Chapter 1

Introduction

1.1 Introduction to Classical Method

An exact solution in analytical form (i.e., exact solution formula) of plate bending problems using classical methods is limited to relatively simple plate geometry, load configuration, and boundary supports. If these conditions are more complicated, the classical analysis methods become increasingly tedious or even impossible. In such cases, approximate methods are the only approaches that can be employed for the solution of practically important plate bending problems. Nevertheless, the classical solutions remain very valuable because they enable one to gain insight into the variation of stresses and strains with basic shape and property changes, and they provide an understanding of the physical plate behavior under an applied loading. In addition, they can be used as a basis for incisively evaluating the results of approximate solutions through quantitative comparisons and order of magnitude bounds.

In continuum mechanics, plate theories are mathematical descriptions of the mechanics of flat plates that draws on the theory of beams. Plates are defined as plane structural elements with a small thickness compared to the planar dimensions. The typical thickness to width ratio of a plate structure is less than 0.1. A plate theory takes advantage of this disparity in length scale to reduce the full three-dimensional solid mechanics problem to a two-dimensional problem. The aim of plate theory is to calculate the deformation and stresses in a plate subjected to loads.

Of the numerous plate theories that have been developed since the late 19th century, two are widely accepted and used in engineering. These are

- The Kirchhoff theory of plates (classical plate theory)
- The Mindlin–Reissner theory of plates (first-order shear plate theory)

1.1.1 <u>Kirchhoff theory of plates (classical plate theory)</u>

The Kirchhoff theory of plates is a two-dimensional mathematical model that is used to determine the stresses and deformations in thin plates subjected to forces and moments. This theory is an extension of Euler-Bernoulli beam theory and was developed in 1888 by Love[1using assumptions proposed by Kirchhoff. The theory assumes that a mid-surface plane can be used to represent a three-dimensional plate in two-dimensional form.

The following kinematic assumptions that are made in this theory:

1. Straight lines normal to the mid-surface remain straight after deformation.
2. Straight lines normal to the mid-surface remain normal to the mid-surface after deformation.
3. The thickness of the plate does not change during a deformation.

1.1.2 Mindlin–Reissner theory of plates

The Mindlin-Reissner theory of plates is an extension of Kirchhoff plate theory that takes into account shear deformations through-the-thickness of a plate. The theory was proposed in 1951 by Raymond Mindlin. A similar, but not identical, theory had been proposed earlier by Eric Reissner in 1945. Both theories are intended for thick plates in which the normal to the mid-surface remains straight but not necessarily perpendicular to the mid-surface. The Mindlin-Reissner theory is used to calculate the deformations and stresses in a plate whose thickness is of the order of one tenth the planar dimensions while the Kirchhoff-Love theory is applicable to thinner plates.

The form of Mindlin–Reissner plate theory that is most commonly used is actually due to Mindlin and is more properly called Mindlin plate theory. The Reissner theory is slightly different. Both theories include in-plane shear strains and both are extensions of Kirchhoff-Love plate theory incorporating first-order shear effects.

Mindlin's theory assumes that there is a linear variation of displacement across the plate thickness but that the plate thickness does not change during deformation. This implies that the normal stress through the thickness is ignored; an assumption which is also called the plane stress condition. On the other hand, Reissner's theory assumes that the bending stress is linear while the shear stress is quadratic through the thickness of the plate. This leads to a situation where the displacement through-the-thickness is not necessarily linear and where the plate thickness may change during deformation. Therefore, Reissner's theory does not invoke the plane stress condition.

The Mindlin-Reissner theory is often called the first-order shear deformation theory of plates. Since a first-order shear deformation theory implies a linear displacement variation through the thickness, it is incompatible with Reissner's plate theory.

1.2 Finite Difference Method

FDM makes **point wise approximation** to the governing equations i.e. it ensures continuity only at the node points. Continuity along the sides of grid lines are not ensured. It do not give the values at any point except at node points. It do not give any approximating function to evaluate the basic values (deflections, in case of solid mechanics) using the nodal values. It makes stair type approximation to sloping and curved boundaries. It needs larger number of nodes to get good results while FEM needs fewer nodes. With FDM fairly complicated problems can be handled.

1.3 Numerical method

The limitations of human mind are such that it cannot grasp the behavior of its complex surroundings and creations in one operation. Thus the process of sub dividing all systems into their individual components or elements, whose behavior is readily understood, and then rebuilding the original system from such components to study its behaviors in a natural way in which the engineer, scientist or even the economist proceeds.

In many situations and adequate model is obtained using a finite number of well-defined components. We shall term such problems discrete. In others the sub-divisions is continued indefinitely and the problem can only be defined using the mathematical fiction on an infinitesimal. This leads to differential equations or equivalent statements which imply an infinite number of elements. We shall term such systems continuous.

With the advent of digital computers, discrete problems can generally be solved readily even if the number of elements is very large. As the capacity of all computers is finite, continuous problems can only be solved exactly by mathematical manipulation. Here, the available mathematical techniques usually limit the possibilities to oversimplified situations.

To overcome the intractability of realistic type of continuum problems, various methods of discretization have from time to time been proposed both by engineers an mathematicians. All involved an approximation which, hopefully, approaches in the limit the true continuum solution as the number of discrete variable increases. The discretization of continuous problems has been approached differently by mathematicians and engineers. Mathematicians have developed general techniques applicable directly to differential equations governing the problem, such as finite difference approximations, various weighted residual procedures, or approximate techniques for determining the stationary of properly defined functions. The engineer, on the other hand often approaches the problems more intuitively by creating an analogue between real discrete elements and finite portions of a continuum domain. For instance, in the field of solid mechanics Mc Henry, Hrenikoff, New mark and indeed Southwell in the 1940's, showed that reasonably good solution to an elastic continuum problem can be obtained by replacing small portions of the continuum by an arrangement of simple elastic parts.

Later, in the same context, Argyris and Turner et al. showed that a more direct, but no less intuitive, substitution of properties can be made much more affectively by considering that small portions or elements in a continuum behave in a simplified manner. It is from the engineering direct analogue view that the term finite element was born. Clough appears to be the first to use this term, which implies in it a direct use of a standard methodology to discrete systems. Both conceptually from the computation view point, this is of the utmost importance. The first allows an improved understanding to be obtained; the second offers a unified approach to the variety of problems and the developments of standard computation procedures.

Since the early 1960's much progress has been made, and today the purely mathematical and analogue approaches are fully reconciled. It is the object of this text to present a view of the FEM as general discretization procedures of a continuum problems posed by mathematically defined statements.

In the analysis of problems of a discrete nature, a standard methodology has been developed over the years. The civil engineers , dealing with the structures, first calculates force displacement relationship for each element of the structure and then proceeds to assemble the whole by following a well-defined procedure of establishing local equilibrium at each node or connecting point of the structure. The resulting equations can be solved for the unknown displacements.

All such analysis follows a standard pattern which is universally adaptable to discrete systems. It is thus possible to define a standard discrete system. The existence of a unified treatment of standard discrete problems leads us to a first definition of Finite element process as a method of approximations to continuum problems such that

(a) The continuum is divided into a finite numbers of parts, the behavior of which is specified by a finite number of parameters
(b) The solution of the complete system as an assembly of its elements follows precisely the same rules as those applicable to standard discrete problems.

Table shows the process of evolution which lead to the present day concepts of finite element analysis.

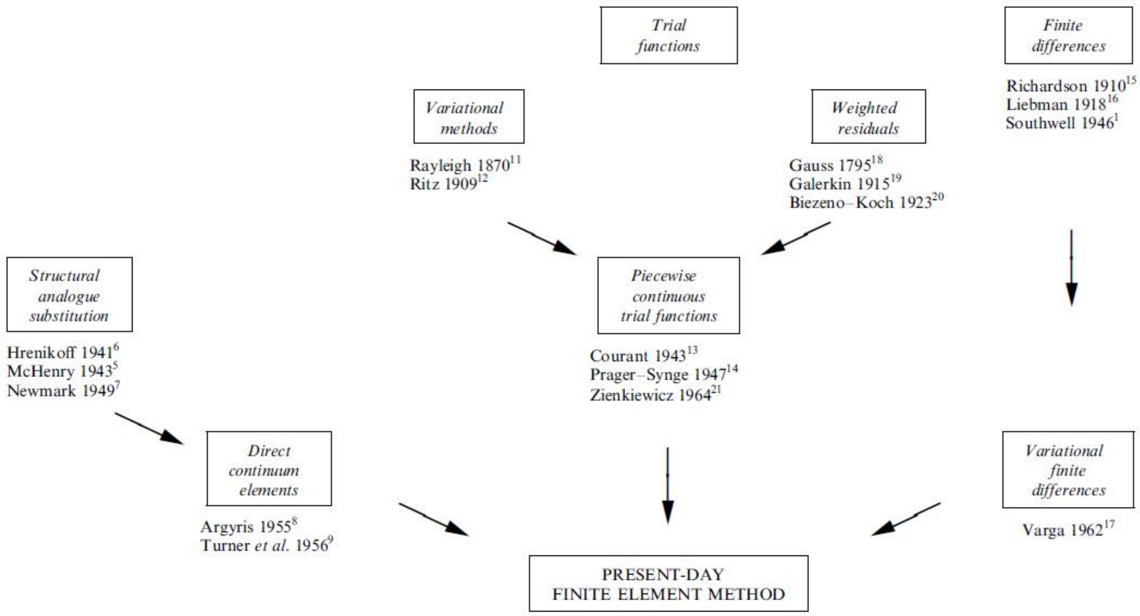

Figure1. Evolution of FEM

1.4 Literature Review

In the earlier period, some of the researchers have been dealt exact solutions of plate deflections with different boundary conditions. Bhattacharya (1986) investigated the deflection of plates under static and dynamic loads by using a new finite difference analysis. This approach gives the forth order bi-harmonic equation which varies from node to node and found the true mode shape of the plate at each node. Defu and sheikh (2005) have presented the mathematical approach for large deflection of rectangular plates. Their analysis, based on the two forth order and second degree partial differential Von karman equations, found lateral deflection to applied load. This solution can be used to direct practical analysis of plates with different boundary conditions. Bakker et al. (2008) have studied the approximate analysis method for large deflection of rectangular thin plate with simply supported boundary condition under action of transverse loads. This approach gives the shape of initial and total deflection of plates. From this analysis, the large deflection behaviour of plate under transverse load can be expressed as a function of the pre to post buckling in-plane stiffness of plate. Liew et al. (2001) developed the differential quadrature method and harmonic differential quadrature method for static analysis of three dimen-sional rectangular plates. This methodology can be used to found the bending and buckling of plates, which are simply supported and clamped boundary conditions only.

In the past, some researchers utilized FEA in solving problem plates with holes. Jain (2009) recently analyzed the effect of D/A ratio (where D is hole diameter and A is plate width) upon stress concentration factor and deflect-tion in isotropic and orthotropic plates under transverse static loading. Chaudhuri (1987) presented a theory for stress analysis by using rhombic array of alternating method for multiple circular holes. He worked on effects of stress concentration on a laminated plate with hole by finite element method. Paul and Rao (1989) presented a theory for evaluation of stress concentration factor of thick and FRP laminated plate with the help of Lo-Christensen-Wu higher order bending theory under transverse loading.

