nodes appear. Thus, physical elements of various sizes and shapes are all mapped in to the same size and shape in reference coordinates.

For example, an actual triangular element mapped in to a natural coordinate system, is always an isosceles triangle having the length of sides equal to unity. The family of elements mapped are called master elements. The displacements are directed parallel to global coordinates not parallel to natural coordinates.

In terms of generalized coordinates ai, bi the displacement models are given by the equations

- U(

Consider a three-node, linear triangular element. Let Ç and y be the natural coordinates for the triangular element . The master element is as shown in an earlier figure

The displacement models as linear polynomial

are given by $u(Ç , y) = a1+a2 Ç +a3 y$

$v(Ç , y) = b1+b2 Ç +b3 y$

where u and v are displacements field inside the element, a1, a2, a3, b1, b2, and b3 are the generalized coordinates to be determined from the following nodal conditions.

At Ç = 1, y = 0; u(1,0) = ui v(1,0) =

vi At Ç = 0, y = 1; u(0,1) = uj

v(0,1) =

vj At Ç = 0, y = 0; u(0,0) = uk

v(0,0) =

vk

where ui, vi, uj, vj, uk and vk are the nodal

displacements. u(Ç , y) = a1+a2 Ç +a3 yv(Ç , y)

= b1+b2 Ç +b3 y

At Ç = 1, y= 0; u(1,0) = ui v(1,0) = vi
ui = a1+a2 (1) +a3 (0) = a1+a2
eq1 vi =
b1+b2 (1) +b3 (0) = b1+b2
eq2

At Ç = 0, y= 1; u(0,1) = uj v(0,1) = vj
uj = a1+a2 (0) + a3 (1) = a1 +a3 eq 3 vj = b1+b2 (0) + b3 (1) = b1 +b3 eq

At Ç = 0, y= 0; u(0,0) = uk v(0,0) = vk
uk = a1 vk = b1 or a1 = uk b1 =
vk Substituting the values of a1 b1
in the eqs 1 2 3 4 above

ui = a1+a2 uk +a2 a2 = ui - uk
vi = b1+b2 vk +b2 b2 = vi – vk

uj = a1 +a3 = uk +a3 a3 = uj - uk
vj = b1 +b3 = vk +b3 b3 = vj –vk

Thus we have

$a1 = uk$ and $b1 = vk$

$a2 = ui - uk$ $b2 = vi - vk$

$a3 = uj - uk$ $b3 = vj - vk$

Substitution of these constants into equation

$u(Ç , y) = a1+a2\, Ç +a3\, y$ $v(Ç , y) = b1+b2\, Ç +b3\, y$

$u(Ç , y) = uk+ Ç(ui- uk)+ y(uj- uk)$ $v(Ç , y) = vk+ Ç(vi- vk)+ y(vj- vk)$

$$\begin{aligned} u(Ç , y) &= uk + Ç\, ui – Ç\, uk + y\, uj – y\, uk \\ &= Ç\, ui + y\, uj + uk - Ç\, uk - y\, uk \\ &= Ç\, ui + y\, uj + (1 - Ç - y)\, uk \\ &= Ni\, ui + Nj\, uj + Nk\, uk \end{aligned}$$

$$\begin{aligned} v(Ç , y) &= vk+ Ç\, vi- Ç\, vk+ y\, vj- y\, vk \\ &= Ç\, vi + y\, vj + vk – Ç\, vk - y\, vk \\ &= Ç\, vi + y\, vj + (1 - Ç - y)\, vk \\ &= Ni\, vi + Nj\, vj + Nk\, vk \end{aligned}$$

where $Ni = Ç$, $Nj = y$ and $Nk = 1- Ç- y$ are the shape functions of linear triangular element. The shape functions are linear over the entire element.

Isoparametric Linear Quadrilateral Element:

Consider the general quadrilateral element defined in x- and y- coordinates shown in an earlier fig. Let i, j, k and l be the nodes labeled in the counter clockwise direction from node i.

Let u and v be the displacements field within the element.
The general quadrilateral element can be expressed in terms of the master element defined in Ç, y coordinates and is square shaped. The shape functions for the element can be derived using the Lagrange shape function formula in the Ç and y directions. Let us first derive the shape function Ni at node i.

In the Lagrange formula for two-dimensional element, replacing P by i and since element is linear, we have

$Ni(Ç, y) = Ni(Ç)\, Ni(y)$ Where $Ni(Ç)$ is the shape function at node i.

It can be defined by treating separately as one-dimensional case in Ç coordinate. Therefore Ni(Ç) = (Ç - Çj) / (Çi - Çj)

Since, there are only 2 nodes i and j along the –ve y side of the element Using nodal coordinates in natural coordinate system,
we have Ç i = -1, Ç j = +1 substituting in the equation

Ni (Ç) = (Ç – Ç j) / (Ç i – Ç j) = (Ç – 1) / (-1 – 1) Ni (Ç) = (Ç-1) / -2

Similarly, we can obtain the Ni(y) along the -ve Ç side of element using the eqn. Ni(y) = (y – y l) / (y i - y l)

Since, along the –ve side Ç only i and l nodes are present substituting their nodal coordinates
y i = -1 y l = 1 into equation above we obtain

Ni (y) = (y – 1) / (-1 - 1) Ni (y) = (y - 1) / -2

Thus the shape function at node l is got by multiplying Ni (Ç) Ni (y)

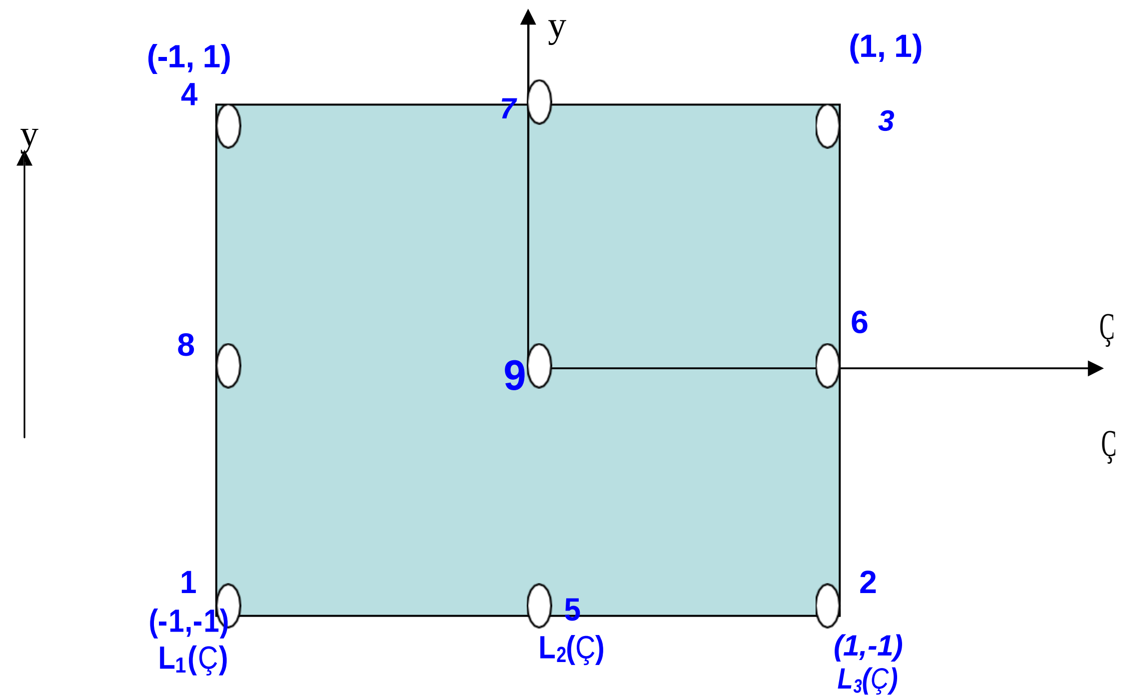
Ni(Ç, y) = [(Ç-1) /-2][(y-1)/-2]
= ¼(1-Ç)(1-y)

Similarly, we can find the remaining shape function at j, k and l nodes. Thus, all the four shape functions can be written as

Ni(Ç, y) = ¼(1-Ç)(1-y)
Nj(Ç, y) = ¼(1+Ç)(1-y)
Nk(Ç, y) = ¼(1+Ç)(1+y)
Nl(Ç, y) = ¼(1-Ç)(1+y)

While implementing in a computer program, following general equation can be used.

Np(Ç, y) = ¼(1+Ç Çp)(1+y yp) for p = i, j, k and l.



The stresses at Ç = y = 0 are now given by $o^{0} = D\ B^{0}\ q$

[D]			
30 x 106 / (1-.09) =32.96x106	1	0.3	0
	0.03	1	0
	0	0	0.35

[q]
0
0
0.002
0.003
0.006
0.0032
0 44
0

Problem:

For the simple bar shown in the figure determine the displacements, stress and the reaction. The cross section of the bar is $500mm^2$, length is 1000mm, and the Young's Modulus is $E = 2X10^5$ N/mm^2. Take load P = 1000N.

Results:

Deformation at fixed end = 0 Deformation at mid section = 0.005mm

Deformation at free end where the load is acting = 0.01mm Stress in the Bar = σ = 2 N/mm^2

Reaction Force = R_1 = -1000N

~ ANSYS Multiphysics Utility Menu - - [J@]~

Eile ~elect !,ist elot Plot!:.trls l.H!orl<Plane Patamoto" ~OCTO Me!YClr1s !!et>

f>N5YS Toolbar ®

SAVE_DB | RESUM_DS! QUIT! POWRGRPH!

EJ **Prererences**

8 W;;·13#11'"

Iii Element Type

Iii Real Constants

Iii Material Props

Iii Sections

Iii Modeling

Iii Meshing

Iii Checking Ctrl.

m IlUmberlng Ctrls

m Alchive Model

m CoupRng/ Ceqn m

flOTRAN Set Up m

Multl-fleld Set Up

m loads

m Physics

I!I Path Operations

Itt **Solution**

m **General PostptOC**

m **TimeHist Postpro**

IB Topological Opt

IB ROM Tool

IB Design Opt

@ **Prob Design**

IE Radiation Opt

ttl **Run-Time Stats**

~ **Session Editor**

~ Finish

~ ANSYS Mulliphyslcs Ulilily Menu - ~~1:El

EUe 2elect bl.t elot ~ ytoricl>Iono PO(omet.... Macro Meo.uCtris t[elp

ANSVS Toolbar ®

8 Preprocessor
S ji§,UJ,liHM
~ AddjEditjDelete
~ Switch Bern Type
~AddDOf
~ Remove DOfs
~ Elem Tech Control
8 Real Constants
13 Material Props
ttl Sections
8 Modeling
ttl Meshing
8 Checking Ctrls
(±J Numbering Ctrls
III Archive Model
|±| Coupling / Ceqn
|±| FIOTRAN Set Up
III Multi-Reid Set Up
III loads
III Physics
III Path Operations
III Solution
III General Po.tproc
III TlmeHl.t Postpro
III Topological Opt
III ROM Tool
IE Oe.lgn Opt
IE Prob Oe.lgn
IB Radiation Opt
I!I Run-**Time Stats**
Il:::ISession Editor
Il:::IFinish

|Pick a menu lern or enter an IWSYS Corrwnand (PR£P1)

.:start ~ ~1WSYS~1(5U ...