In the last few decades, authors have considered plates with stiffener. Although the stiffened rectangular plates have been thoroughly studied, the application of stiffeners to circular plates is not so popular.. Troipsky (1976) carried out his work for stiffened plates under bending, stability and vibration. Pape and Fox (2006) presented an infinite series approach for verity of plate aspect ratio by using stiffened elastic beam.

For smoothen the elements, Xuan et al. (2007) have used boundary integral method. Das et al. (2009) have developed a quite general method which can be applied to any classical boundary conditions. Jain (2009) presented analysis of stress concentration and deflection in isotropic and orthotropic rectangular plates with central circular hole under transverse static loading. He considered three types of elements to solve square plate problems with various boundary conditions and loadings. Shaiov and Vorus (1986) developed an integral equation formulation for an elasto-plastic plate bending. This technique is used for finding plasticity characteristics as well as the external lateral loading. Lot of work is available in deflection of plates under different loading conditions.

Chapter 2

Numerical Method

2.1 Introduction to FEM

Finite element analysis (FEA) is a powerful computational numerical technique used for solving engineering problems having complex geometries that are subjected to general boundary conditions. While the analysis is being carried out, the field variables are varied from point to point, thus, possessing an infinite number of solutions in the domain. So, the problem is quite complex. In this method all the complexities of the problems, like varying shape, boundary conditions and loads are maintained as they are but the solutions obtained are approximate. Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering. The fast improvements in computer hardware technology and slashing of cost of computers have boosted this method, since the computer is the basic need for the application of this method. A number of popular brand of finite element analysis packages are now available commercially. Some of the popular packages are STAAD-PRO, GT-STRUDEL, NASTRAN, NISA and ANSYS. Using these packages one can analyze several complex structures. The finite element analysis originated as a method of stress analysis in the design of aircrafts. It started as an extension of matrix method of structural analysis. Today this method is used not only for the analysis in solid mechanics, but even in the analysis of fluid flow, heat transfer, electric and magnetic fields and many others. Civil engineers use this method extensively for the analysis of beams, space frames, plates, shells, folded plates, foundations, rock mechanics problems and seepage analysis of fluid through porous media. Both static and dynamic problems can be handled by finite element analysis. This method is used extensively for the analysis and design of ships, aircrafts, space crafts, electric motors and heat engines.

While the analysis is being carried out, the field variables are varied from point to point, thus, possessing an infinite number of solutions in the domain. In FEA, the continuum is divided into a number of elements connected not only at their nodes but also along the hypothecated inter-element boundaries. The elements are joined only at nodes and some conditions at the boundaries are introduced to improve the accuracy. Each element is free to deform and can have different material and geometrical properties. The shape of the element depends upon the problem. From one-dimensional axial element to three dimensional solid elements are all in use.

The basis of the FEA can be divided in three steps as follows

- Select suitable field variables and the elements.

- Discritise the continua.
- Select interpolation functions.
- Find the element properties.
- Assemble element properties to get global properties.
- Impose the boundary conditions.
- Solve the system equations to get the nodal unknowns.
- Make the additional calculations to get the required values.

$$[\mathrm{K}]\{Q\} = \{F\}$$

[K] - Overall or global stiffness matrix
$\{Q\}$ - Global vector of unknown nodal displacements
$\{F\}$ - Vector of applied loads

Finite element analysis (FEA) has the following advantages and disadvantages while comparing with classical methods:-

2.1.1 Merits of FEM over Classical Approach

- In classical methods, exact equations are formed and exact solutions are obtained where as in finite element analysis exact equations are formed but approximate solutions are obtained.
- Solutions have been obtained for few standard cases by classical methods, whereas solutions can be obtained for all problems by finite element analysis.
- Whenever the following complexities are faced, classical method makes the drastic assumptions' and looks for the solutions:
  (a) Shape
  (b) Boundary conditions
  (c) Loading
  To get the solution in the above cases, rectangular shapes, same boundary condition along a side and regular equivalent loads are to be assumed. In FEM no such assumptions are made. The problem is treated as it is.

- When material property is not isotropic, solutions for the problems become very difficult in classical method. Only few simple cases have been tried successfully by researchers. FEM can handle structures with anisotropic properties also without any difficulty.

- If structure consists of more than one material, it is difficult to use classical method, but finite element can be used without any difficulty.
- Problems with material and geometric non-linearity’s cannot be handled by classical methods. There is no difficulty in FEM.
- It uses displacement functions for each element of the system, making it more adaptable to practical problems while other numerical methods (Rayleigh-Ritz) use displacement function for the entire structural system.

2.1.2 Demerits of FEM

- It requires requires enormous modelling effort and computing time for nonlinear analysis.
- It does not deal with continuum having buckling, distorted model and material non-linearity.
- It doesn’t deal with large deflection engineering problems.
- Only some specific numerical results are available for specified engineering problems.
- Hence FEM is superior to the classical methods only for the problems involving a number of complexities which cannot be handled by classical methods without making drastic assumptions. For all regular problems, the solutions by classical methods are the best solutions. In fact, to check the validity of the FEM programs developed, the FEM solutions are compared with the solutions by classical methods for standard problems.

2.2 Selection of elements

Based on the shapes elements can be classified as

- One dimensional elements
- Two dimensional elements
- Axi-symmetric elements
- Three dimensional elements.

One-dimensional elements:

These elements are suitable for the analysis of one dimensional problem and may be called as line elements also.

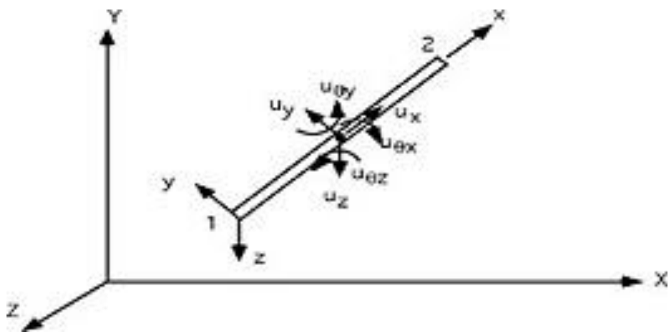

Figure2. Line Elements

Two-dimensional elements:

Common two dimensional problems in stress analysis are plane stress, plane strain and plate problems. Two-dimensional elements often used are three noded triangular elements shown in Fig. It has the distinction of being the first and most used element. These elements are known as Constant Strain Triangles (CST) or Linear Displacement Triangles. Six noded and ten noded triangular elements are also used by the analysts. Six noded triangular elements are known as Linear Strain Triangle (LST) or as Quadratic Displacement Triangle.

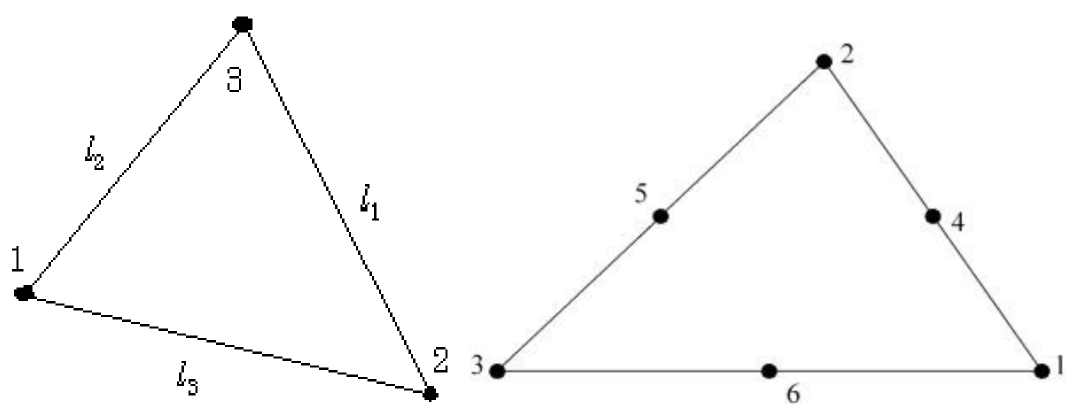

Figure3. Triangular Elements

A simple but less used two dimensional element is the four noded rectangular elements whose sides are parallel to the global coordinate systems. This system is easy to construct automatically but it is not well suited to approximate inclined boundaries.

Quadrilateral Elements are also used in finite element analysis. Initially quadrilateral elements were developed by combining triangular elements.

But it has taken back stage after isoparametric concept was developed. Isoparametric concept is based on using same functions for defining geometries and nodal unknowns. Even higher order triangular elements may be used to generate quadrilateral elements.

In isoparametric elements the geometry and field interpolation functions are of the same order. Using isoperimetric concept even curved elements are developed to take care of boundaries with curved shapes

Axi-symmetric elements

These are also known as ring type elements. These elements are useful for the analysis of axi-symmetric problems such as analysis of cylindrical storage tanks, shafts, and rocket nozzles. Axi-symmetric elements can be constructed from one or two dimensional elements. One dimensional axi-symmetric element is a conical frustum and a two dimensional axi-symmetric element is a ring with a triangular or quadrilateral cross section.

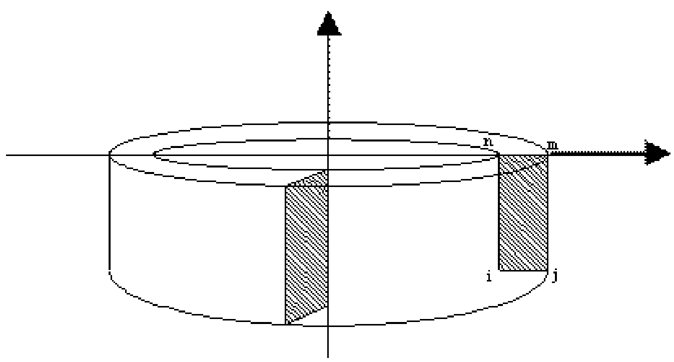

Figure 4. Axis-symmetric elements

Three dimensional elements:

Similar to the triangle for two dimensional problems tetrahedron is the basic element for three dimensional problems. Tetrahedron is having four nodes, one at each corner. Three dimensional elements with eight nodes are either in the form of a general hexahedron or a rectangular prism, which is a particular case of a hexahedron. The rectangular prism element is many times called as a brick element also. In these elements also one can think of using higher order elements

2.3 Discretizing the continua

Nodes are the selected finite points at which basic unknowns (displacements in elasticity problems) are to be determined in the finite element analysis. The basic unknowns at any point inside the element are determinedby using approximating/interpolation/shape functions in terms of the nodal values of the element.

There are two types of nodes viz. external nodes and internal nodes. External nodes are those which occur on the edges/ surface of the elements and they may be common to two or more elements. In Fig. below, nodes, 1 and 2 in one dimensional element, nodes 1 to 9 in 10 noded triangular element. These nodes may be further classified as (i) Primary nodes and (ii) Secondary nodes.