B Preprocessor

B Element Type

~ '·tnN!bti."mc

Ell Switch Elem Type

Ell Add oaf

Ell Remove DOfs

Ell Elem Tech Control

IE Real Constants

IE Material Props

IE Sections

IE Modeling

IE Meshing

IE Checking Ctrls

IE Numbering Ctrls

IE Archive Model

III Coupling / Ceqn III

FlOTRAN Set Up III

Multi-field Set Up ľ±l

Loads

ľ±l Physics

ľ±l Path Operations

ľ±l Solution

ľ±l GeneralPostproc

ľ±l TimeHist Postpro

III Topological Opt

III ROM Tool

ľ±l Design Opt

ľ±l Prob Design

El Radiation Opt

IE Run-Time Stats

~ Session Editor

~ Finish

Add ...

Delete

De&ed Element Tvoes:
lONE DEFiNED

B Preprocessor
B Element Type
~ '·tnN!bti."mc
Ell Switch EIem Type
Ell Add oaf
Ell Remove DOfs
Ell Elem Tech Control
IE Real Constants
IE Material Props
IE Sections
IE Modeling
IE Meshing
IE Checking Ctrls
IE Numbering Ctrls
IE Archive Model
III Coupling / Ceqn III
FlOTRAN Set Up III
Multi-field Set Up ±
Loads
± Physics
± Path Operations
± Solution
± GeneralPostproc
± TimeHist Postpro
III Topological Opt
III ROM Tool
± Design Opt
± Prob Design
El Radiation Opt
IE Run-Time Stats
~ Session Editor
~ Finish

Add•••

OK

Apply

B Preprocessor
B Element Type
~ '·tnN!bti."mc
Ell Switch EIem Type
Ell Add oaf
Ell Remove DOfs
Ell Elem Tech Control
IE Real Constants
IE Material Props
IE Sections
IE Modeling
IE Meshing
IE Checking Ctrls
IE Numbering Ctrls
IE Archive Model
III Coupling / Ceqn III
FlOTRAN Set Up III
Multi-field Set Up ⊞
Loads
⊞ **Physics**
⊞ **Path Operations**
⊞ **Solution**
⊞ **GeneralPostproc**
⊞ **TimeHist Postpro**
III Topological Opt
III ROM Tool
⊞ **Design Opt**
⊞ **Prob Design**
El Radiation Opt
IE **Run-Time Stats**
~ Session Editor
~ Finish

...,. I lIWI

Add ... Options ... Delete

~ ANSYS Mulliphyslcs Ulilily Menu - ~~1:El

EUe 2elect bl.t elot ~ ytoricl>lono PO(omet.... Macro Meo.uCtris t[elp

ANSVS Toolbar ®

8 Preprocessor

B ;m' .MtM§

IB Element Type

3u

~ AddjEditjDelete

~ Thickness Func

IB Material Props

S **Sections**

IB Modeling

IB Meshing

IB Checking ctrls

13 Numbering Ctrls

IB Archive Model

IB Coupling / Ceqn

IB FlOTRAN Set Up

IB Multi-field Set Up

IE loads

IE Physics

III Path Operations

III Solution

III GeneralPostproc

III TlmeHlst Postpro

III Topological Opt

III ROM Tool

III Design Opt

III Prob Design

III Radiation Opt

III Run-Time Stats

E::l Session Editor

~ Finish

Pick a menu lern or arter an IWSYS Corrwnand (PR£P1)

'. *start* ~.o~... ~lWSYSfUJphysI(5U ... rlB!~licrosoitPowerPoot... ~ :;rJ Iij • ~.A C)~ IO:36PM

B Preprocessor
I!I Element Type
B Real Constants
E1 r.nnritihtj'hr:C
Eil Thickness func:
iii Material Props
iii Sections
iii Modeling
IE Meshing
IE Checking Ctrls
IE Numbering Ctrls
IE Archive Model
IE Coupling / Ceqn
IE FlOTRAN Set Up
IE Multl·Reld Set Up
I!Jloads
I!I Physics
I!I Path Operations
I±I **Solution**
I±I GeneralPostproc
I±I **TimeHist Postpro**
I±I Topological Opt
I±I ROM Tool
ttl **Design Opt**
8 Prob Design
8 Radiation Opt
13 Run-Time Stats
~ Session Editor
r!jj Finish

NOliE OEFINEO

Add .••

~iphYSics Utility Menu

E.

8 Preprocessor

- I!I Element Type
- 8 Real Constants
  - EI j,»ntJjjhtJ.t§tlij
  - C Thickness fUn<
- I!I Material Props
- I!I 5e<:tlons
- El Modeling
- El Meshing
- El Checking Ct,ls
- m Humbe,Ing Ct,ls
- m A,chlve Model
- m Coupling / Ceqn
- m fLOTRAN Set Up
- El Multl·neld Set Up
- El Loads
- III Physics
- III Path Ope,atlons

rd **Solution**

± **General Postproc**

ttl **TimeHist Postpro**

III TopologicalOpt

III ROM Tool

ttl Design Opt

ttl **Prob Design**

ttl **Radiation Opt**

ttl **Run-Time Stats**

~ **Session Editor**

~ finish

T~ 1 UNn

~iphYSics Utility Menu

E.

8 **Preprocessor**
!! Element Type
8 Real Constants
EI j,»ntJjjhtJ.t§tlij
C Thickness fUn<
!! Material Props
!! 5e<:tlons
El Modeling
El Meshing
El Checking Ct,ls
m Humbe,Ing Ct,ls
m A,chlve Model
m Coupling / Ceqn
m fLOTRAN Set Up
El Multl·neld Set Up
El Loads
III Physics
III Path Ope,atlons
Solution
General Postproc
TimeHist Postpro
III TopologicalOpt
III ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-Time Stats
~ **Session Editor**
~ finish

NONE OEft4£1)

Cross~¥.a AREA 500

Ir«ial sir'*' ISIRN

OK Apply Cancel Help

B Preprocessor
I!I Element Type
B Real Constants
E1 r.nnritihtj'hr:C
Eil Thickness func:
iii Material Props
iii Sections
iii Modeling
IE Meshing
IE Checking Ctrls
IE Numbering Ctrls
IE Archive Model
IE Coupling / Ceqn
IE FlOTRAN Set Up
IE Multl·Reld Set Up
I!Jloads
I!I Physics
I!I Path Operations
I±I **Solution**
I±I **GeneralPostproc**
I±I **TimeHist Postpro**
I±I Topological Opt
I±I ROM Tool
ttl **Design Opt**
8 **Prob Design**
8 **Radiation Opt**
13 **Run-Time Stats**
~ **Session Editor**
r!jj Finish

Add... Edit... Delete

B Preprocessor
- Element Type
- Real Constants
- Material Props
  - Mat"rial Library
  - Temperat .. e lJnIts
  - Electromao Units
  - §ffl'JU;!'§@Qa
  - (onv"rt AlPx
  - (hange Mat Num
  - Failure (rlt"rla
  - Write to file
  - R" ..d (rom File
- Sections
- Modeling
- Meshing
- Checking Ctrls
- **Numbering Ctrls**
- **Archive Model**
- Coupling / Ceqn
- FlOTRAN Set Up
- Multi-field Set Up
- **Loads**
- **Physics**
- **Path Operations**

Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-rmeStats
Session Editor
Finish

- Favor~..
- StructU'01
- Thermal
- CFD
- Electromat,jtletlc.
- Acou.tic.
- Fluid.
- Piezoelectric.
- Piezoreslstlv~y
- Thermoelectr~

YS Mulliphysic~ UlilUy Menu

.:Jll~lr

J

8 **Preprocessor**
tB Element Type
tB Real Constants
8 Material Props
tB Materiallibtary
ril Temperature Units
ril Eledromag Units
e 4m·g'611§rmSm
ril Convert ALPH
ril Change Mat Num
IB Failure Criteria
~ Write to File
El Read from File
ttl **Sections**
tB Modeling
tB Meshing
tB Checking (lrl.
m Numbering Clrls
m Archive Model
m Coupling / Ceqn m
FLOTRAN Set Up m
Multi-Held Set Up m
Loads
m Physics
m Path Operations
lll Solution
lll General Postproc
lll TlmeHist Postpro
l!l Topological Opt
l!l ROM Tool
IE Oesign Opt
IB Prob Design
IE Radiation Opt
IB Run-Time Slats
ril Session Editor
rilFonish

- _erial Models Available

(ill Favo":es
.- structural
(ill.
(ill Nonlinear
~ Density
(ill Thermal Expansion
(ill D.mplng
~ Friction Coefficient
(ill Specl.llzed M.terl.l,
(ill Therm.1
(ill CFD

Pick a mew lem or enler an ANSYS COIMWIlld (PREP7)

~iphYSics Utility Menu

E.

8 Preprocessor
- Element Type
- Real Constants
- Material Props
  - Materiallibr.ry
  - Temperature Units
  - flectromao Unit.
  - Aft"J'6"AtifjMij
  - Convert AlPM
  - ChanO" Mat Num
  - Failure Criteria
  - Write to File
  - Reed from File
- Sections
- Modeling
- Meshing
- Checking Ctrls
- **Numbering Ctrls**
- **Archive Model**
- Coupling / Ceqn
- fLOTRAN Set Up
- Multi-field Set Up
- **loads**
- **Physics**
- **Path Operations**

Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Oesign Opt
Prob Design
Radiation Opt
Run-TIme Stats
Session Editor
Fonish

Molerial Models Available

- Favotles
- Structural
  - lIne.r
    - El.stlc
      - **$l1li**
      - Orthotroplc
      - Anisotropic
  - Nonlln•• r
  - Density
  - thermel Exp.nslon
  - Damping

B Preprocessor
- Element Type
- Real Constants
- Material Props
  - Mat"rial Library
  - Tempe,at.. e lJnIts
  - Elect,omao Units
  - Affl'JU;!';@Qa
  - Conv",t AlPx
  - Change Mat Num
  - Failu,e C,lt",la
  - W,ite to file
  - Read ('om File
- Sections
- Modeling
- Meshing
- Checking Ct,ls
- Numbering Ctrls
- Archive Model
- Coupling / Ceqn
- FLOTRAN Set Up
- Multi-field Set Up
- Loads
- Physics
- Path Operations

Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-rmeStats
Session Editor
Finish

Material Models Defned -------..,

lato"al Madej Numbe

Material Models Avoioble

- Favorles
- Sb'urtu'al
  - ~lilear
    - ~ElMbc
      - ~m
      - ~ Oott-dropic
      - ~-opic
  - Norhar
  - ~ Denstty
  - Thermal Exponsion
  - Dampint,j

linear Isotropic Material Properties for Material Number 1

T1

TmJ t stures

EX FI2.:.,:el:l::...._

PRXV

Add Temperature | Delete Temperature

B Preprocessor
- Element Type
- Real Constants
- Material Props
  - Mat"rial Library
  - Temperat .. e lJnIts
  - Electromao Units
  - §ffl'JU;!'§@Qa
  - (onv"rt AlPx
  - (hange Mat Num
  - Failure (rlt"rla
  - Write to file
  - R" ..d (rom File
- Sections
- Modeling
- Meshing
- Checking Ctrls
- **Numbering Ctrls**
- **Archive Model**
- Coupling / Ceqn
- FlOTRAN Set Up
- Multi-field Set Up
- **Loads**
- **Physics**
- **Path Operations**

Solution
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-rmeStats
Session Editor
Finish

Material Models Available

- Favor~..
- structU'al
  - Llneor
    - Elastic
      -
      - OrthOU0Ili<
      - ArOsotrol>ic
  - Nonlinear
  - Density
  - Thermal E)(I)aMion
  - Damping

~ ANSYS Multiphysics Utility Menu - [J@~

!:ile :;ieleet list ~Iot Plotgrls !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar ®

B Preprocessor
- HI Element Type
- HI RealCoostants
- HI Material Props
- m Sections
- **B** ;rmC"tm1m
  - Create m
  - Operate
  - IIJ Move / Modify
  - IIJ Copy
  - IIJ Renect
  - IIJ Check Geom
  - IIJ Oelete
  - IIJ CycliC Sector
  - IIJ CMS
  - 1:1 Genl plane strn
  - 1:1 Update Geom
- I±I Meshing
- I±I Checking ctrls
- **I±I Numbering Ctrls**
- **I±I Archive Model**
- I±I Coupling / Ceqn
- I±I FLOTRAN Set Up
- I±I Multi-field Set Up
- **8 Loads**
- **ttl Physics**
- **ttl Path Operations**

ttl Solution

8 General Postproc

IE TimeHist Postpro

I±I Topological Opt

8 ROM Tool

8 Oesign Opt

8 Prob Design

8 Radiation Opt

8 Run-rune stats

~ANSYS Multiph ics Util

Eile :;ielect list elot Plotgr1s !!£or!4'!one Pao.:_s Macro Me!!uCtris tlelp

ANSYS Toolbar

®

B Preprocessor

iii Element Type
iii Real Constants
iii Material Props
iii Sections
B Modeling
B Create
B ICeypoints
Fl On Working Plane
C1 ,tit·fitffJSi
Fl Online
Fl On Line w/Ratlo
Fl On Node
FlICP between KPs
Fl Fillbetween I(P.
l!! KPat center
l!! Hard PT on line
l!! Hard PT on area
[±] **Lines** [±]
Areas (±)
Yolumes [±]
Nodes
[±] **Elements**
~ Contact Pair
[±] **Piping Models**
ttl **Circuit**
~ Racetrack l:oil
8 Transducers
IE Operate
iii Move / Modify
iii Copy
iii Reflect
13 Check Geom
13 Delete
13 Cyclic Sect.,..
13 eMS

[KJ C,eateKeypo!nts In Active Coordinate System

WT Keypo!nt number

X,Y,Z location In active CS

L-_.....J'll -~l._l _-----'

OK Apply Cancel

fie 2elect bist eiot Plot!:;.trls 'N_orkPlane Patameters Macro MeQuClrIs ~

®

SAVE_pel RfSUM_DBI QUITI POWRGRPHI

B Preprocessor
- Element Type
- **Real Constants**
- **Material Props**
- **Sections**
- Modeling
  - Create
    - Keypoints
      - On Working Plane
      - **'b',tiMji,**
      - On line
      - On line wIRatio
      - On Node
      - ICP between ICPs
      - Fill between ICPs
      - ICP at center
      - Hard PT on line
      - Hard PT on .r"a
    - Line.
    - Area.
    - Volume.
    - Node.
    - Elements
    - Contact Pair
    - Piping Models
    - **Circuit**
    - Racetrack Coil
    - **Transducers**
  - Operate
  - Move / Modiry
  - Copy
  - ReHect
  - Check Geom
  - Delete
  - **Cyclic Sector**
  - EMS

X,V,Z **location in active CS** 1 10--1----,1 L_I

OK Apply Cancel

..!.L

~ANSYS Multiph ics Util

Eile :;ielect list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar

®

B Preprocessor

iii Element Type

iii RealCoostants

iii Material Props

iii Sections

B Modeling

B Create

B ICeypoints

Fl On Working Plane

C1 ,tit·fitffJSi

Fl Online

Fl On Line w/Ratlo

Fl On Node

FlICP between KPs

Fl Fillbetween I(P.

I!I KPat center

I!I Hard PT on line

I!I Hard PT on area

|±| **Lines** |±|

Areas (±)

Yolumes |±|

Nodes

|±| **Elements**

~ Contact Pair

|±| **Piping Models**

ttl **Circuit**

~ Racetrack c:oil

8 Transducers

IE Operate

iii Move / Modify

iii Copy

iii Reflect

13 Check Geom

13 Delete

13 Cyclic Sect.,..

13 eMS

X,V,l Loc-.In act;e CS 11(0)

01(Apply Cancel ~

Eile 2elect bl.t elot ~ ytOfld>lono Potomet.... Macro Meo.uCtris t[elp

ANSVS Toolbar

®

8 Preprocessor
IB Element Type
IB Real Constants
IB Material Props
IB Sections
8 Modeling
8 Create
8 Keypoints
~ On Working Pian e
e 'bi'Htmi,
~ Online
~ On line w/Ratio
~OnNode
~ KP between KPs
~ fill betw een KPI
iii KP at center
iii Hard PT on line
iii Hard PT on are.
III Line.
III Areas
III volumes
III Nodes
III Elem ents
1:1 Contact Pair
III Piping Models
III Circuit
1:1 Racetrack eoil
m Transduc .. r.
III Operate
III Move / Modify
III Copy
IB Renect
IB Check Geom
IB Delete
IB Cyclic Sector
IB CMS

~

~ ■ ",""_ "-

Pick a menu lern or enter an IWSYS Corrwnand (PR£P7)

.: *start* ~ I ~1WSYS~su ..•

15r.J.. • ~ .A. -:-I~~,,! 10:39 PM

~

~ ANSYS Multiphysics Utility Menu ~@~

Eile :i.elect list eJot Plot!:;trls l!'!.orkPiane Pa(ameters Macro Me!luCtris !:!elp

e

Ei:l Preferences
13 Preprocessor
!!! Element Type
ı±ı Real Constants
8 Material Props
IE Sections
13 Modeling
13 Create
ttl Keypoints
ttl Areas
ttl Volumes
IE Nodes
IE Elements
~ Contact Pair
ttl Piping Models
ttl Circuit
Ei:l Racetrack Coli
I1J Transducers
I1J Operate
I1J Move / Modlry
I1J Copy
I1J ReHect
ttl Check Geom
ttl Delete
I±) Cyclic Sector
ttl CMS
~ Genl plane strn
~ Update Geom
ttl Meshing
ttl Checking Ctrls
ı±ı Numbering [trls
ttl Archive Model
ttl Coupling / Ceqn
ttl FlOTRAN Set Up
ttl Multi-field Set Up
ı±ı Loads

~ANSYS Multiph ics Util

Eile :;ieleet list elot Plotgr1s !!£or!4'!one Pao.:_s Macro Me!!uCtris tlelp

ANSVS Toolbar

®

Preprocessor

iii Element Type
iii Real Constants
iii Material Props
iii Sections
B Modeling
B Create
m Keypoints
B lines
B line.
~a·ziiiihi@
Flln Active Coord
Fl Overlaid on Area
Fl Tangent to line
Fl Tan to 2 Line.
Fl Normal to line
Fl Norm to 2 Line.
Fl At angle to line
Fl Angle to 2 line.
!!! **Arcs**
l± **Splines**
FlUne Fillet
ttl **Areas**
IE **Volumes**
ttl **Nodes**
ttl **Elements**
~ Contact Pair
IE **Piping Models**
IE **Circuit**
~ Racetradc Coil
IB **TransdtKers**
IB Operate
IB Move / Modify
IBCopy
IB Reflect
IB Check Geom

r Ih>pick
r
r

0

Maximum. • Z
Minimum. .; Z
KeyP No.

r. **Lise of Ieems**
r **Min" Max .. Inc**

OK Apply
Cancel
Help

I[ISTR) Pick or enter end keypcints ri Ine

Elle :;ieleet list elot Plot~trls :!iorl4'tone Pllt-.... tlaCro Me!luCtris tlelp

ANSVS Toolbar

SAVE_OB RESUM_DB QlJITI POWRGRI'HI

Preferences

f3 Preprocessor

!il Element Type

!il Real Con.tants

!il Material Prop.

!il Section'

B Modeling

B Create

HI Keypoinu

B line.

B Line.

~ a;zliilH'Cm;

~ In Active Coord

~ Overlaid on Area

~ TIIngent to line

~ Tan to 2 Lines

~ Nornlal to line

~ Nornl to 2 Line.

~ At angle to line

~ Angle to 2 lines

I!J Arcs

I±I Splines

~ line Fillet

ttl Areas

IE Volumes

IE Nodes

IE Elements

~ Contact Pair

S Piping Models

IB Circuit

~ Racetrack Coil

IE Transducers

IB Operate IB

Move / Modify

IBCopy

IB Reflect

IB CheckGeom

r Ullpick

r

r

0

Maximum. • 2

Hinim.u.m. • Z

KeyP Mo.

r. List of Ieems

r Min .. Max .. Inc

Apply

Cancel

Help

[ISTR] Pick cx enter end keypoOts rJ lne

~ ANSYS Multfphysfcs Utility Menu -

~@11f)

E~e :ielect bist eiot Plotr.:.trls l'l!o,kPl.no p.t.meters Mocro MenuClrls ~

..:Jll~lr1m

®

ANSVS Toolba,

SAVE_DBI RESUM_DBI QUITI POWRGRPHI

J

~ Preferences

8 **Preprocessor**

tB Element Type

tB Real Constants

tB Material Props

tB Sections

tB Modeling

B Meshing

tB Mesh Attributes

~MeshTool

8 .1EJ4lt\" lll

~ Mesher Opts

IE Concatenate

iii Mesh

iii Modify Mesh

iii Check Mesh

iii Clear

Iiil Checking Clrl.

Iiil Numbering Clrls

Iiil Archive Model

Iiil Coupling / Ceqn

Iiil flOTRAN Set Up

Iiil Multi-Held Set Up

Iiilloads

Iiil Physics

Iiil Path Operations

III Solution

III General Postproc

III TlmeHist Postpro

iii Topological Opt

iii ROM Tool

iii Design Opt

IB Prob Design

tB Radiation Opt

IB Run-Time Slats

~ Session Editor

~Fonish

Pick. mew tem or enler an ANSYS COIMWIlld (PREP7)

~ ANSYSMultiphysics Utility Menu - [J@~

!:ile .:ieleet list ~lot Plotgrls !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar ®

B Preprocessor

- HI Element Type
- HI RealCoostants
- HI Material Props
- m Sections
- m Modeling
- 8 Meshing
  - m Mesh Attributes
  - El Me.hTool
  - 8 Site Cntrl.
    - III SmartSI:e
    - 8 ManualSI.e
      - Global
      - Area.
      - -
      - III Keypolnt.
      - I!llayer.
    - I± **Concentrat (Ps**
  - ~ Mesher Opt.
  - I±I **Concatenate**
  - I±I Mesh
  - I±I Modify Mesh
  - I±I Check Mesh
  - **IE Clear**
- I±I Checking Ctrls
- ttl **Numbering Ctr'ls**
- ttl **Archive Model**
- I±I Coupling / Ceqn
- HI FIOTRANSet Up
- I±I Multi-field Set Up
- IB loads
- HI Physics
- HI Path Operations