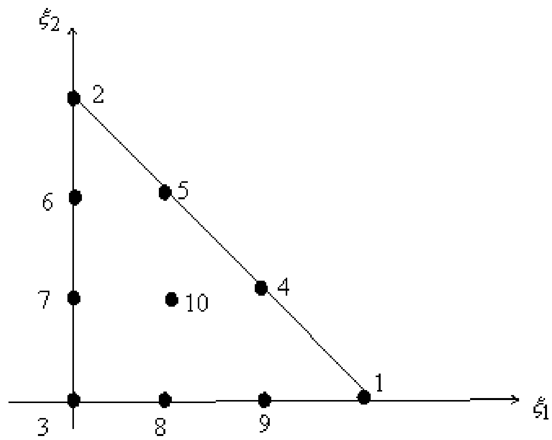

Figure 5. Types of nodes

Primary nodes occur at the ends of one dimensional element or at the corners in the two or three dimensional elements. Secondary nodes occur along the side of an element but not at corners. Fig. above shows such nodes.

Internal nodes are the one which occur inside an element. They are specific to the element selected i.e. there will not be any other element connecting to this node. Such nodes are selected to satisfy the requirement of geometric isotropy while choosing interpolation functions.

2.4 Co-ordinate system

The following terms are commonly referred in FEM

(i) Global coordinates

(ii) Local coordinates and

(iii) Natural coordinates

However there is another term 'generalized coordinates' used for defining a polynomial form of interpolation function. This has nothing to do with the 'coordinates' term used here to define the location of points in the element.

Global coordinates:

The coordinate system used to define the points in the entire structure is called global coordinate system. Figure below shows the Cartesian global coordinate system used for some of the typical cases.

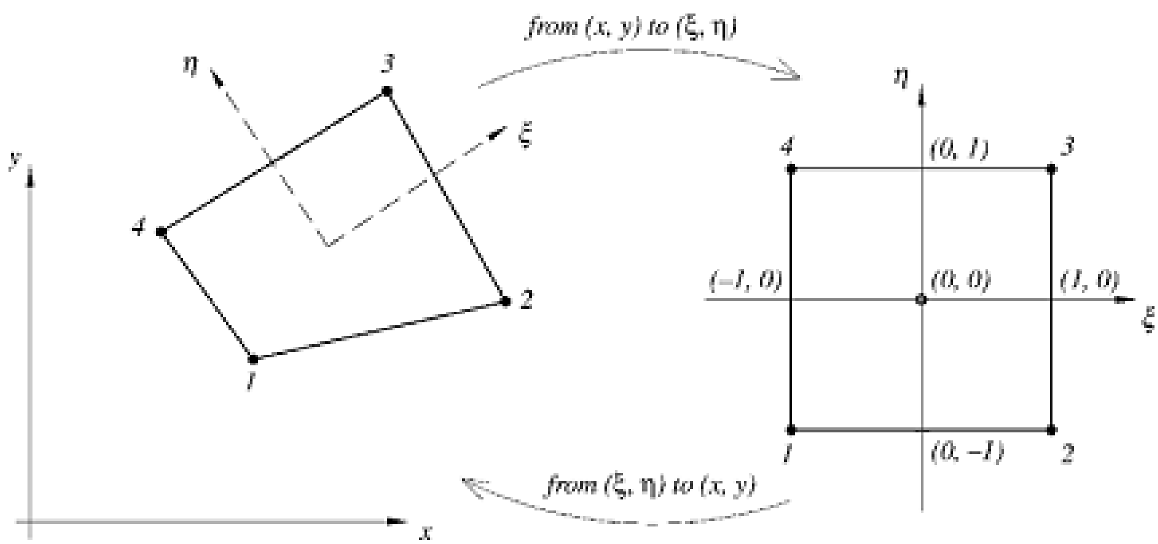

Figure 6. Global Coordinates

Local coordinates:

For the convenience of deriving element properties, in FEM many times for each element a separate coordinate system is used. For example, for typical elements shown in Fig. the local coordinates may be as shown in fig. However the final equations are to be formed in the common coordinate system i.e. global coordinate system only.

Natural coordinates:

A natural coordinate system is a coordinate system which permits the specification of a point within the element by a set of dimensionless numbers, whose magnitude never exceeds unity. It is obtained by assigning weightages to the nodal coordinates in defining the coordinate of any point inside the element. Hence such system has the property that i^{th} coordinate has unit value at node *i* of the element and zero value at all other nodes.

The use of natural coordinate system is advantages in assembling element properties (stiffness matrices), since closed form integrations formulae are available when the expressions are in natural coordinate systems.

Natural coordinate systems are used for one dimensional, two dimensional and three dimensional elements.

Natural coordinates for rectangular elements are as shown in Fig. In these cases the centroid of the area is the origin. The relationships between the local coordinates and the Cartesian coordinates are based on isoperimetric concept, which is taken up in the latter chapter. It may be noted here that the coordinates ξ and η vary from –1 to 1. The relationship between global coordinates and the natural coordinates are x = Σ Li xi and y = Σ Li yi. The derivation of Li is discussed in the chapter 'isoperimetric elements'. When the expressions are formed in these coordinate systems, instead of seeking integrations in the closed form expressions, numerical technique is usually employed.

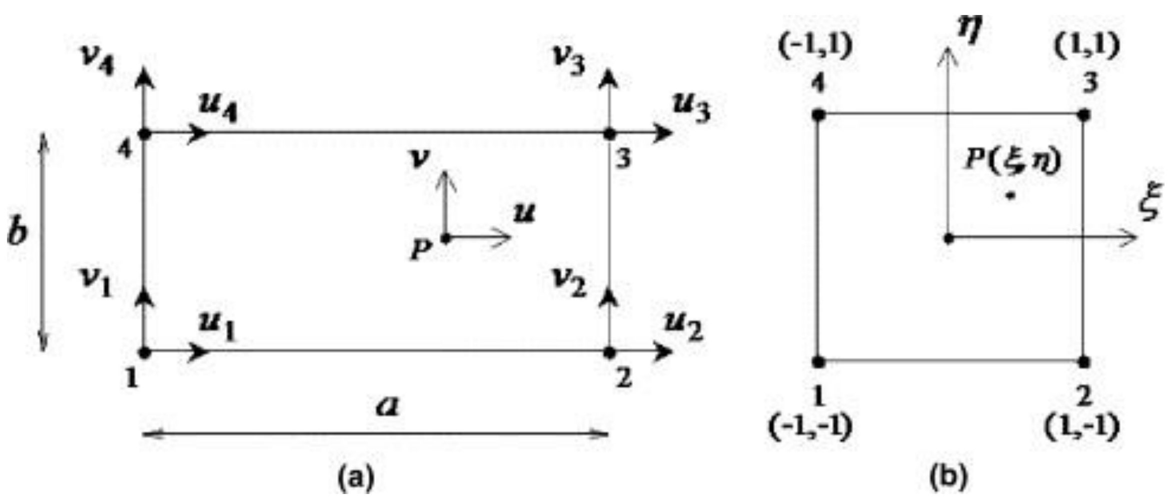

Figure 7. Natural Coordinates

2.5 Selecting interpolation functions

In the finite element analysis (FEA) aim is to find the field variables at nodal points by rigorous analysis, assuming at any point inside the element basic variable is a function of values at nodal points of the element. This function which relates the field variable at any point within the element to the field variables of nodal points is called shape function. This is also called as interpolation function and approximating function. In two dimensional stress analysis in which basic field variable is displacement,

$$u = \Sigma N_i u_i \quad , \quad v = \Sigma N_i v_i$$

Where summation is over the number of nodes of the element. For example for three noded triangular element,displacement at $P\ (x, y)$ is

$$u = \Sigma N_i u_i = N_1 u_1 + N_2 u_2 + N_3 u_{3.}$$

$$v = \Sigma N_i v_i = N_1 v_1 + N_2 v_2 + N_3 v_{3.}$$

i.e. $$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \qquad (6)$$

i.e. $\{\delta\} = [N]\{\delta\}_e$

$$2 \times 1 \quad 2 \times 6 \quad 6 \times 1$$

Where q is displacement at any point in the element

[N] Shape function , $\{\delta\}_e$ is vector of nodal displacements

In case of 4 noded rectangular element $\{\delta\} = [N]\{\delta\}_e$

$$2 \times 1 \quad 2 \times 8 \quad 8 \times 1$$

Shapes function for rectangular elements using Lagrange polynomials

If only continuity of basic unknown (displacement) is to be satisfied, Lagrange polynomials can be used to derive shape functions. Lagrange polynomial in one dimension is defined by

$$L_k(x) = \prod_{\substack{m=1 \\ m\neq k}}^{n} \frac{x - x_m}{x_k - x_m}$$

Equation above takes the value equal to zero at all points except at point *k*. At point *k* its value is unity. This is exactly the property required for the interpolation functions.

Although Lagrangian interpolation functions are for only one dimension, we may extend the concept to two and three dimensions by forming the product of the functions which hold good for the individual one dimensional coordinate directions i.e.,

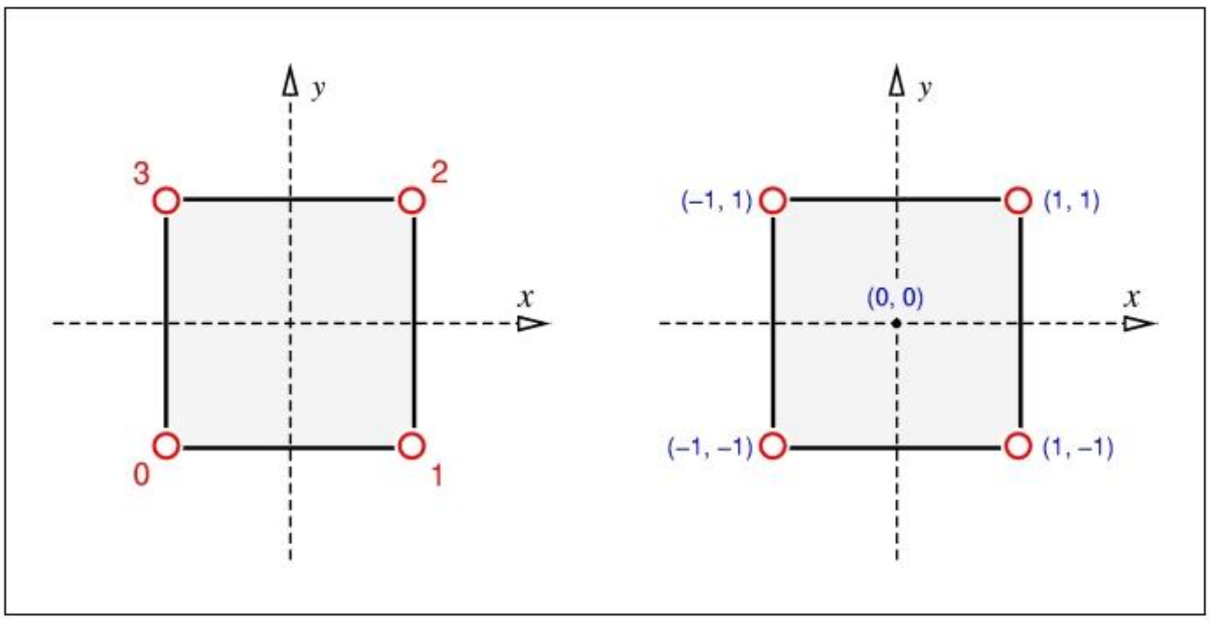

Figure 8. Typical 4 noded quadrilateral element

$$N_1 = L_1(\xi)L_1(\eta)$$

Thus for 4 noded rectangular element shown in Fig. 8

$$N_1 = L_1(\xi)L_1(\eta) = \frac{\xi - \xi_2}{\xi_1 - \xi_2}\frac{\eta - 4}{\eta_1 - \eta_4}$$