HI Solution

HI General Postproc

HI TmeHist Postpro

~ANSYS Multiph ics Util

Eile :;ielect list elot Plotgr1s !!£or14'!one Pao.:_s Macro Me!!uCtris tlelp

ANSVS Toolbar ®

B Preprocessor
iii Element Type
iii RealCoostants
iii Material Props
iii Sections
iii Modeling
8 Meshing
W Mesh Attributes
El M".hTool
8 Size Cntrl.
W SmartSI:"
8 ManualSI.e
W Global
W Area.
8 line.
I:I All lines
PJ mttm'ii.igi
iii Copy Diy.
iii Flip 81a.
iii Clr Size
iii KeypoInt.
I± **Layers**
I± **Concentrat KPs**
~ Mesher Opts
13 Concatenate
IB Mesh
IB Modify Mesh
IB Check Mesh
IB Clear
IB Checking Clrls
S Numbering Ctrls
iii Archive Model iii
Coupling *ICeqn* iii
ROTRAN Set Up
iii Multi-field Set Up
iii loads

r lh>piclc
r Box
Polygon r Cirole
r Loop

Coune	1
Haxim,1JJll	1
HinimUJll	1
Line No.	1

r· **Lise of Ieems**
r **Min, Max, Inc**

OK Apply

Resee

fie 2elect bist eiot Plot!:;.trls 'N_orkPlane Patameters Macro MeQuClrIs ~

®

SAVE_pel RfSUM_DBI QUITI POWRGRPHI

B Preprocessor

I!! Element Type

I!! Real Constants

ttl Material Props

IB Sections

I!! Modeling

BMeshing

I!! Mesh Attributes

!:il MeshTool

B Size Cntrls

IB SmartSize

8 ManualSize

I!! Global

IB Areas

8 lines

I:I All Lines

~m@I'M

p (opyDIvs

P Flip Bias

P (IrSIze

m Keypoints

m Layers

IB Concentrat KPs

!:il Mesher Opts

ttl Concatenate

I!! Mesh

I!! Modify Mesh

I!! Check Mesh

I!! dear

I!! Checking ctrls

I!! Numbering ctrls

I!! Archive Model

I!! Coupling / Ceqn

I!! FLOTRANSet Up

I!! Multi-field Set Up

I!! Loads

..!.L

Element edge length

No. of element <ivisions

(NOIVis usedoolyi SJ2E is blank or zero)

KYNOIV SIZE,NOIV *can* be changed J;1 Yes

SPACE Spacing ,atio

ANGSIZ Division *arc* (deQrees)

(us. ANGSIZ ooIy f rurber r:J division. (NOIV) and

elementedge length (SIZE)are blank or .ero)

Cle., .ttached areasend vdumes

Apply Cancel Help

~ANSYS Multiph ics Util

Eile :;ielect list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar ®

B **Preprocessor**
iii Element Type
iii Real Constants
iii Material Props
iii Sections
iii Modeling
8 Meshing
W Mesh Attributes
El M".hTool
8 Size Cntrl.
W SmartSI:"
8 ManualSI.e
W Global
W Area.
8 line.
I:I Alllines
PJ mttm'ii.igi
ill Copy Diy.
ill Flip 81a. ill
Clr Size
iii Keypolnt.
I± Layers
I± Concentrat KPs
~ Mesher Opts
13 Concatenate
IB Mesh
IB Modify Mesh
IB Check Mesh
IB Clear
IB Checking Clrls
S Numbering Ctrls
iii Archive Model
iii Coupling *ICeqn* iii
ROTRAN Set Up iii
Multi-field Set Up iii
loads

(I'I>IVIs used *only* **/** SIZE Is blank or zero)

P' Yes

(use ANGSIZ only If number of divisions (NOlV) and element edge length (SIZE) are blank or zero)

etellr Ilttliched areas and volumes

OK Apply Cancel

fie 2elect bist eiot Plot!:;.trls 'N_orkPlane Patameters Macro MeQuClrIs ~

®

SAVE_pel RfSUM_DBI QUITI POWRGRPHI

B Preprocessor

I!! Element Type
I!! Real Constants
ttl Material Props
IB Sections
I!! Modeling
BMeshing
I!! Mesh Attributes
!:il MeshTool
B Size Cntrls
IB SmartSize
8 ManualSize
I!! Global
IB Areas
8 lines
I:I All Lines
~m@I'M
p (opyDIvs
P Flip Bias
P (IrSIze
m Keypoints
m Layers
IB Concentrat KPs
!:il Mesher Opts
ttl Concatenate
I!! Mesh
I!! Modify Mesh
I!! Check Mesh
I!! dear
I!! Checking ctrls
I!! Numbering ctrls
I!! Archive Model
I!! Coupling / Ceqn
I!! FLOTRAN Set Up
I!! Multi-field Set Up
I!! Loads

..!.L

Eile 2elect bl.t elot ~ ytOflcl>lono Potomet.... Macro Meo.uCtris t[elp

_j

FIlle 6 Coarse

Size Corbols:

8 Preprocessor
IB Element Type
IB Real Constants
IB Material Props
IB Sections
IB Modeling
8 Meshing
IB Mesh Attributes
e @btiffll
8 **Size Cntrls**
~ Mesher Opts
[!J **Concatenate**
IB Mesh
IB Modiry Mesh
IB Check Mesh
III Clear
IE Checking Ctrls
IE Numbering Ctrls
IE Archive Model
I!J Couplino / Ceqn
I!J FLOTRAN Set Up
I!J Multl-neld Set Up
I!J Loads
I!J Physics
I!J Path Operations
I!J Solution
IE General Postproc
IE TimeHlst po,tpro
I±I Topological Opt
I±I ROM Tool
III Design Opt
IB Prob Design
IB Radiation Opt
IB Run-Time Stats
~ Session Editor
~ Finish

!leal
!leal
!leal

Copy

Clear

Clear

Lines 3

li' Flodlol

r Mopped r P

13 or 4 10 d 3

M•• h Cl•• r

1Element. 3

Rem.

H""

Pick a menu lern or arter an IWSYS Corrwnand (PR£P1)

~ANSYS Multiph ics Util

Eile :;ielect list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar

®

B Preprocessor

iii Element Type

iii Real Constants

iii Materlall'rops

iii

iii Mo~I"lin.

8 Meshing

iii

E!'.I

111

I!|I

111

111

111

111

iii

r Box

r: **Circl.**

Co",,~ 1

1

1

Line No.. 1

C **Jlin ~ Hax. Inc**

OK Apply

Resee **Cancel**

~ ANSYS Multiphysks Utilily Menu

!:ile aelect !,1st !!lot Plotl::trls ~or1<l'lano Pot_or. Macro Me[luCtrls t[elp

B Preprocessor

IB Element Type
IB Real Constants
IB Material Props
S **Sections**
IB Modeling
B Meshing
8 Mesh Attributes
e ®btifi1¦
ttl **Size Cntrls**
~ Mesher Opts
ttl **Concatenate**
8 Mesh
8 Modify Mesh
8 Check Mesh
8 Clear
I!J Checking Ctrl.
I!J Numbering Ctrl.
I!J Archive Model
I!J Coupling / Ceqn
I!J FLOTRAN Set Up
I!J Multi-Reid Set Up
I!J Loads
I!J PhYSico
IE Path Operation.
IE Solution
IE GeneralPostproc
I!I TlmeHlst Postpro
I!I Topological Opt
I!I ROM Tool
I!I Oesign Opt
I!I Prob Design
IB Radiation Opt
IB Run-Time Slats
EJ **Session Editor**
~Fmi.h

Pick a meoo lern or enter an AN5YSConYMnd (PREP7)

.:start ~ ~AN5YS~U ... rf.!I~ 10~41PM

~ ANSYS Multiphysics Utility Menu - rg@~

Elle :!elect list elot Plot!:;trls :!iorl4'tone Pllt-.... tlaCro Me!luCtris tlelp

ANSVSToolbar ®

SAVE_OBIRESUM_DBIQ1JITI POWRGRI'HI

HI Pre;processor
8 Solution
m Analysis Type
m .mmffl§
m load Step Opts
m Sf Manaoement (CMS)
EI Result. Trackino
m Solve
m Manual Re.onlno m
Multl-neld Set Up m
ADAMS ConnectIon m
DiagnostIcs
Cil Unabridged Menu
I!I General Postproc
I!I TlmeHlst Postpro
I!I Topological Opt
I!I ROM Tool
I!I Design Opt
I!I Prob Design
IE Radiation Opt
I±I Run-Time Stats
~ Session Editor
1:1 Finish

Pick a menu lern or enter an AH5YSComrnond (SOlUTION)

~ ANSYS Multiphysics Utility Menu - [J@~

!:ile :;ielect list ~Iot Plotgrls !!£ori4'!one Pao.:_s Macro Me!luCtris tlelp

ANSVSToolbar ®

IB **Preprocessor**

B Solution

m Analysis Type

B Define loads

m Settings

▪

m Delete

m Operate

!!J load Step Opts

!!J Sf Management (CMS)

c:I Results Tracking

!!J Solve

!!J Manual Rezoning !!J

Multl·neld Set Up I!I

ADAMS Connection I!I

Diagnostics

l:a Unabridged Menu

General Postpro(

TimeHist Postpro

I!I Topological Opt

I!I ROM Tool

I!I Design Opt

IE **Prob Design**

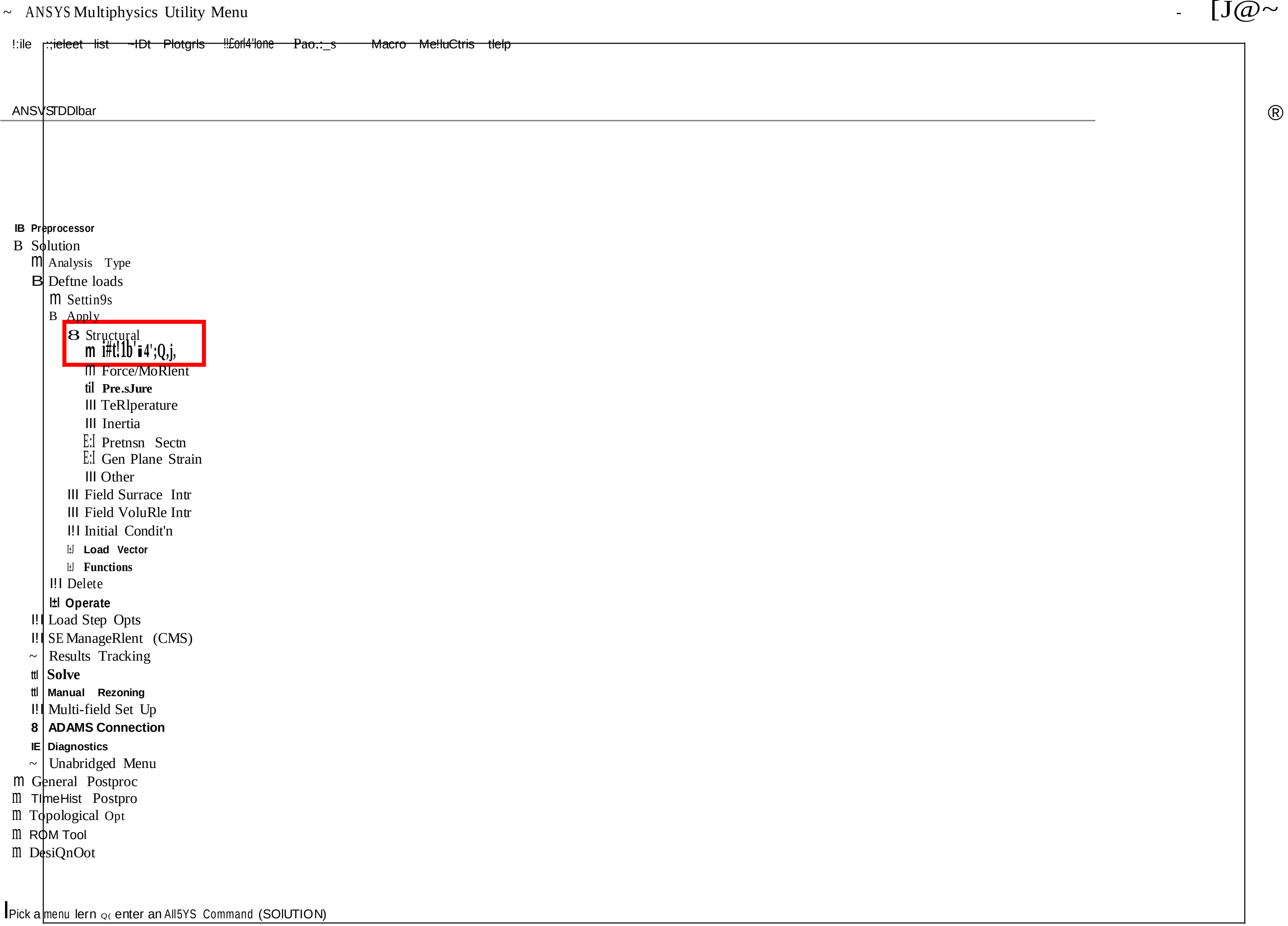
ttl **Radiation Opt**

IB **Run-Time Stats**

~ **Session Editor**

l:a Finish

Pick a menu lern or enter an AllSYS Command (SOIUTION)