$= \frac{(\xi-1)}{-1-1}\frac{\eta-1}{-1-1}$ $\quad = \frac{1}{4}(1-\xi)(1-\eta)$ (1)

$N_2 = L_2(\xi)L_2(\eta) = \frac{\xi-\xi_2}{\xi_1-\xi_2}\frac{\eta-4}{\eta_1-\eta_4}$

$= \frac{(\xi-(-1)}{1-(-1)}\frac{\eta-1}{1-(-1)} = \frac{1}{4}(1+\xi)(1-\eta)$(2)

$N_3 = L_3(\xi)L_3(\eta) = \frac{\xi-\xi_2}{\xi_1-\xi_2}\frac{\eta-4}{\eta_1-\eta_4}$

$= \frac{(\xi+1)}{1-(-1)}\frac{\eta+1}{1-(-1)}$ $\quad = \frac{1}{4}(1+\xi)(1+\eta)$(3)

$N_4 = L_4(\xi)L_4(\eta) = \frac{\xi-\xi_2}{\xi_1-\xi_2}\frac{\eta-4}{\eta_1-\eta_4}$

$= \frac{(\xi-1)}{-1-1}\frac{\eta+1}{1-(-1)} = \frac{1}{4}(1-\xi)(1+\eta)$(4)

Thus, $N_i = \frac{1}{4}(1+\xi\xi_i)(1+\eta\eta_i)$(5)

2.6 Element stiffness matrix for isoparametric quadrilateral element

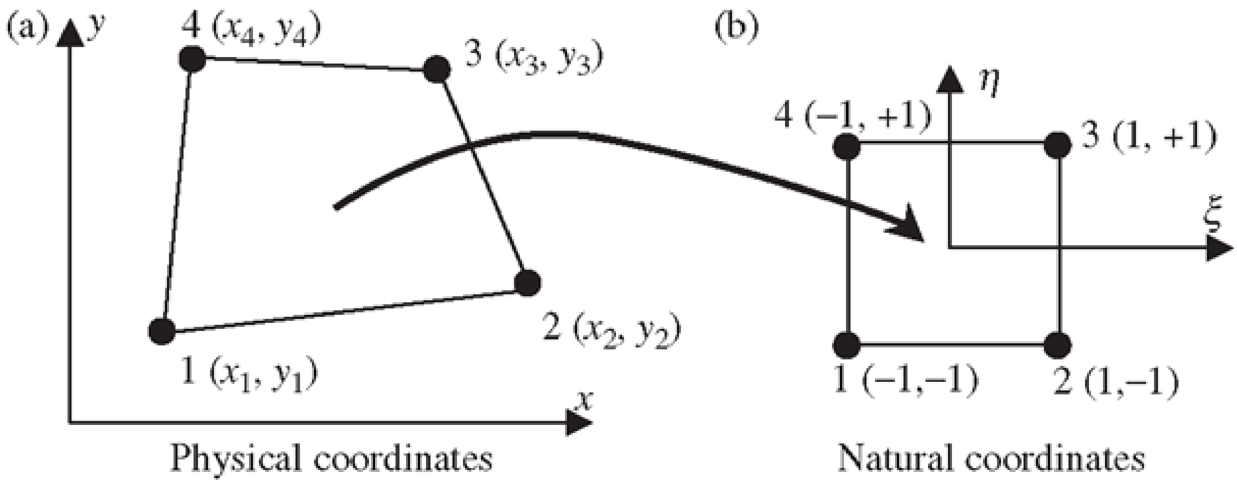

Figure 9. Physical coordinates to Natural Coordinates

Co-ordinate transformation:

Shape functions are used for defining deflection at any point in terms of the nodal displacement. Taig suggested use of shape function for coordinate transformation from natural local coordinate system to global Cartesian system and successfully achieved in mapping parent element to required shape in global system. Thus, the Cartesian coordinate of a point p(x,y) in an element may be expressed as

$$x = N_1x_1 + N_2x_2 + N_3x_{3.} + N_1x_4$$
$$y = N_1y_1 + N_2y_2 + N_3y_{3.} + N_1y_4$$

Or in matrix form$\{x\} = [N]\{x\}_e$,
Where N is shape functions and $\{x\}_e$are the coordinates of nodal points of the element. The shape functions are to be expressed in natural coordinate system.

Noting that shape functions are such that at node i, N_i= 1 and all others are zero, it satisfy the coordinate value at all the nodes. Thus any point in the quadrilateral is defined in terms of nodal coordinates.

For parent element, the shape functions are from equation (1) to (5),

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta),$$

$$N_2 = \frac{1}{4}(1+\xi)(1-\eta),$$

$$N_3 = \frac{1}{4}(1+\xi)(1+\eta),$$

$$N_4 = \frac{1}{4}(1-\xi)(1+\eta),$$

The above functions are used for defining the displacement as well as for defining the geometry of any point within the element in terms of nodal values.

When shape functions are used for the geometry, then

$$\begin{Bmatrix} x \\ y \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix} \begin{Bmatrix} x_1 \\ y_1 \\ x_2 \\ y_2 \\ x_3 \\ y_3 \\ x_4 \\ y_4 \end{Bmatrix}$$

The above relation helps to determine the (*x*, *y*) coordinates of any point in the element when the corresponding natural coordinates ξ and η are given.
Same shape functions are also used to define the displacement at any point P(x, y) in the element from equation (6)

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix}$$

In assembling the stiffness matrix we need the derivatives of displacements with respect to global *x*, *y* system. It is easy to find derivatives with respect to local co-ordinates ξ and η but it needs suitable assemblyto get the derivatives w.r.t. to global Cartesian system.

The relationship between the coordinates can be computed using chain rule of partial differentiation. Thus,

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

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

i.e. $$\begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{Bmatrix} \begin{Bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{Bmatrix} = [J] \begin{Bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{Bmatrix}$$

Where $$[J] = \begin{Bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{Bmatrix}$$

The matrix [*J*] shown above is called Jacobian matrix. It relates derivative of the function in local coordinate system to derivative in global coordinate system.

Let, $[J] = \begin{bmatrix} J_{11} & J_{12} \\ J_{21} & J_{22} \end{bmatrix}$

Where

$J_{11} = \frac{\partial x}{\partial \xi}$, $J_{12} = \frac{\partial y}{\partial \xi}$

$J_{21} = \frac{\partial x}{\partial \eta}$, $J_{22} = \frac{\partial y}{\partial \eta}$

We already know that

$$x = \sum_{i=1}^{4} = N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_{3.} + N_1 x_4$$

$$y = \sum_{i=1}^{4} = N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_{3.} + N_1 y_4$$

$$J_{11} = \frac{\partial \mathrm{x}}{\partial \xi} = \frac{\partial \mathrm{N}_1}{\partial \xi}\mathrm{x}_1 + \frac{\partial \mathrm{N}_2}{\partial \xi}\mathrm{x}_2 + \frac{\partial \mathrm{N}_3}{\partial \xi}\mathrm{x}_3 + \frac{\partial \mathrm{N}_4}{\partial \xi}\mathrm{x}_4$$

Similarly J_{12} , J_{21} and J_{22} can also be assembled.

Then we get

$$[J] = \begin{bmatrix} \sum_{i=1}^{4} \frac{\partial N_i}{\partial \xi} x_i & \sum_{i=1}^{4} \frac{\partial N_i}{\partial \xi} y_i \\ \sum_{i=1}^{4} \frac{\partial N_i}{\partial \eta} x_i & \sum_{i=1}^{4} \frac{\partial N_i}{\partial \eta} y_i \end{bmatrix}$$

For any specified point the above matrix can be assembled. Now,

$$\begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} = [J] \begin{Bmatrix} \frac{\partial}{\partial \mathrm{x}} \\ \frac{\partial}{\partial \mathrm{y}} \end{Bmatrix} ,$$

$$\begin{Bmatrix} \frac{\partial}{\partial \mathrm{x}} \\ \frac{\partial}{\partial \mathrm{y}} \end{Bmatrix} = [J]^{-1} \begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} J_{11}^* & J_{12}^* \\ J_{21}^* & J_{22}^* \end{bmatrix} \begin{Bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{Bmatrix} ,$$

Where J_{11}^* , J_{12}^* , J_{21}^* and J_{22}^* are the elements of Jacobian inverse matrix. Since for a given point Jacobian matrix is known its inverse can be calculated and Jacobian inverse matrix is assembled. With this transformation relation known, we can express derivatives of the displacements as shown below:

$$= \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} \end{Bmatrix} = \begin{bmatrix} J_{11}^* & J_{12}^* & & 0 \\ J_{21}^* & J_{22}^* & & \\ 0 & & J_{11}^* & J_{12}^* \\ & & J_{21}^* & J_{22}^* \end{bmatrix} \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial v}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{Bmatrix}$$

The strain displacement relation is given by

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \\ 0 & 1 & 1 & 0 \end{bmatrix} \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial u}{\partial y} \\ \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} \end{Bmatrix} = \begin{bmatrix} J_{11}^* \quad J_{12}^* \\ J_{21}^* J_{22}^* & 0 \\ 0 & J_{11}^* \quad J_{12}^* \\ & J_{21}^* J_{22}^* \end{bmatrix} \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial v}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{Bmatrix}$$

But$u = \Sigma N_i u_i$, $v = \Sigma N_i v_i$

$$\therefore \begin{Bmatrix} \frac{\partial u}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \\ \frac{\partial v}{\partial \xi} \\ \frac{\partial u}{\partial \eta} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} & 0 & \frac{\partial N_3}{\partial \xi} & 0 & \frac{\partial N_4}{\partial \xi} & 0 \\ \frac{\partial N_1}{\partial \eta} & 0 & \frac{\partial N_2}{\partial \eta} & 0 & \frac{\partial N_3}{\partial \eta} & 0 & \frac{\partial N_4}{\partial \eta} & 0 \\ 0 & \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} & 0 & \frac{\partial N_3}{\partial \xi} & 0 & \frac{\partial N_4}{\partial \xi} \\ 0 & \frac{\partial N_1}{\partial \eta} & 0 & \frac{\partial N_2}{\partial \eta} & 0 & \frac{\partial N_3}{\partial \eta} & 0 & \frac{\partial N_4}{\partial \eta} \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix}$$

Substituting it in equation above strain displacement matrix [B] is obtained as,

$$\{B\} = \begin{bmatrix} J_{11}^* \quad J_{12}^* \\ J_{21}^* J_{22}^* & 0 \\ 0 & J_{11}^* \quad J_{12}^* \\ & J_{21}^* J_{22}^* \end{bmatrix} \begin{bmatrix} \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} & 0 & \frac{\partial N_3}{\partial \xi} & 0 & \frac{\partial N_4}{\partial \xi} & 0 \\ \frac{\partial N_1}{\partial \eta} & 0 & \frac{\partial N_2}{\partial \eta} & 0 & \frac{\partial N_3}{\partial \eta} & 0 & \frac{\partial N_4}{\partial \eta} & 0 \\ 0 & \frac{\partial N_1}{\partial \xi} & 0 & \frac{\partial N_2}{\partial \xi} & 0 & \frac{\partial N_3}{\partial \xi} & 0 & \frac{\partial N_4}{\partial \xi} \\ 0 & \frac{\partial N_1}{\partial \eta} & 0 & \frac{\partial N_2}{\partial \eta} & 0 & \frac{\partial N_3}{\partial \eta} & 0 & \frac{\partial N_4}{\partial \eta} \end{bmatrix}$$

Then element stiffness matrix is given by

$$[K] = \oint [B]^T [D][B] dV$$

In this case,

$$[K] = t \iint [B]^T [D][B] \, \partial x \, \partial y$$

It can be shown that elemental area in Cartesian coordinates (x, y) can be expressed in terms of the area in local coordinates (ξ, η) as $\partial x\ \partial y = |J|\ \partial\xi\ \partial\eta$

Where $|J|$ is the determinant of the Jacobian.

$[K] = t \iint [B]^T [D][B] |J|\ \partial\xi\ \partial\eta$ (7)

Integration is to be performed so as to cover entire area i.e. the limit of integration is from ξ is form –1to 1 and η is also from –1 to 1. It is difficult to carry out all the multiplications in equation (7) and then the integration. It is convenient to go for numerical integration.

In finite element analysis Gauss quadrate formula is preferred since in these values at n sampling point scan be used to fit in $2n - 1$ degree variation, as the evaluation of functions like $[B]^T[D][B]dV$ is a time consuming process. In this method, the numerical integration is achieved by the following expression.

$$\int_{-1}^{1} f(\xi)\ \partial\xi = \sum_{i=1}^{n} W_i f(\xi)_i$$

Where W_i=weight function and $f(\xi)_i$ is values of the function at pre-determined sampling points.In Gauss quadrature formula sampling points are cleverly placed. In this, both n sampling points and weights are treated as variables to make exact 2n– 1 degree polynomial. This is an open quadrature formula, the function values need not be known at end points but they must be known at predetermined sampling points.

The location of sampling points ξ_i and weight function ω_i are determined using Legendre polynomials. Hence this method is sometimes called as Gauss Legendre quadrature formula. Table 13.1 shows location of Gauss sampling points ξ_i and corresponding weight function W_i for different number (n) of Gauss integration scheme.

Table 1 :The location of sampling points ξ_i and weight function W_i

n	Cusps	weights	n	cusps	weights
2	±0.57735	1	4	±0.339981	0.652145
				±0.861136	0.347855
3	0	8/9	5	0	0.568889
	±0.774597	5/9		±0.538469	0.478629
				±0.906180	0.236927

For two dimensional problem n = 2 means 2 × 2 = 4 Gaussian points and for three dimensional problems it works out to be 2 × 2 × 2 = 8. Thus,

$$\int_{-1}^{1}\int_{-1}^{1} f(\xi,\eta)\,\partial\xi\,\partial\eta = \int_{-1}^{1}\sum_{i=1}^{n} W_i f(\xi,\eta) d\eta$$

$$=\sum_{j=1}^{n} W_j\{\sum_{i=1}^{n} W_i f(\xi,\eta)\} = \sum_{j=1}^{n}\sum_{i=1}^{n} W_j W_i f(\xi,\eta)$$

Element force vector:

$$f^e = t_e\left[\int_{-1}^{1}\int_{-1}^{1} N^T \det J\, d\xi d\eta\right]\begin{Bmatrix} f_x \\ f_y \end{Bmatrix}$$

2.7 ANSYS analysis

Finite element analysis software ANSYS is a capable way to analyze a wide range of different problems. ANSYS can solve various problems such as elasticity, fluid flow, heat transfer, and electro-magnetism. Beside those, it can also do nonlinear and transient analysis. ANSYS analysis has the following steps for problem solving:

i. Modeling: includes the system geometry definition and material property selection. In this step user can draw either 2D or 3D representation of the problem.

ii. Meshing: this step involves discretizing the model according to predefined geometric element.

iii. Solution: this step involves applying boundary conditions and loads to the system and solves the problem.

iv. Post processing: this involves plotting nodal solutions (unknown parameters), which may be of displacements/stresses/reactive forces etc.

In this analysis, shell63 element is used with four nodes. Shell63 element type is having both bending and membrane capabilities and these elements permit both in-plane and normal loads. At each node, this element is having six degrees of freedom: three nodal translations and three rotations about x, y, and z directions. This element includes Stress stiffening and large deflection capabilities.

Chapter 3

Problem and Results

3.1 <u>Problem</u>

Clamped edge, simple support boundary and mixed boundary conditions are applied for the considered rectangular plate. The detailed description of the problem is described.

Uniform distributed load on thin plate:

Throughout the analysis, uniformly distributed load (pressure) of 50 Pa is applied on an isotropic square plate as shown in Figure. The variations in deflections of the plate with different boundary conditions are illustrated in Tables 2,3,4 and 5, when the plate thickness is varying in the range of 0.001 and 0.06.

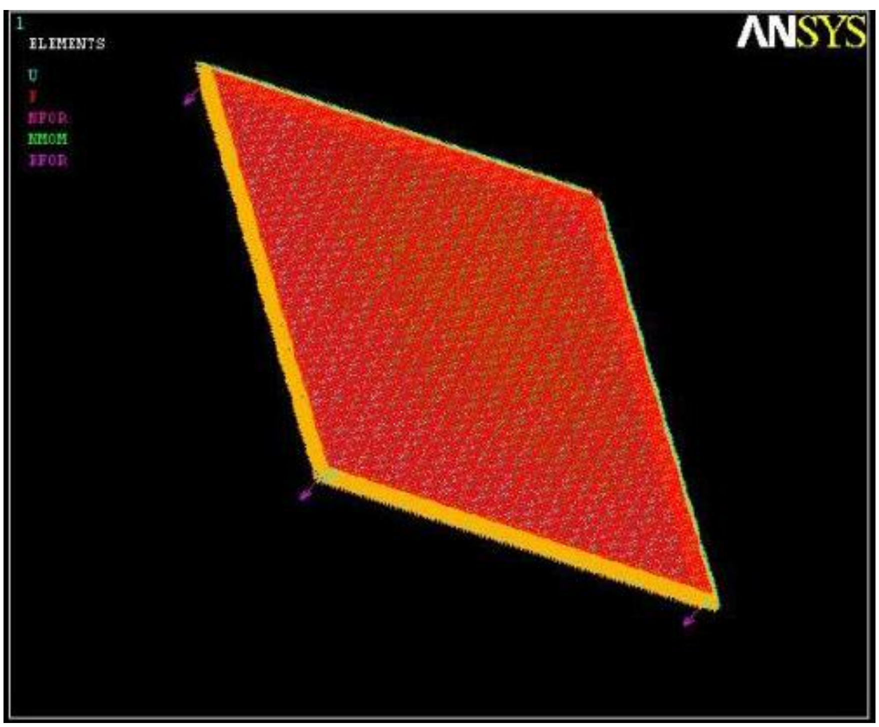

Figure 10. Uniformly Distributed Load on a plate in ANSYS

Concentrated load on thin plate:

Point load or concentrated load of 50 N is applied for different boundary conditions with same dimension using the stated case and the deflection of plate by varying thickness of plate is calculated. The deflection values are tabulated as shown in Table 6, 7, 8 and 9 when the plate thickness is varying in the range of 0.001 and 0.06

.

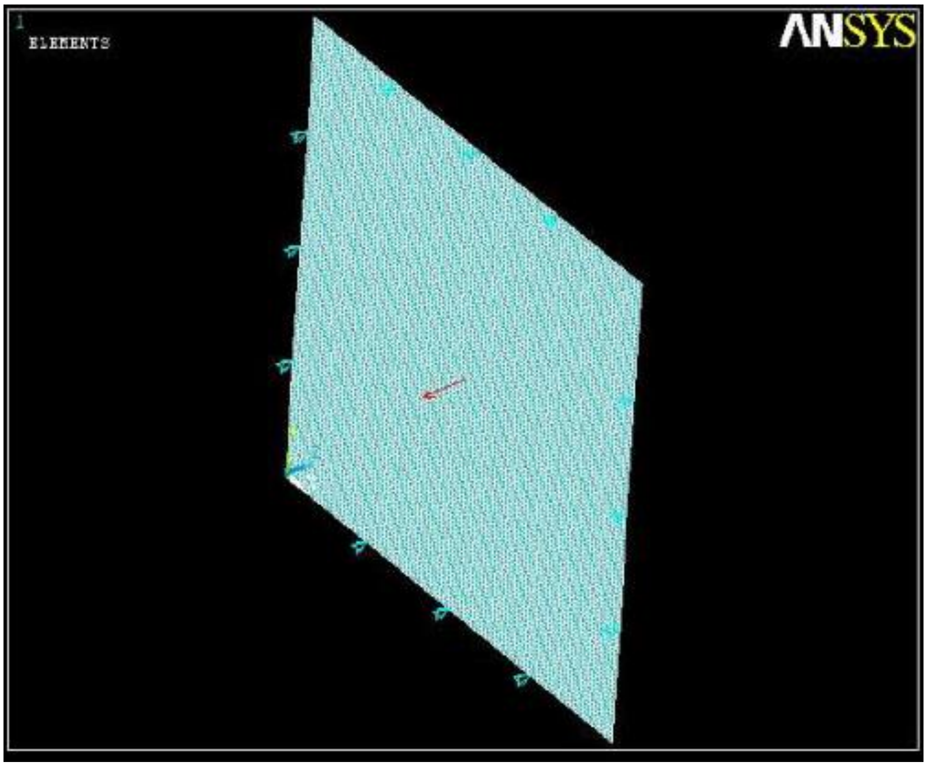

Figure 11. Concentrated Load at center of a Plate in ANSYS

Boundary Conditions:

Four types of boundary conditions has been used in this project which are listed below and the figures in ANSYS are as shown

1. All sides simply supported subjected (S-S-S-S)
2. Opposite sides clamped and simply supported (C-S-C-S)
3. Adjacent sides clamped and simply supported (C-C-S-S)
4. All sides clamped or fixed (C-C-C-C)