ANSYS Multiphysics Utility Menu
ANSYS Toolbar
Preprocessor
Solution
Analysis Type
Define Loads
Settings
Apply
Structural
Force/Moment
Pressure
Temperature
Inertia
Pretnsn Sectn
Gen Plane Strain
Other
Field Surface Intr
Field Volume Intr
Initial Condit'n
Load Vector
Functions
Delete
Operate
Load Step Opts
SE Management (CMS)
Results Tracking
Solve
Manual Rezoning
Multi-field Set Up
ADAMS Connection
Diagnostics
Unabridged Menu
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
DesignOpt

~ANSYS Multiph ies Util

Eile :;ieleet list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar

®

IB Preprocessor
B Solution
m Analysis Type
B Deftne loads
m Settin9s
B Apply
8 Structural
B Displacement
~ Dnlines
~ On Area.
~ On Keypolnts
~ '61;@#1
~ On Node Compo!
III Symmetry B.C.
III Antlsymm B.C.
III Force/Moment
III Pressure:
I±I Temperature
I±I Inertia
~ Pretnsn Sectn
~ Gen Plane Strain
I±I Other
I±I Field Surrace Intr
I±I Field Volume Intr
ttl Initial Condit'n
ttl Load Vector
ttl functions
IB Delete
m Operate
m load Step Opts
m SEManagement (015)
E1I Results Traddng
m Solve
m Manual Rezoning
m Multi-foeld Set Up
m ADAMS Connection

r Box
r Circle

1
U.xilll.UJll. • 11
Minilllua • 1
"'ode No.. 1

r. Lise of Ieems:
r Min, Max, Inc

OK Apply
Reset Cancel

I [OJ Pickar enter nodes far cfsplacement conwarts

~ANSYS Multiph ics Util

Eile :;ieleet list elot Plotgr1s !!£or!4'lone Pao.:_s Macro Me!luCtris tlelp

ANSYS Toolbar

®

IB Preprocessor
B Solution
m Analysis Type
B Deftne loads
m Settin9s
B Apply
8 Structural
B Displacement
~ Dnlines
~ On Area.
~ On Keypolnts
~ '61;@#1
~ On Node Compo!
III Symmetry B.C.
III Antlsymm B.C.
III Force/Moment
III Pressure:
l±l Temperature
l±l Inertia
~ Pretn.n Sectn
~ Gen Plane Strain
l±l Other
l±l Field Surrace Intr
l±l Field Volume Intr
ttl Initial Condit'n
ttl Load Vector
ttl functions
IB Delete
m Operate
m load Step Opts
m SEManagement (015)
E1l Results Traddng
m Solve
m Manual Rezoning
m Multi-foeld Set Up
m ADAMS Connection

o

Apply as

If Constant value then:

VALUE Displacement value

OK Apply Cancel

lPick a menu lern or enter an All5YS Command (SOIUTION)

~ANSYS Multiph ics Util

Eile :;ieleet list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar

®

IB Preprocessor
B Solution
m Analysis Type
B Deftne loads
m Settin9s
B Apply
8 Structural
B Displacement
~ Dnlines
~ On Area.
~ On Keypolnts
~ '61;@#1
~ On Node Compo!
III Symmetry B.C.
III Antlsymm B.C.
III Force/Moment
III Pressure:
I±I Temperature
I±I Inertia
~ Pretnsn Sectn
~ Gen Plane Strain
I±I Other
I±I Field Surrace Intr
I±I Field Volume Intr
ttl Initial Condit'n
ttl Load Vector
ttl functions
IB Delete
m Operate
m load Step Opts
m SEManagement (015)
E1l Results Traddng
m Solve
m Manual Rezoning
m Multi-foeld Set Up
m ADAMS Connection

Pick a menu lern or enter an All5YS Command (SOIUTION)

~ANSYS Multiph ics Util

Eile :;ielect list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar

®

IB Preprocessor
B Solution
m Analysis Type
B oeftne loads
m Settin9s
B Apply
8 Structural
m Displacement
8 Force/Moment
FI On Keypoints
~ iMA@1'
FI On Node (ORlpol
1:1 from Reactions
1:1 from Mao Analy
!!! Pressure
III Tenlperature
III Inertia
~ Pretnsn Sectn
~ Gen Plane Strain
|±| Other
|±| Field Surface Jntr
|±| Field Yolume: Jntr
|±| Initial Condit'n
IB Load Vector
|±| functions
ttl Delete
IB Operate
|±| Load Step Opts
m SEManagement (eMS)
~ Results Traclcing
m Solve
m Manual Rezoning
m Multi-foeld Set Up m
ADAMS Connection m
Diagnostics
~ unebndaed Menu

r Unp1ck

(" Box

(" Circl.

1

Me.xim.U1Il. • 11

Hini"wa • 1

Mode No.. Z

r. Lise of Ieeas

r Min" Max.. Inc:

OK Apply

Resee Cancel

~ANSYS Multiph ics Util

Eile :;>elect list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSYS Toolbar

®

IB Preprocessor
B Solution
m Analysis Type
B oeftne loads
m Settin9s
B Apply
8 Structural
m Displacement
8 Force/Moment
Fl On Keypoints
~ iMA@1'
Fl On Node (ORIpol
1:1 from Reactions
1:1 from Mao Analy
!!! Pressure
III Tenlperature
III Inertia
~ Pretnsn Sectn
~ Gen Plane Strain
I±I Other
I±I Field Surface Jntr
I±I Field Yolume: Jntr I±I
Initial Condit'n
IB Load Vector
I±I functions
ttl Delete
IB Operate
I±I Load Step Opts
m SE Management (eMS)
~ Results Traclcing
m Solve
m Manual Rezoning
m Multi-foeld Set Up
m ADAMS Connection
m Diagnostics
~ unebndaed Menu

Apply as

If Constant value then:

VALUE Force/moment value

OK Apply C.ncel

IPick a menu lem or enter an All5YS Command (SOIUTION)

Elle :;ieleet list elot Plot~trls :!iorl4'tone Pllt-.... tlaCro Me!luCtris tlelp

ANSVSToolbar ®

SAVE_OB RESUM_DBIQlJITI POWRGRI'HI

HI Pre;processor
8 Solution
iii Analysis Type
8 Define loads
iii Settings
8 Apply
8 Structural
iii Displacement
El Force/Moment
Fl On Keypolnts
Fl On Node CompO!
1:1 from Reactions
1:1 from Mao Anoly
lll Pressure
lll Temperature
l!l Inertia
~ Pretnsn Sectn
~ Gen Plane Strain
l±! other
l±! Field Surface Intr
l±l Field Yolunle Jntr
l±l Initial Condit"n
l±l Load Vector
l±l Functions
ttl Delete
IB Operate
l±! Load Step Opts
l±! SEManagement (CMS)
~ Results Tracking
IB Solve
IB Manual Rezoning
IB Multi-field Set Up
IB ADAMS Connection
IB Diagnostics
~ unebndced Menu

Pick a menu lern or enter an AH5YSComnond (SOlUTJOO)

~ ANSYS Mulliphyslcs Ulilily Menu - ~~1:El

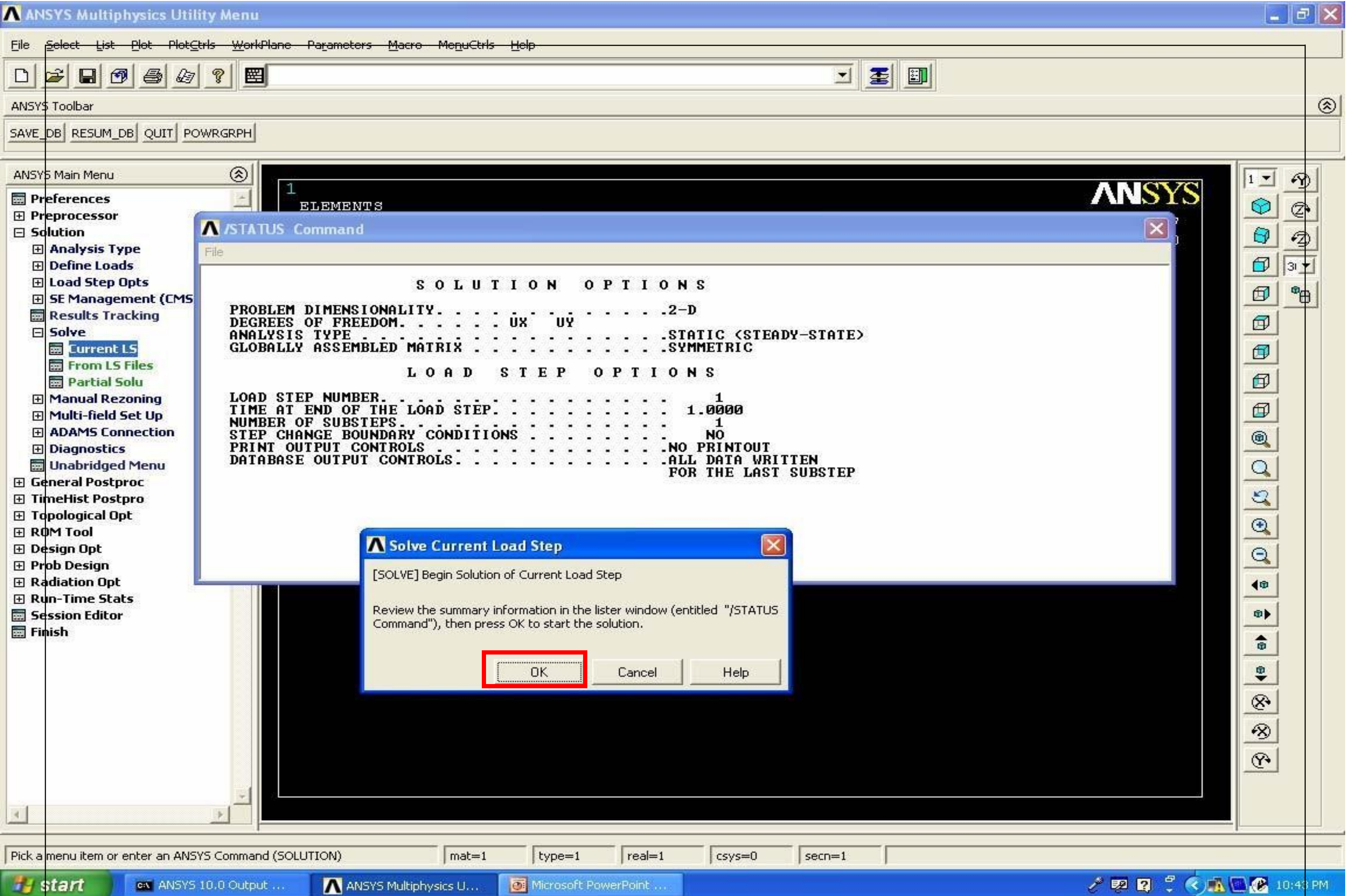
EUe 2elect bl.t elot ~ ytoricl>lono PO(omet.... Macro Meo.uCtris t[elp