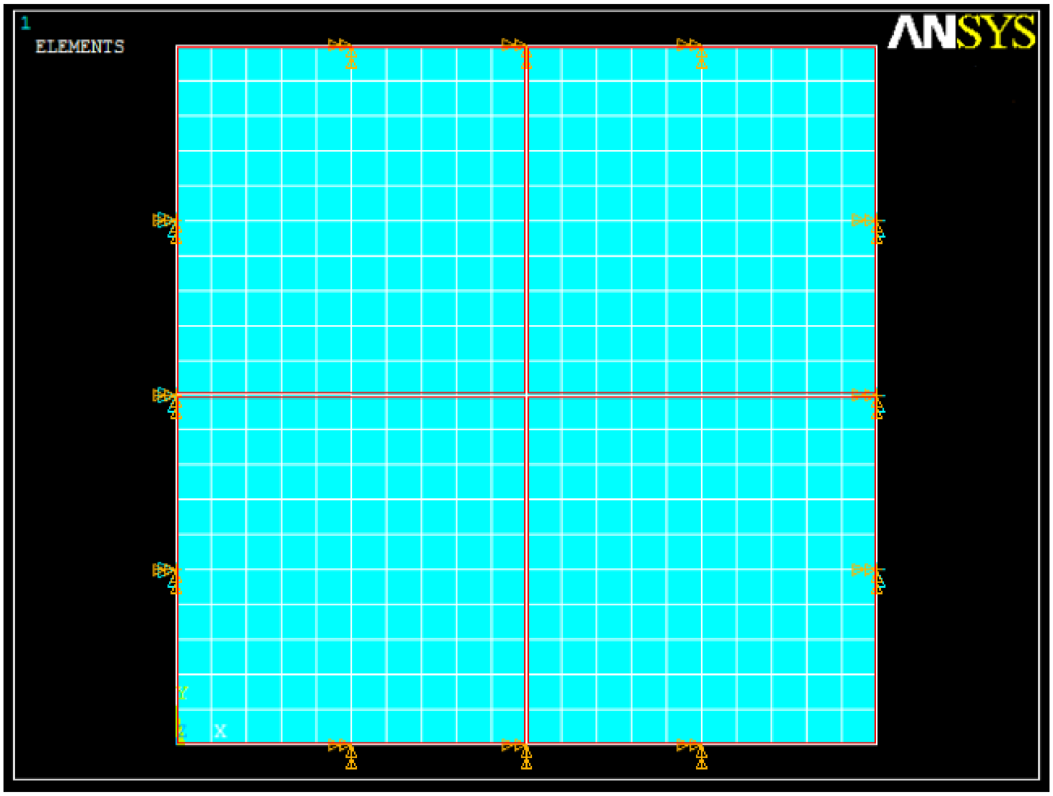

Figure 12. Boundary Condition: C-S-C-S

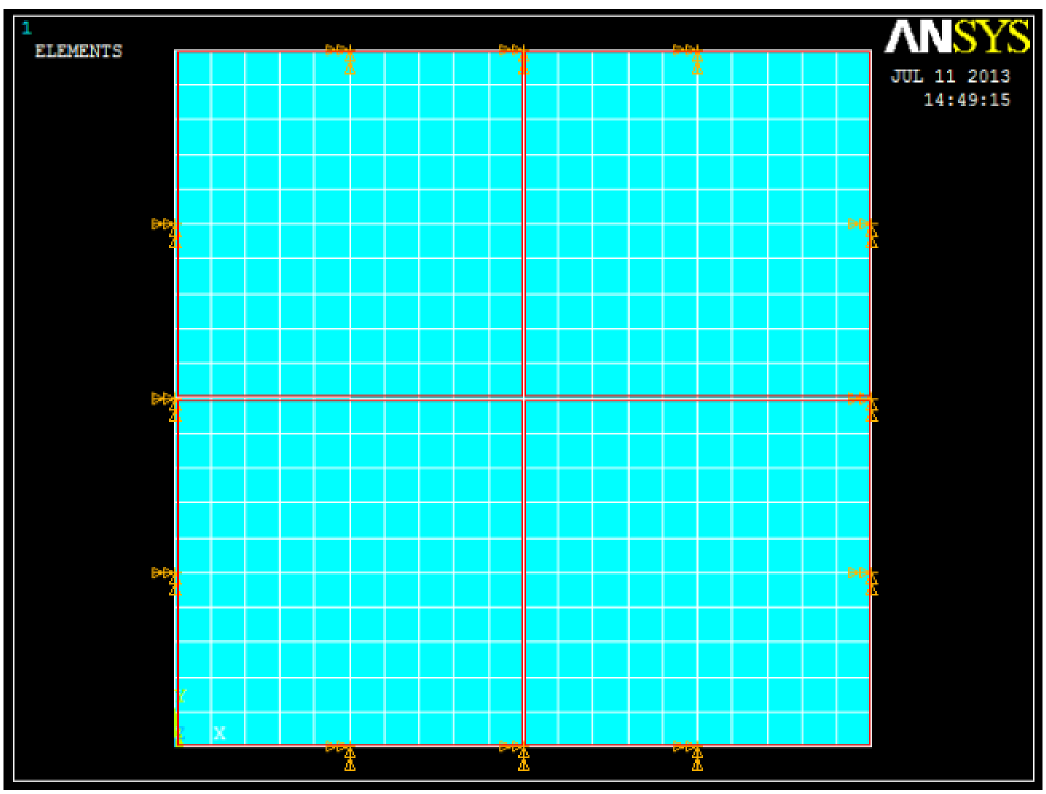

Figure 13. Boundary Condition: S-S-S-S

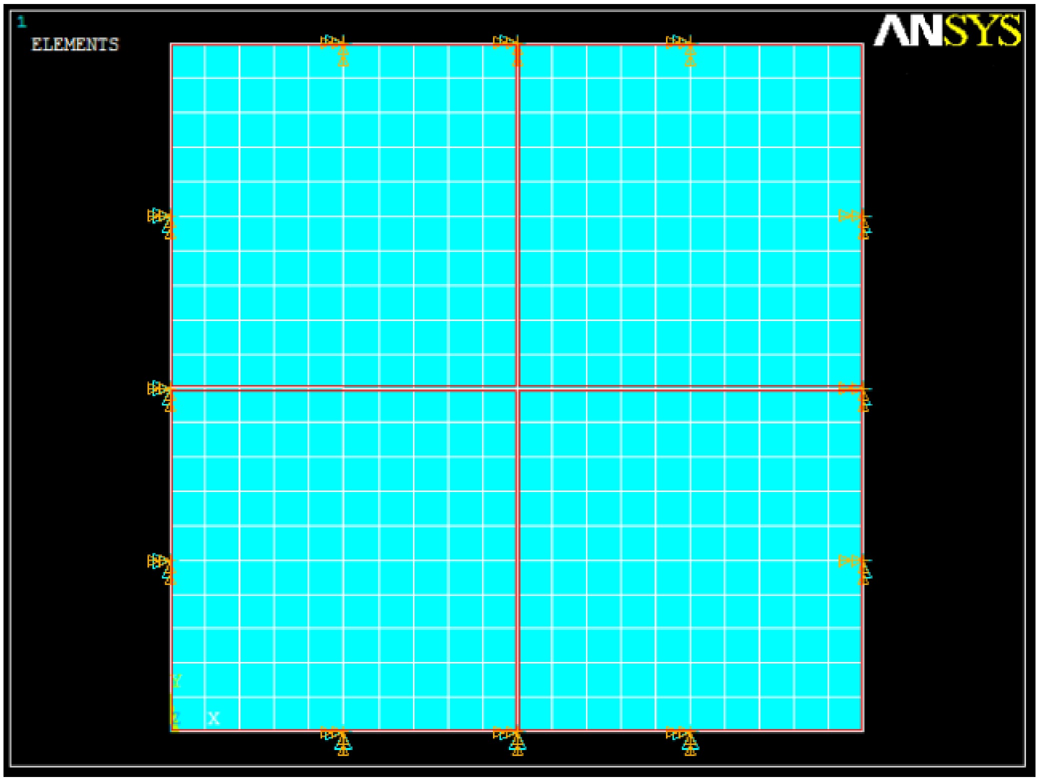

Figure 14. Boundary Condition: C-C-C-C

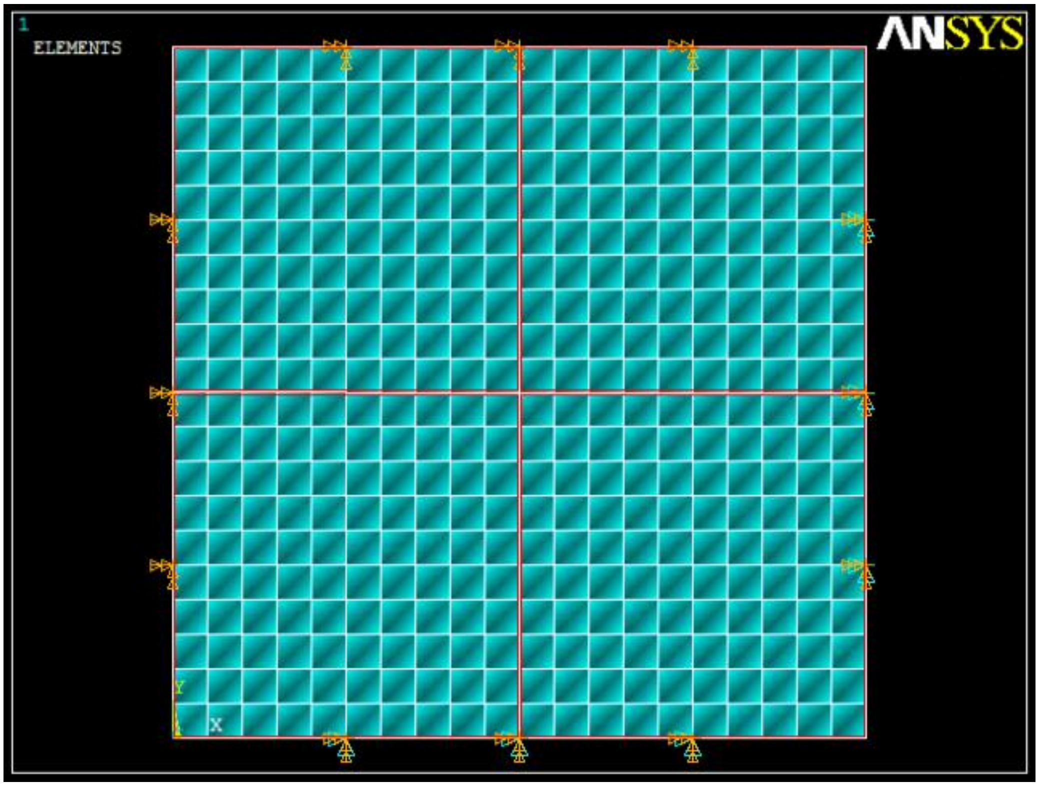

Figure 15. Boundary Condition: C-C-S-S

%Geometrical and material properties of plate

```
% SI units are used
a = 1;                  % Length of the plate (along X-axes)
b = 1;                  % Length of the plate (along Y-axes)
E = 70e9;               % elastic modulus ( Aluminum Plate)
nu = 0.33;              % Poisson's ratio
t = 0.001;              % plate thickness
I = t^3/12;
```

$D = \frac{Eh^3}{12\,(1-\vartheta^2)}$ Elasticity matrix

3.2 Results

Table 2: Deflection of plate: All sides simply supported subjected to uniform distributed load (S-S-S-S)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$3.15*10^{-2}$	$3.09*10^{-2}$	$3.13*10^{-2}$
2	0.002	$4.98*10^{-3}$	$4.87*10^{-3}$	$4.89*10^{-3}$
3	0.003	$1.74*10^{-3}$	$1.65*10^{-3}$	$1.688*10^{-3}$
4	0.004	$7.89*10^{-4}$	$7.8*10^{-4}$	$7.81*10^{-4}$
5	0.005	$2.29*10^{-4}$	$2.25*10^{-4}$	$2.284*10^{-4}$
6	0.006	$1.48*10^{-4}$	$1.43*10^{-4}$	$1.451*10^{-4}$

Table 3: Deflection of plate: Opposite sides clamped and simply supported subjected to uniform distributed load(C-S-C-S)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$1.49*10^{-2}$	$1.44*10^{-2}$	$1.45*10^{-2}$
2	0.002	$1.97*10^{-3}$	$1.91*10^{-3}$	$1.95*10^{-3}$
3	0.003	$7.34*10^{-4}$	$7.3*10^{-4}$	$7.333*10^{-4}$
4	0.004	$2.376*10^{-4}$	$2.3*10^{-4}$	$2.33*10^{-4}$
5	0.005	$1.22*10^{-4}$	$1.16*10^{-4}$	$1.19*10^{-4}$
6	0.006	$6.79*10^{-5}$	$6.74*10^{-5}$	$6.773*10^{-5}$

Table 4: Deflection of plate: Adjacent sides clamped and simply supported subjected to uniform distributed load(C-C-S-S)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$1.72*10^{-2}$	$1.65*10^{-2}$	$1.67*10^{-2}$
2	0.002	$3.59*10^{-3}$	$3.5*10^{-3}$	$3.567*10^{-3}$
3	0.003	$6.595*10^{-4}$	$6.49*10^{-4}$	$6.55*10^{-4}$
4	0.004	$2.36*10^{-4}$	$2.25*10^{-4}$	$2.30*10^{-4}$
5	0.005	$1.07*10^{-4}$	$9.95*10^{-5}$	$9.98*10^{-5}$
6	0.006	$7.68*10^{-5}$	$7.59*10^{-5}$	$7.634*10^{-5}$

Table 5: Deflection of plate: All sides clamped or fixed subjected to uniform distributed load(C-C-C-C)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$9.66*10^{-3}$	$9.59*10^{-3}$	$9.62*10^{-3}$
2	0.002	$1.27*10^{-3}$	$1.2*10^{-3}$	$1.23*10^{-3}$
3	0.003	$2.48*10^{-4}$	$2.35*10^{-4}$	$2.37*10^{-4}$
4	0.004	$9.14*10^{-5}$	$9.04*10^{-5}$	$9.09*10^{-5}$
5	0.005	$7.83*10^{-5}$	$7.7*10^{-5}$	$7.806*10^{-5}$
6	0.006	$4.58*10^{-5}$	$4.45*10^{-5}$	$4.489*10^{-5}$

Table 6: Deflection of plate: All sides simply supported subjected to Concentrated load (S-S-S-S)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$9.01*10^{-2}$	$8.83*10^{-2}$	$8.91*10^{-2}$
2	0.002	$1.97*10^{-2}$	$1.9*10^{-2}$	$1.95*10^{-2}$
3	0.003	$7.35*10^{-3}$	$7.3*10^{-3}$	$7.345*10^{-3}$
4	0.004	$2.79*10^{-3}$	$2.7*10^{-3}$	$2.72*10^{-3}$
5	0.005	$7.20*10^{-4}$	$7.06*10^{-4}$	$7.11*10^{-4}$
6	0.006	$3.89*10^{-4}$	$3.77*10^{-4}$	$3.79*10^{-4}$

Table 7: Deflection of plate: Opposite sides clamped and simply supported subjected to Concentrated load(C-S-C-S)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$5.31*10^{-2}$	$5.2*10^{-2}$	$5.25*10^{-2}$
2	0.002	$7.67*10^{-3}$	$7.6*10^{-3}$	$7.63*10^{-3}$
3	0.003	$1.71*10^{-3}$	$1.65*10^{-3}$	$1.68*10^{-3}$
4	0.004	$8.68*10^{-4}$	$8.62*10^{-4}$	$8.656*10^{-4}$
5	0.005	$5.211*10^{-4}$	$5.21*10^{-4}$	$5.18*10^{-4}$
6	0.006	$2.57*10^{-4}$	$2.44*10^{-4}$	$2.50*10^{-4}$

Table 8: Deflection of plate: Adjacent sides clamped and simply supported subjected to Concentrated load(C-C-S-S)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$5.86*10^{-2}$	$5.8*10^{-2}$	$5.9*10^{-2}$
2	0.002	$9.17*10^{-3}$	$9.01*10^{-3}$	$9.07*10^{-3}$
3	0.003	$2.55*10^{-3}$	$2.51*10^{-3}$	$2.59*10^{-3}$
4	0.004	$8.788*10^{-4}$	$8.71*10^{-4}$	$8.74*10^{-4}$
5	0.005	$5.52*10^{-4}$	$5.47*10^{-4}$	$5.471*10^{-4}$
6	0.006	$2.7*10^{-4}$	$2.59*10^{-4}$	$2.67*10^{-4}$

Table 9: Deflection of plate: All sides clamped or fixed subjected to Concentrated load (C-C-C-C)

S No.	Thickness of Plate (m)	Exact solution Deflection (m)	Deflection by Matlab (m)	From ANSYS Deflection (m)
1	0.001	$4.25*10^{-2}$	$4.18*10^{-2}$	$4.213*10^{-2}$
2	0.002	$3.24*10^{-3}$	$3.2*10^{-3}$	$3.27*10^{-3}$
3	0.003	$1.12*10^{-3}$	$1.05*10^{-3}$	$1.09*10^{-3}$
4	0.004	$4.65*10^{-4}$	$4.54*10^{-4}$	$4.578*10^{-4}$
5	0.005	$1.14*10^{-4}$	$1.05*10^{-4}$	$1.07*10^{-4}$
6	0.006	$9.99*10^{-5}$	$9.96*10^{-5}$	$9.91*10^{-5}$

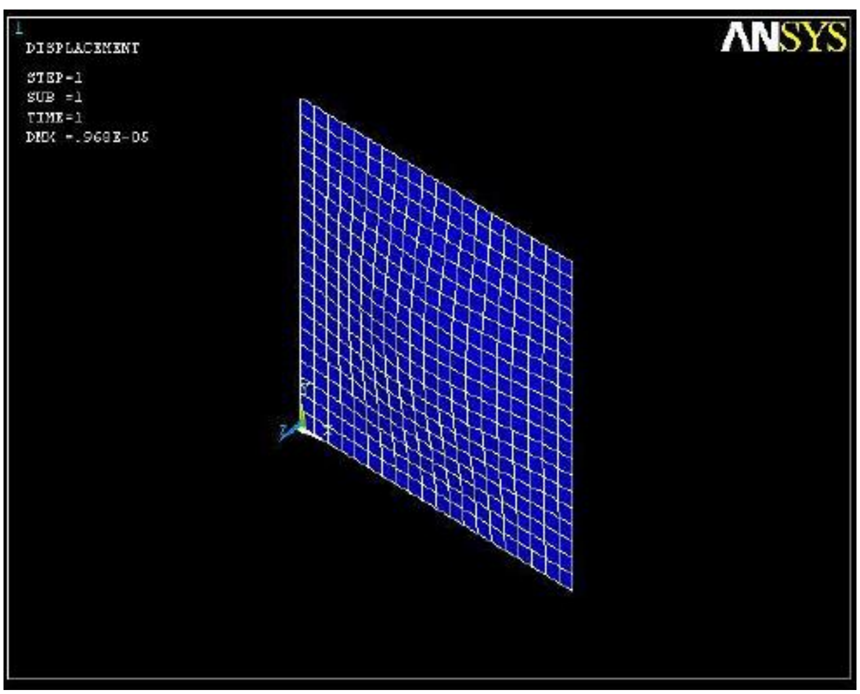

Figure 16. Deflection of plate: All sides clamped or fixed supported subjected to UDL pressure.

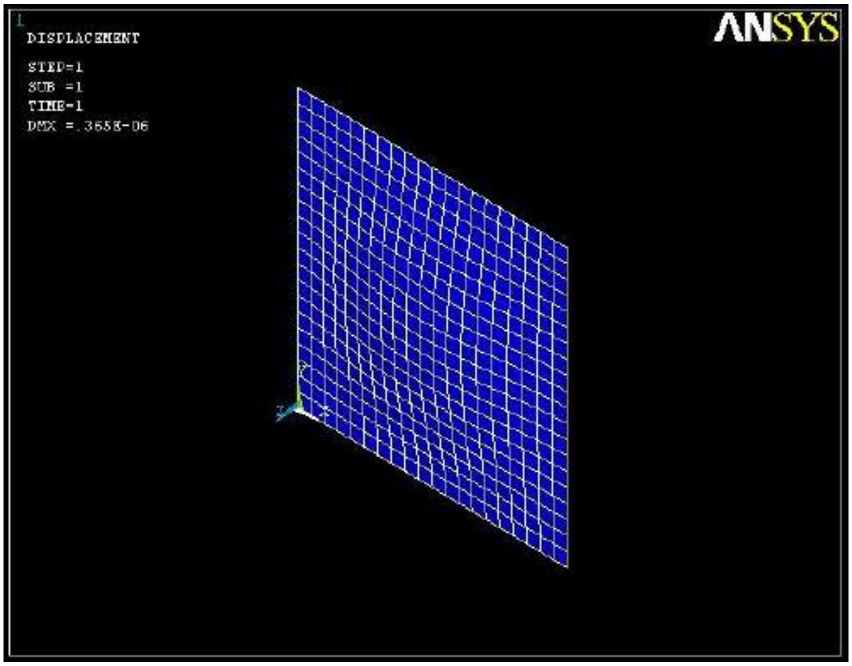

Figure 17. Deflection of plate: All sides Simply supported under UDL pressure

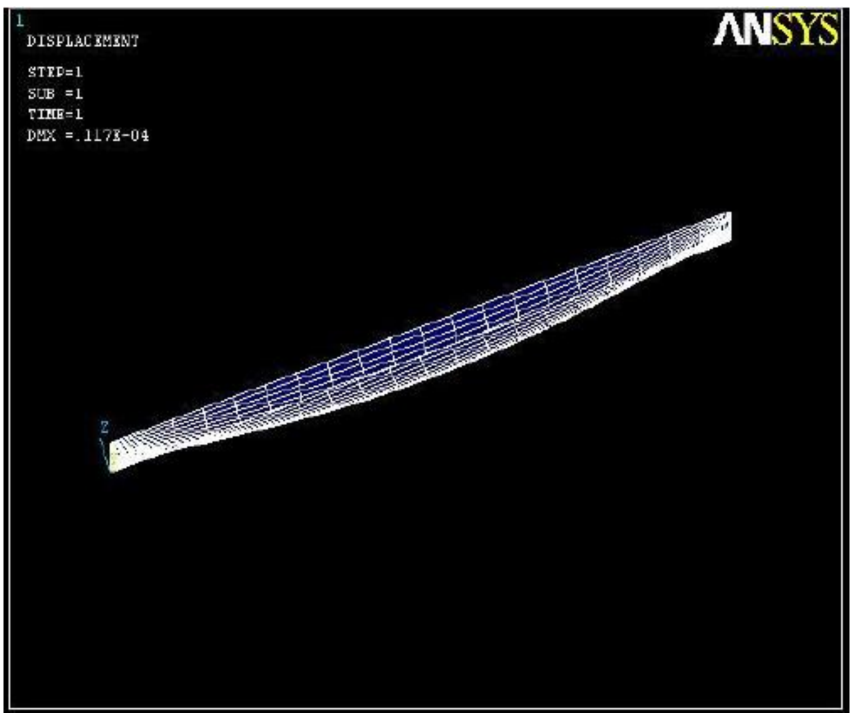

Figure 18. Deflection of plate: Opposite sides clamped and simply supported subjected to Concentrated load