ANSVS Toolbar ®

ffi Preprocessor
S Solution
ffi Analysis Type
ffi Define loads
ffi load step Opts
ffi Sf Management (CMS)
~ Results Tracking
~ CurrentLS
~ From lS Files
~ Partial Solu
(B Manual Rezoning
ffi Multi-field Set Up
(±) ADAMS Connection
III Diagnostics
1:1 Unabridged Menu
III General Postproc
III TlmeHist Postpro
III Topological Opt
III ROMTool
III Design Opt
III Prob Design
III Radiation Opt
III Run-Time Stat.
1:1 Session Editor
1:1 Finish

Pick a menu lern or enter an IWSYS Corrwnand (SOWTION)

10·43 PM


ANSYS Multiphysics Utility Menu
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
ANSYS Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
ANSYS Main Menu
Preferences
Preprocessor
Solution
Analysis Type
Define Loads
Load Step Opts
SE Management (CMS
Results Tracking
Solve
Current LS
From LS Files
Partial Solu
Manual Rezoning
Multi-field Set Up
ADAMS Connection
Diagnostics
Unabridged Menu
General Postproc
TimeHist Postpro
Topological Opt
ROM Tool
Design Opt
Prob Design
Radiation Opt
Run-Time Stats
Session Editor
Finish
ELEMENTS
ANSYS
/STATUS Command
File
S O L U T I O N O P T I O N S
PROBLEM DIMENSIONALITY. 2-D
DEGREES OF FREEDOM. . . . . . UX UY
ANALYSIS TYPE . . . . .STATIC (STEADY-STATE)
GLOBALLY ASSEMBLED MATRIX . . . . .SYMMETRIC
L O A D S T E P O P T I O N S
LOAD STEP NUMBER. 1
TIME AT END OF THE LOAD STEP. 1.0000
NUMBER OF SUBSTEPS. 1
STEP CHANGE BOUNDARY CONDITIONS . . . . NO
PRINT OUTPUT CONTROLS . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . .ALL DATA WRITTEN
FOR THE LAST SUBSTEP
Solve Current Load Step
[SOLVE] Begin Solution of Current Load Step
Review the summary information in the lister window (entitled "/STATUS Command"), then press OK to start the solution.
OK
Cancel
Help
Pick a menu item or enter an ANSYS Command (SOLUTION)
mat=1
type=1
real=1
csys=0
secn=1
start
ANSYS 10.0 Output ...
ANSYS Multiphysics U...
Microsoft PowerPoint ...
10:43 PM

The window above shows the input that is used in preprocessor and solution part. Make sure that the data given above are correct and then press OK.

'V Solution is done!

Preferences
Preprocessor
13 Solution
Analysis Type
Define loads
IB load Step Opts
S Sf Management
~ Results Tracking
13 Solve
e 3iihA';ILi
~ From lS Files
~ Partial Solu
IE Manual Rezoning
IB Multi-field Set Up
ADAMS Connection
Diagnostics
1:1 Unabridged Menu
General Postproc
TimeHlst Postpro
Topological Opt
ROM Tool
Oeslgn Opt
III Prob Oesign
III Radiation Opt
Run-Time Stats
~ Session Editor
~ Finish

SOLUTION OPTIONS

PROBLEM DIMEHSIONALITY [redacted] 2-D
DEGREES OF FREEDOM UX UY
ANALYSIS TYPE STATIC (STEADY-STATE)
GLOBALLY ASSEMBLED MATRIX SYMMETRIC

LOAD STEP OPTIONS

LOAD STEP NUHBER 1
TIME AT END OF THE LOAD STEP [redacted] 1-0000
NUMBER OF SUBSTEPS 1
STEP CHANGE BOUNDARY CONDITIONS NO
PRINT OUTPUT CONTROLS NO PRINTOUT
DATABASE OUTPUT CONTROLS ALL DATA WRITTEN
FOR THE LAST SUBSTEP

~ ANSYS Multiphysks Utilily Menu - ~~118

Eile aelect !,1st !!lot Plotl::trls ~or1<l'lano Pot_or. Macro Me[luCtrls t[elp

8 **Preprocessor**
IB Solution
8 General Posl:proc
~ Data & file Opts
~ Results Summary
IB Read Results
HI Failure Criteria
8 Plot Results
~ Deformed Shape
tt m'.!i!'yrn!
IB Vector Plot
IB Plot Path Item
(±) **Concrete Plot**
IB ThinfUm
IB Ust Results
IB Query Result.
1:1 Options for Outp
1:1 Results Viewer
1:1 Write PGR file
I1J Nodal Cales
I1J Element Table
I1J Path Operation.
I1J Surface Operation.
I1J Load Ca.e
I1J Check Elem Shape
1:1 Write Results
IB ROMOperations
IB Submodeling
IB fatigue
IB Safety factor
IB Defone/Modify
~ Nonlinear Diagnostlcs
~ Reset
IB Manual Rezoning
IB TuneHist Postpro
IB TopGlooical Opt

Pick a menulem or enter an AN5YS ConYMnd (POST!)

~ .:start ~ ~AN5YS~U ... rf.!l~ / "'-'llij ': |< .AQt':~lO:43PM

Eile 2elect bl.t elot ~ ytOflcl>lono Potomet.... Macro Meo.uCtris t[elp

ANSYS Toolbar

®

~ Contour Nodal Solution Data

rEl

Item to be contoured

~Favorites

rJi Nodal ScIWon

r$ OOF ScIWon

CZ'

CZ' YoC'"_- o Idsplacemert

CZ' Displacement vector sun

~ stress

Iii:! Total strain

1m Elastic Strain

Iii:! Plastic Strain

1m Creep Strain

Iii:! ThermalStrain

1m Total Mechanical and ThermalStrain

~ SwellinQtrain

...:J

..!.|

Undlsplaced shape key

ṛ Undlsplaced shape key IDeformed shape only

Scale Factor i-IA:""U7"'to:-C=-a7"'lc:-ul.-:t-ed-:----------:::J.,r----...;;;:;O

..:J

AdditionalOptions

ffi Preprocessor

ffi Solution

B General Postproc

~ Data II: File Opts

~ Results Summary

ffi Read Results

HI failure Criteria

B Plot Results

~ Deformed Shape

B Contour Plot

~ Wm'tbm

E:;J Element Solu

E:;J Elem Table

1:1 line Elem Res

I±I Vector Plot

I±I Plot Path Item

IE Concrete Plot

IE ThinFilm

IE List Results

IE Query Results

1:1 Options ror Outp

1:1 Result. Viewer

1:1 Write PGR File

IE Nodal Cales

IE Element Table

IE Path Operations

IE Surrace Operations

IE load Case

III Check aem Shape

~ Write Result.

ffi ROMOperation.

ffi Submodeling

ffi Fatigue

ffi Safety Factor

ffi DefonejModify

~ Nonlinear OiaQOOStics

IPick a menu lem or enter an IWSYS Corrwnand (POSTI)

~ ,; start ~.o~... ----- ~IWSYSfUJphysl(5U... rlB! ~licrosoitPowerPoot ... / :;rJ lij • ~ .A C)~ IO·44PM

~ ANSYS Multiphysics Utility Menu - [J@~

!:ile ::ielect list ~lot Plotgrls !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar ®

IB Preprocessor
W Solution
a General Postproc
£ill Data lie file Opts
£ill Results Summary
W Read Results
IB failure Criteria
a Plot Results
El Deformed Shape
8 Contour Plot
Cllftth'tnm
1:1 Element solu
1:1 Elem Table
!:I line flem RIO.
I!I Yector Plot
I!I Plot Path Item
l±l Concrete Plot
l±l ThInfilm
l±l **List Results**
l±l **Query Results**
~ Options for Outp
1:1 **Results Viewer**
~ Write PGRfile
l±l Nodal Calcs
l±l Element Table
ttl **Path Operations**
ttl **Surface Operations**
IB Load Case
l±l Check flem Shape
~ Write Results
W ROM Operations
l±l submodeling
l±l fatigue
l±l Safety factor l±l
DefinejModify r!lllonlinear
IliaQnostics

IB Preprocessor
W Solution
a General Postproc
£ill Data lie file Opts
£ill Re.sults Summary
W Read Results
III failure Criteria
III Plot Results
III list Results
III Query Results
CII Options ror Outp
CII Results Viewer
E!!! Write PGR File
I!I Nodal Calcs
El Element Table
~ •thfttj]bn"
~ Plot Elem Table
~ list Elem Table
~ Abs Value Option
~ Sum or Each Item
~ Add Items
~ Multiply
1:1 **Find MaMinlum**
1:1 **Find Mininlum**
~ EKponentiate
~ Cross Product
~ Dot Product
~ Erase Table
8 **Path Operations**
fB **Surface Operations**
W load(ase
W Check Elem Shape
I!'il Write Results
W ROM Operations
W SUbmodeIing
Wfatioue

Add ...

Upd te

c.. ,ently Oefrned Data and Status:

rmestarro status

lB Preprocessor
W Solution
a General Postproc
£ill Data lie file Opts
£ill Re.sults Summary
W Read Results
III failure Criteria
III Plot Results
III list Results
III Query Results
CII OptIons for Outp
CII Results Viewer
El!I Write PGR File
I!I Nodal Calcs
El Element Table
~ •thfttj Ibn"
~ Plot Elem Table
~ list Elem Table
~ Abs Value Option
~ Sum of Each Item
~ Add Items
~ Multiply
1:1 Find MaMinlum
1:1 Find Mininlum
~ EKponentiate
~ Cross Product
~ Dot Product
~ Erase Table
8 Path Operations
fB Surface Operations
W load(ase
W Check Elem Shape
I!'il Write Results
W ROM Operations
W SUbmodeling
Wfatioue

[ETAa£j Del... AdcItlonol E~ Table Items

lab Usor IabeIf .. bm

nem,,- R'"*' dotalom

SMISC,
NMISC

(For "By sequence nom", enter sequence
no. in Selection box. See Table 4.xx-3
in Elements Manual for seq. numbers.)

OK Apply Cancel Help

~iphYSics Utility Menu

E.