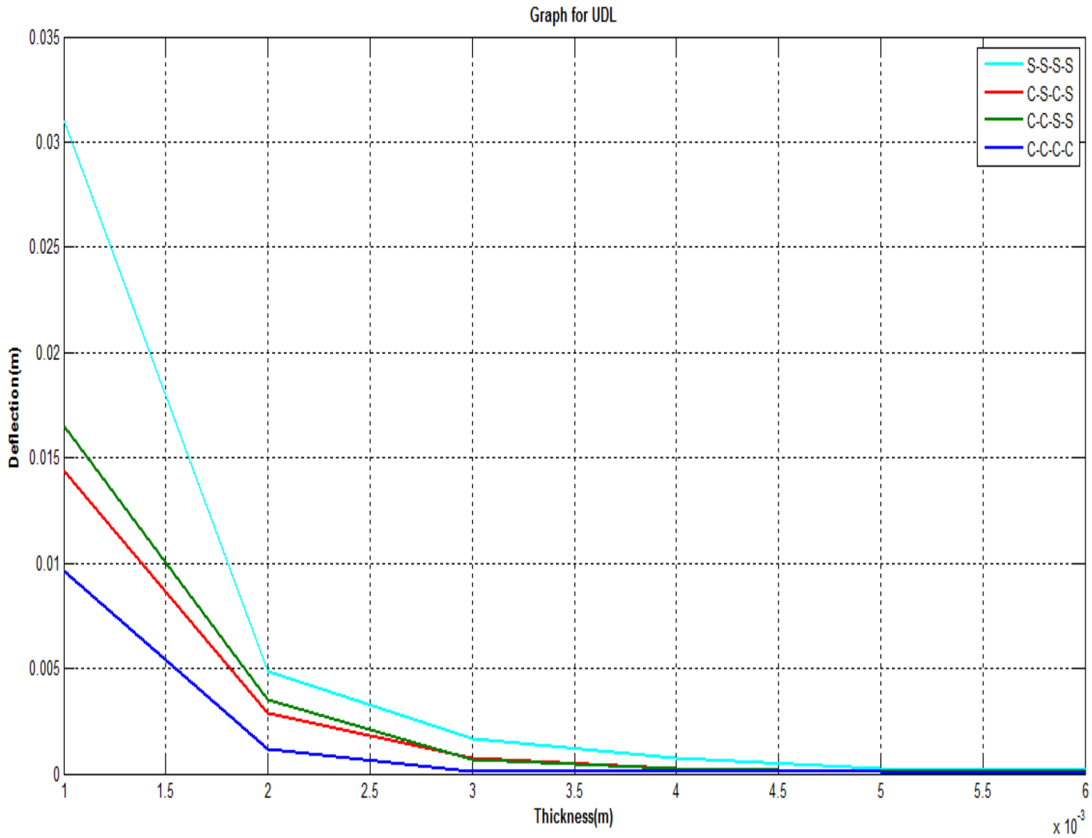

Figure 19. Deflection VS thickness graph for UDL

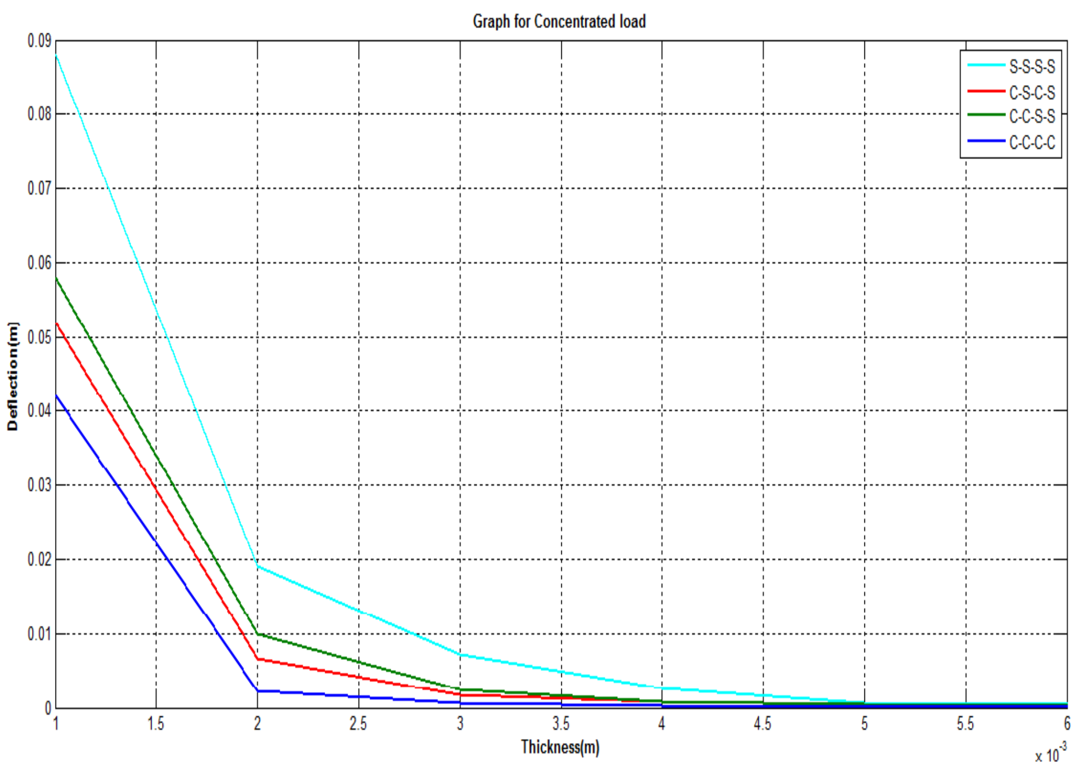

Figure 20. Deflection VS thickness graph for Concentrated Load

Table 10: Result of optimum thickness of plate

Load Condition	Boundry Condition	Optimum Thickness (m)
UDL	S-S-S-S	.0045
	C-C-S-S	.0035
	C-S-C-S	.0035
	C-C-C-C	.003
Concentrated Load	S-S-S-S	.005
	C-C-S-S	.004
	C-S-C-S	.004
	C-C-C-C	.0035

Chapter 4

Conclusions

This project mainly focused on the finite element model for finding field variables of an isotropic rectangular plate. The analysis has been performed by considering a four noded rectangular element as a basic geometric shape. During the analysis, plate thick varies from 0.001 to 0.006 m, under different load conditions (UDL of 500 N/m2 and concentrated point load of 50 N) and different boundary conditions (simply supported, clamped, opposite clamped n simply supported and adjacent clamped n simply supported). Later, for the same structure and load/boundary conditions, analysis has been performed using analysis software ANSYS. Finally, the results obtained from FEA and ANSYS have been compared with exact solutions, which are calculated from Kirchhoff plate theory. From the numerical results, optimum thickness (thickness at which plate gives best performance) for the same plate under different boundary and load conditioned has been calculated. Analysis results showed that the results obtained from FEA and ANSYS are closely converging to the results obtained from exact solutions.

It has been observed that deflections of plate decreases due to end restraining and gives the most optimum thickness under both Concentrated and Uniformly Distributed Load.

References

[1] **Q.S. Li, Jie Liu, H.B. Xiao**. "A new approach for bending analysis of thin circular plates with large deflection" *International Journal of Mechanical Sciences,* Volume 46, Issue 2, February 2004, Pages 173-180.

[2] **I. Shufrin, O. Rabinovitch, M. Eisenberger.** "A semi-analytical approach for the non-linear large deflection analysis of laminated rectangular plates under general out-of-plane loading" *International Journal of Non-Linear Mechanics,* Volume 43, Issue 4, May 2008, Pages 328-340.

[3] **Hui-Shen Shen**. "Large deflection of composite laminated plates under transverse and in-plane loads and resting on elastic foundations" *Composite Structures,* Volume 45, Issue 2, June 1999, Pages 115-123.

[4] **Bakker MCM, Rosmanit M, Hofmeyer H** (2008). "Approximate large deflection analysis of simply supported rectangular plates under transverse loading using plate post-buckling solutions", *Struct. Design*, 46(11): 1224-1235.

[5] **Bhattacharya MC** (1986). "Static and Dynamic Deflections of Plates of Arbitrary Geometry by a New Finite Difference Approach*". J. Sound. Vib*., 107(3): 507-521.

[6] **Chaudhuri RA** (1987). "Stress concentration around a part through hole weakening laminated plate*". Comput. Struct*., 27(5): 601-609.

[7] **Das D, Prasanth S, Saha K** (2009). "a variational analysis for large deflection of plates under uniformly distributed load through domain mapping technique". *Int. J. Eng. Sci. Technol*., 1(1): 16-/32.

[8] **José MarÍa Mínguez, Jeffrey Vogwell. "**Plates with holes under lateral load pressure" *Original Research Article Engineering Failure Analysis*, Volume 5, Issue 4, 1 December 1998, Pages 299-315.

[9] **Defu W, sheikh E** (2005). "Large deflection mathematical analysis of rectangular plates", *Journal of engineering mechanics*, 131(8): 809-821.

[10] **Jain NK** (2009). "Analysis of Stress Concentration and Deflection in Isotropic and Orthotropic Rectangular Plates with Central Circular Hole under Transverse Static

Loading”, *52th international meet World Academy of Science, Engineering and Technology, Bangalore.*

[11] **M.N.Y. Anwar, M.Y.Z. Koujouk .** “Deflection of concentrically supported and eccentrically loaded thin circular plate”. *Journal of Computational and Applied Mathematics*, Volume 36, Issue 2, 27 August 1991, Pages 189-207.

[12**]** **Liew KM, Teo TM, Han JB** (1999). “Three-dimensional static solutions of rectangular plates by variant differential quadrature method”. *Int. J. Mech. Sci*., 43: 1611-1628.

[13] **Paul TK, Rao KM** (1989). “Finite element evaluation of stress concentration factor of thick laminated plates under transverse loading*”. Comput. Struct*., 48(2): 311-317.

[14] **Pape D, Fox AJ** (2006). “Deflection Solutions for Edge Stiffened Plates”, *Proceedings of the 2006 IJME - INTERTECH Conference, Session: Eng*, pp. 203-091.

[15] **Troipsky MS** (1976)."Stiffened Plates, Bending, Stability, and Vibration", *Journal of applied mechanics*, 44(3),p. 516-525.

[16] **Shaiov MA, Vorus WS** (1986). “Elasto-plastic plate bending analysis by a boundary element method with initial plastic moments”. *Int. J. Solid. Struct*., 22(2): 1213-1229.