IB Preprocessor
iii Solution
8 General Po~tproc EI
Data lk File Opts EI
Results Summary iii
Read Results
iii Failure Criteria
iii Plot Results
iii list Results
iii Query Results
m Options ror Outp
m Results Viewer
m Write PGR File
eJ Nodal Calcs
8 Element Table
e "tiffinm
!!il Plot Elem Table
!!il List Elem Table
~ Abs Value Option
~ Sum or Each Item
~ Add Items
~ Multiply
~ **find Maxinlum**
F:l **find Miniolum**
!!il EKponentiate
~ Cross Product
~ Dot Product
~ Erase Table
ttl **Path Operations**
ttl **Surrace Operations**
IE load Case
iii Check Elem Shape
!'!l Write Results
iii ROM Operations
iii SUbmodeling
IilFatioue

label	Item	CorrG	Trnestamo	status
	IS	I	Trne- 1.0000	(Cu"ent)

Add ... Update Delete


~ANSYS Multiph ics Util
Eile :;>elect list elot Plotgr1s !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp
ANSVSToolbar
®
IB Preprocessor
W Solution
a General Postproc
£ill Data lie file Opts
£ill Results Summary
W Read Results
III failure Criteria
a Plot Results
El Deformed Shape
8 Contour Plot
El Nodal Solu
E:l Element Solu
q '@lmnm
1:1 Une [110mRes
!!l Yector Plot
III Plot Path Item
III Concrete Plot
III Thlnfilm
I±I List Results
I±I Query Results
~ Options for Outp
1:1 Results Viewer
~ Write PGR file
III Nodal Calcs
III Element Table
ttl Path Operations
ttl Surface Operations
IB Load Case
B Check Elem Shape
~ Write Results
S ROM Operations
S Submodeling
S Fatigue
S Safety factor S
DefinejModi(y r!lllonlinear
OiaQnostics
Itiab Item to b. plotted
Avofab Averaoe at common nodes?
OK
Apply
Cancel

~ ANSYS Multiphysics Utility Menu

!:ile ::ielect list ~lot Plotgrls !!£orl4'lone Pao.:_s Macro Me!luCtris tlelp

ANSVS Toolbar

IB Preprocessor
W Solution
a General Postproc
£ill Data lie file Opts
£ill Results Summary
W Read Results
IB failure Criteria
a Plot Results
El Deformed Shape
8 Contour Plot
El Nodal Solu
E:l Element Solu
cl @imnm
1:1 Une [110m Res
!!l Yector Plot
!!l Plot Path Item
l±l Concrete Plot
l±l ThInfilm
l±l **List Results**
l±l **Query Results**
~ Options for Outp
1:1 **Results Viewer**
~ Write PGR file
l±l Nodal Calcs
l±l Element Table
ttl **Path Operations**
ttl **Surface Operations**
IB **Load Case**
l±l Check Elem Shape
~ Write Results
W ROM Operations
l±l Submodeling
l±l fatigue
l±l Safety factor l±l
DefinejModi(y r!lllonlinear
IliaQnostics

E~e :ielect bist eiot Plotr...trls l/io,kPl.no p.t.melers Mocro MenuCb1s -

.:Jll~lr

ANSVS Toolba,

SAVE_DBI RESUM_DBI QUITI POWRGRPHI

~ Preferences
IB Preprocessor
ffi Solution
S General Postproc
~ Data 8< File Opts
~ Results Summary
ffi Read Results
ffi Failure Criteria
ffi Plot Results
S Ust Results
~ Detailed Summary
~ iteration Summry
~ Percent Error
ffi Sorted Listing
~ Nodal Solution
~ Element Solution
~ Section Solution
~ Superelem OOF
~ SpotWeld Solution
~ 'MiMlm
Ei!l Nodal Load.
Ei!l Elem Table Oatil
Ei!l Vector Oatil
c:l Path Item.
c:l Linearized Strs
III Query Result.
~ Options ror Outp
c:l Result. Viewer
c:l Write PGR File
III Nodal Cales E1
Element Table IB
Path Operations
m Surrace Operations
IB Load Case
IB Check Elem Shape
~ Write Results
I!l ROMOperations

l'icka menu lem or enler anANSY5 COIMWIIld(POST!)

®
J

Eile :i.elect list eJot Plot!;;trls l!'I.orl<Plane Pa(ameters Macro Me!luCtris !:ielp

e

E'!! Preferences
!!! **Preprocessor**
!!J **Solution**
13 General Postproc
~ Data & file Opts
~ **Results Sunlmary**
S Read Results
S Failure Criteria
S Plot Results
13 List Results
~ Detailed Summary
~ Iteration Summry
~ **Percent Error**
l±l **Sorted listing**
~ Nodal Solution
E'!! Element Solution
E'!! Section Solution
E'!! Superelem DDf
E'!! SpotWeld Solution
!! ;M'M#i1m
E'!! Nodal loads
E'!! Elem Table Data
E'!! Vector Data
E'!! Path Items
~ **linearized Strs**
S Query Results
~ Options for Dutp
~ Results Viewer
~ Write PGRfile
S Nodal Calcs
S Element Table
ttl **Path Operations**
ttl **Surrace Operations**
IB Load Case
S Check Elem Shape
~ Write Results
III ROM Operations

```
PRINT REACTION SOLUTIONS PER NODE

 ***** POST1 TOTAL REACTION SOLUTION LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIHE=    1.0000      LOAD CASE=   0

 THE FOLLOWING X,Y,Z SOLUTIONS ARE IN THE GLOBAL COORDINATE SYSTEH

    NODE       FX            FY
       1  -11100.0        0.0000

TOTAL UALUES
UALUE   -1000.0        0.0000
```

<u>Comparison between Ansys and Theoretical results</u>

	Ansys	Theoretical
Deformation	0.01mm	0.01mm
Stress	2 N/mm^2	2 N/mm^2
Reaction	-1000N	-1000N

9 Finite Element Analysis Programs

9.1 Overview

Computer implementation of finite elements and solution procedures for engineering analysis is addressed. The end product is a general-purpose finite element analysis program. For such software to be used as an effective CAE tool, the programming should be hardware independent. The chosen finite elements and numerical methods must be accurate and reliable. The program should be executable on a given platform of choice – single processor, multi-processor, parallel processor, etc.

A general purpose FEA program consists of three modules: a pre-processor, a solver, and a post-processor. Commercial FEA programs can handle very large number of nodes and nodal degrees of freedom provided a powerful hardware is made available. User's manual, theoretical manual, and verification problems manual, document a commercial FEA program.

Surveys of general-purpose programs for finite element analysis have been published [9.1]. At present FEA programs are used rather than written. Understanding of the organization, capabilities, and limitations of commercial FEA programs is generally more important than an ability to develop or even modify a FEA code. The emphasis on programming the FEM which was a major preoccupation in many recent textbooks [9.2 to 9.4] is therefore absent in this book.

The purpose of this chapter is to describe the organization and desirable capabilities of a general-purpose FEA program. A brief description of widely distributed and extensively used commercial FEA codes is included so that the reader is aware of their current capabilities.

Benchmark constitutes a standard set of test problems devised to assess the performance of FEA codes.

The practical issue of developing a viable FEA program and its implementation in the PC environment is a much larger challenge. Typically, it involves hundreds of human year's effort.

9.2 FEA Program: Organization

The four components shown in Fig. 9.1 are common to virtually all general-purpose FEA programs. The INPUT phase enables the user to provide information relating to geometric representation, finite element discretization, support conditions, applied loads, and material properties. The more sophisticated commercial FEM systems facilitate automated generation of nodes and elements and provide access to a material property database. Plotting of the finite element model is also possible so that errors if any, in the input phase, may be detected and corrected prior to performing computations.

The finite element library comprises the element matrix generation modules. Herein resides the coded formulative process for the individual finite elements. Ideally, the element library is open-ended

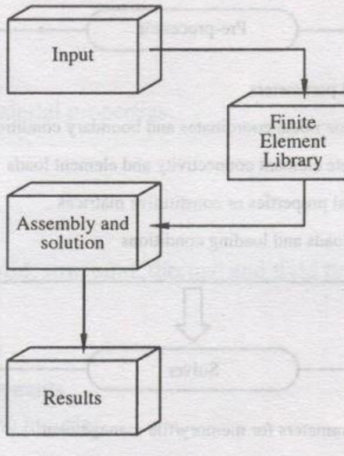


Fig. 9.1 *Components of a general purpose finite element analysis program*

and capable of accommodating new elements to any degree of complexity. This phase generates the required element matrices and vectors.

The assembly module includes all matrix operations necessary to position the element matrices for connection to neighbouring elements and the connection process itself. The latter operation thereby produces the global matrix equation of the finite element model.

The solution phase operates on the governing matrix equation of the problem derived in the previous phase. In the case of a linear static analysis, this may mean no more than the solution of a set of linear algebraic equations for a known right-hand side. In the case of linear vibration and buckling analysis, this may mean the extraction of eigen values and eigen vectors. Transient response analysis will require computations over a time history of applied load.

Finally, the results phase provides the analyst with a record of the solution. The record is commonly a printed list of nodal d.o.f, element strains and stresses, reaction forces corresponding to constrained degrees of freedom and a host of other requested information. As in input phase, there is a trend toward graphical output of results such as plots of displacement and stress contours, modes of vibration and buckling, etc.

A commercial FEM system therefore consists of three basic modules: pre-processor; solver; and post-processor. These modules and their functions are illustrated in Fig. 9.2. The pre-processor allows the user to create geometry or input CAD geometry, and provides the tools for meshing the geometry. The solver takes the finite element model provided by the pre-processor and computes the required response. The post-processor takes the data from the solver and presents it in a form that the user can understand.

9.3 FEA Program: Capabilities

The desirable features of a general-purpose FEA program are a large number of material models; a good library of finite elements; a good number of analysis procedures; and ability to manage the associated data. A brief discussion on these follows.

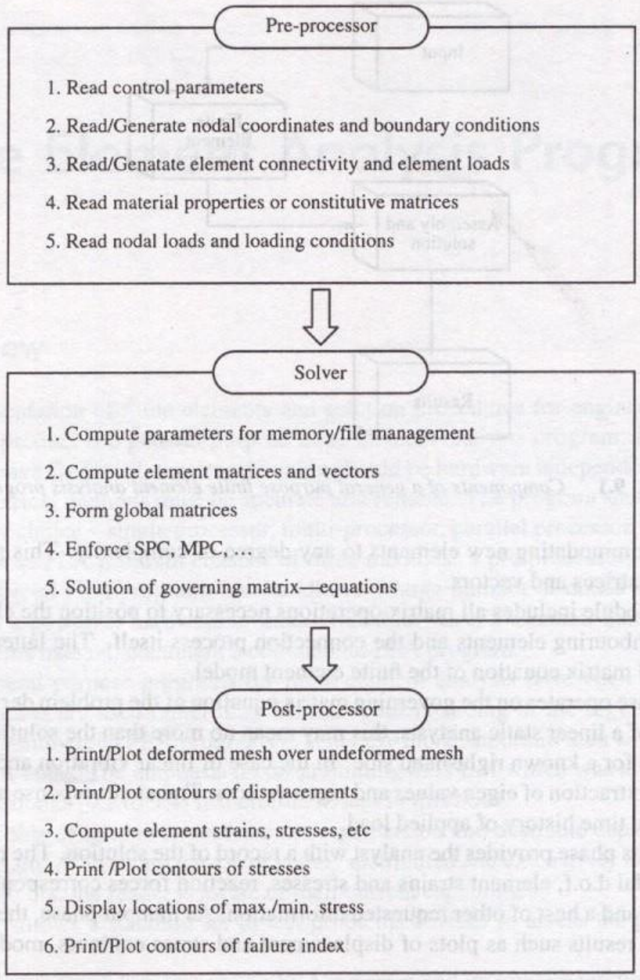


Fig. 9.2 *Finite element analysis program—modules and their functions*

9.3.1 Material models

To cover a large number of metallic and non-metallic materials and a wide range of their behaviour, a general-purpose FEA program should provide a library of material models.

- Homogeneous, isotropic, linear, elastic
- Orthotropic
- Anisotropic
- Laminated composite
- Nonlinear elastic
- Elastic plastic

- Viscoelastic
- Viscoplastic
- Hyperelastic
- Temperature-dependent material properties

9.3.2 Element library

The available elements are for solid, structural, thermal and fluid flow analysis. They can be classified as follows:

- One-dimensional elements
  - 1-D, 2-D, 3-D bar elements
  - Linear/quadratic/cubic in order
- Two-dimensional elements
  - Triangular/quadrilateral in shape
  - Linear/quadratic/cubic in order
  - With straight/curved edges
- Axisymmetric ring elements
  - Triangular/quadrilateral in shape
  - Linear/quadratic/cubic in order
  - With flat/curved surfaces
- Three-dimensional elements
  - Tetrahedra/hexahedra/pentahedra in shape
  - Linear/quadratic/cubic in order
  - With flat/curved faces
- Beam elements
  - Euler–Bernouli theory/shear deformation theory
  - 1-D, 2-D, 3-D beam elements
- Plate elements
  - Kirchoff theory/Mindlin theory
  - Triangular/quadrilateral shapes
  - Linear/quadratic/cubic in order
  - With straight/curved edges
- Shell elements
  - Flat shell elements/facet approximation
  - Curved shell elements: triangular/quadrilateral shapes; quadratic/cubic orders
  - Axisymmetric shell elements: with curved surfaces; linear/quadratic/cubic in order

- Special elements
  - Spring
  - Gap
  - Rigid link
  - Contact
- Crack tip elements

Some of these elements are formulated to handle large displacements, large rotations and finite strains. Some formulations use reduced integration with hourglass control.

9.3.3 Procedures library

- Linear static analysis
- Linear dynamic analysis
  - Free vibration
  - Forced vibration
  - Transient response: mode superposition
  - Transient response: direct integration
  - Acoustic excitation and response
  - Spectrum response
- Linear buckling analysis
- Non linear analysis
  - Geometric nonlinearity
  - Material nonlinearity
  - Combined geometric and material nonlinearity
  - Contact problems
- Aero-elastic analysis
  - Divergence
  - Flutter
- Design optimization (sensitivity analysis)
- Thermal analysis: computational
- Fluid dynamics: computational
- Fracture mechanics: computational
- Electromagnetics
- Electrostatics
- Magnetostatics

This allows the user to perform a wide variety of analyses. These procedures provide solutions for linear or nonlinear behaviour under static or dynamic loads. Large deformation and finite strain problems, contact problems, can also be addressed using these procedures.

9.3.4 Data processing

- Super elements
- Automated multilevel sub structuring
- Fourier analysis: axisymmetric bodies/shells under non-axisymmetric loads
- Cyclic symmetry
- Efficient numerical methods
  - Direct solver
  - Iterative solver
- Efficient computer systems
  - Super computers
  - Parallel processing systems
- Automatic adaptive mesh refinement

9.4 FEA Program: A Catalogue

A brief description of widely distributed commercial FEA programs is included here so that the reader is aware of their current capabilities.

9.4.1 MSC.Nastran

NASA Structural Analysis (Nastran) is a general-purpose program based on the finite element method developed by MacNeal Schwendler Corporation (MSC). The associated pre- and post-processor is called *MSC.Patran*. This premier FEA software is now available on the PC and runs both on DOS and Windows operating systems.

MSC.Patran provides the industry's most comprehensive and powerful tools for the creation of accurate finite element models. Backed by the world's largest CAE support organization and enhanced by continual use at some of the largest manufacturers, MSC.Patran sets the standard for finite element pre- and post-processing.

MSC.Nastran is the world standard in finite element analysis solutions. Its analysis capabilities give the user the competitive edge. With open choice of platforms from desktop PCs to supercomputers, MSC.Nastran is available where it is needed. MSC.Nastran's unique element technologies provide highly accurate results with lower modelling effort, less solution time, and reduced computer requirements. Using MSC.Nastran one can optimize designs without increasing design cycle time. MSC provides the best documentation, customer support, and user training.

Building better products lighter, stronger, safer, in less time, at less cost are the business benefits of FEA using Nastran.

Analysis procedures in MSC.Nastran include: structural statics; structural dynamics; heat transfer; aero-elastic; magnetic field; piezo electric; acoustic; and hydro-elastic.

9.4.2 NISA

Numerically Integrated Finite Elements for Systems Analysis (NISA) is a family of general-purpose finite element programs for PCs, workstations and supercomputers developed by Engineering Mechanics Research Corporation (EMRC). The associated pre- and post-processor is called DISPLAY. The distinguishing features of the NISA programs are: user-friendly documentation; excellent technical support; flexible purchase options; and best price/performance in the industry.

NISA offers independent modules for a variety of analysis: linear statics: nonlinear statics, dynamics; heat transfer; composites; optimization; fatigue and fracture; fluid dynamics; printed circuit boards; electromagnetic fields; kinematic and dynamic analysis of mechanical systems.

NISA provides an excellent library of isoparametric finite elements. A special module NISA.P ADAPT utilizes *P* elements. This program continually increases the order of the polynomial on a fixed finite element mesh until a reasonable convergence is reached. P refinement and properly designed mesh is efficient and reliable.

NISA offers interfaces to major CAD/CAM systems: pro/engineer; unigraphics; CATIA.

DISPLAY is a powerful interactive graphics pre- and post-processor, which makes complex finite element modelling and results interpretation a cinch.

These programs reflect the latest advances in CAE utilizing finite element methods.

9.4.3 MARC

The right answer for finite element analysis is the general-purpose program called MARC developed by MARC Analysis Research Corporation. Special features of this program are: fully integrated nonlinear solution; powerful automated 3-D contact; accurate, adaptive simulations; parallel processing; and multi-physics. The associated pre- and post-processor is called *Mentat*.

MARC and Mentat allow the user to perform a wide variety of structural, thermal, fluid, and coupled field analyses using finite element method. The analysis procedures provide solutions for simple to complex linear and nonlinear problems in engineering.

The capabilities in MARC include: linear; nonlinear; large deformation and finite strain; automated contact; and adaptive meshing.

MARC has an extensive library of metallic and nonmetallic material models: linear elastic; elastic plastic; elastomers; hyperelastic; rigid plastic flow; creep; viscoelastic; viscoplastic; poro-elasticity and soils; powder metallurgy; composites; and concrete.

Over 140 elements are available in MARC, which are modern, robust and accurate. They are grouped as: truss; beam; plane stress; plane strain; generalized plane strain; plate; shell; membrane; axisymmetric; 3-D solid; special elements (springs, gaps, rigid links, pipe bend, etc.); and user defined elements.

Analysis types supported by MARC are: statics; dynamics; heat transfer; thermo mechanical; fracture mechanics; fluid dynamics; hydrodynamic bearing; joule heating; acoustics; electrostatics; magnetostatics; electromagnetics; design sensitivity and optimization.

Mentat is tightly integrated with the MARC FEA program, allowing all data to be defined interactively through a powerful graphical user interface. Notable capabilities include: geometry creation; solid modelling; mesh generation; analysis support; post-processing; and advanced rendering.

Mentat's optional modules support interfaces to the leading CAD/CAM systems CATIA; pro/engineers; I-DEAS, and auto CAD.

9.4.4 LS-DYNA

LS-DYNA is a general-purpose code based on the FEM for analyzing large/elastic/inelastic deformation dynamic response of solids and structures including structures coupled to fluids. The main solution procedure is based on explicit time integration. An implicit solver is also available with somewhat limited capabilities for structural and heat transfer analysis.

A contact impact algorithm allows difficult contact problems to be easily treated with heat transfer included across the contact interfaces.

Spatial discretization is achieved by the use of four-node tetrahedral, eight-node hexahedral solid elements; two-node beam elements; three-node triangular and four-node quadrilateral shell elements; eight-noded solid shell elements; truss elements; membrane elements; discrete elements; and rigid bodies. A variety of formulations are available for each element type (solid, fluid, structural, discrete).

Specialized capabilities for modelling airbags, sensors, and seat belts have tailored LS-DYNA for applications in the automotive industry.

Adaptive meshing is available for shell elements and is widely used in sheet metal stamping simulations.

LS-DYNA currently has over two hundred material models and over ten equations of state to cover a wide range of material behavior.

LS-DYNA is operational on supercomputers, mainframes, workstations, parallel processing systems, and PCs.

The associated pre- and post-processor is called LS-TAURUS.

LS-DYNA and LS-TAURUS are developed by Livermore Software Technology Corporation.

9.4.5 ANSYS

ANSYS is an integrated design analysis tool based on the FEM developed by ANSYS, Inc. It has its own tightly integrated pre- and post-processor. The ANSYS product documentation is excellent and it includes commands reference; operations guide; modeling and meshing guide; basic analysis procedures guide; advanced analysis guide; element reference; theory reference; structural analysis guide; thermal analysis guide; electromagnetic fields analysis guide; fluid dynamics guide; and coupled field analysis guide. Taken together, these manuals provide descriptions of the procedures, commands, elements, and theoretical details needed to use the ANSYS program. All of the above manuals except the ANSYS theory reference are available online through the ANSYS help system, which can be accessed either as a standalone system or from within the ANSYS program. A brief description of the information found in each of the manuals follows.

Engineering capabilities of ANSYS products are: structural analysis (linear stress, nonlinear stress, dynamic, buckling); thermal analysis (steady state, transient, conduction, convention, radiation, and phase change); CFD analysis (steady state, transient, incompressible, compressible, laminar, turbulent); electromagnetic fields analysis (magnetostatics, electrostatics); field and coupled field analysis (acoustics, fluid–structural, fluid–thermal, magnetic–fluid, magnetic–structural, magnetic–thermal, piezoelectric, thermal–electric, thermal–structural, electric–magnetic); sub-modelling; optimization; and parametric design language.

Element library in ANSYS lists 189 finite elements. They are broadly grouped into: LINK, PLANE, BEAM, SOLID, CONTAC, COMBIN, PIPE, MASS, SHELL, FLUID, SOURCE, MATRIX, HYPER, VISCO, INFIN, INTER, SURF, etc. Under each type, different shapes and orders complete the list. Obviously, ANSYS has the best elements in its library.

Analysis procedures in ANSYS can be grouped into: static analysis; transient analysis; mode frequency analysis; harmonic response analysis; buckling analysis; sub-structuring analysis; and spectrum analysis.

In ANSYS, there are two fundamentally different types of optimization. The first is referred to as design optimization; it works entirely with the ANSYS parametric design language and is contained within its own module (ANSYS /OPT). The second is topology optimization, a form of shape optimization.

ANSYS finite element analysis software enables engineers to perform the following tasks:

- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study physical response, such as stress levels, temperature distributions or electromagnetic fields.
- Optimize a design early in the product development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible.

9.5 Closure

Spectacular advances have been made in the development, documentation, and implementation of commercial FEA programs on PCs, workstations, mainframes, and supercomputer systems. Pre-processors with graphical user interface are also available that can create finite element models of virtually all CAD models. Post-processors are capable of display and animation of the results of every finite element analysis. At present, FEA programs have been integrated in widely used CAD/CAM systems. Computer implementation of finite element procedures is not trivial; it involves hundreds of human years effort not only for development but also for updates.

It is instructive to compare and contrast the desirable features of a general purpose FEA program with the current capabilities of commercial FEA codes. This may provide directions for modifications, extensions and upgrading of commercial FEA codes.

It is recommended that the reader use one of the commercial FEA programs, not necessarily from those described here, to analyze the computational problems listed in the text. This will enable the user to acquire the skills needed to effectively use the FEM in general, and a general-purpose program in particular, in practice.

Advanced applications of the FEM, not considered so far, can be attempted using commercial FEA programs. Some of these are identified and described in the next chapter.

9.6 References

[9.1] Berbbia, C.A., editor. *Finite Element Systems: A Handbook*. Springer Verlag, Berlin, 1985.

[9.2] Krishnamoorthy, C.S. *Finite Element Analysis: Theory and Programming*. Tata McGraw Hill, New Delhi, 2nd ed., 1994.

[9.3] Reddy, J.N. *An Introduction to the Finite Element Method*. McGraw Hill, New York, 2nd ed., 1993.

[9.4] Chandrupatla, T.R. and Belegundu, A.D. *An Introduction to Finite Elements in Engineering*. Prentice Hall of India, New Delhi, 2nd ed., 2001.