DEVELOPMENT OF INTEGRATED FORCE METHOD FOR THE ANALYSIS OF FRAMED AND CONTINUUM STRUCTURES

By

Ganpat Shankerrao Doiphode

Research Guide

Dr. S. C. Patodi

Former Professor of Structural Engineering

सत्यं शिवं सुन्दरम्

Applied Mechanics Department

Faculty of Technology and Engineering

The Maharaja Sayajirao University of Baroda

Vadodara - 390001, Gujarat

SEPTEMBER 2012

DEVELOPMENT OF INTEGRATED FORCE METHOD FOR THE ANALYSIS OF FRAMED AND CONTINUUM STRUCTURES

A Thesis Submitted to
The Maharaja Sayajirao University of Baroda

For the Degree of

Doctor of Philosophy

in

Civil Engineering

By

Ganpat Shankerrao Doiphode

Research Guide
Dr. S. C. Patodi
Former Professor of Structural Engineering

सत्यं शिवं सुन्दरम्

Applied Mechanics Department
Faculty of Technology and Engineering
The Maharaja Sayajirao University of Baroda
Vadodara - 390001, Gujarat

September 2012

ABSTRACT

It is unthinkable for a structural engineer today to consider the analysis of any structure without calling on the aid of the computer to carry out the laborious and lengthy calculations that are often involved in the analysis. Basically there are two different types of method, named based on the primary unknowns, which finally determine the complete solution. In the force or flexibility approach, the redundant forces are chosen as the unknowns whereas in the displacement or stiffness method the displacements of the joints of the structure are considered as unknowns. Out of these two methods, Stiffness method is easier to perform and therefore it has become more popular. Both flexibility and stiffness approaches, however, are indirect approaches because the internal forces are not considered as primary unknowns directly in either method, but they are calculated from redundants in flexibility method and from nodal displacements in stiffness method.

Recently a new formulation, which is based on the Direct Force Determination, is referred in the literature as Integrated Force Method (IFM). It couples equilibrium equations and compatibility conditions in a single matrix and thus gives the internal forces without an intermediate step of finding the displacements or redundants. It is independent of redundant selection and hence it is conducive to programming. Also, this method is not only applicable to framed structures but also to continuum structures like the well known Finite Element Method.

To explore further the IFM, which has been mainly developed by Patnaik and his team at Ohio Aerospace Institute, the method is checked in the present work for its versatility by solving a variety of skeletal (1D) and surface (2D) structure problems. Formulation is developed for a number of 1D elements to handle pinned and rigid jointed plane and space structures whereas to deal with plane stress, plane strain, and plate bending problems a number of 2D elements are developed by representing both stress and displacement variations with suitable functions

simultaneously. For example, rectangular element designated as RECT_5F_8D has five forces and eight displacements as unknowns and their variation inside the element is expressed with separate functions.

A modified form of IFM, named as Dual Integrated Force Method (DIFM), is also developed where displacements are considered as primary unknowns and then using the same internal forces are calculated.

IFM based formulation is developed to analyse both 1D and 2D structures under static and dynamic loading. A rectangular element having 9 forces and 12 displacements (RECT_9F_12D) as unknowns is proposed for the plate bending problems of both isotropic and orthotropic material. A variety of problems are attempted and where possible results are compared with the available classical and/or numerical solutions. To deal with the axisymmetric circular and annular plate bending problems, an element named as CIRC_2F_4D element is also formulated and validated by solving a variety of problems under different loading and support conditions.

Integrated Force based methodology is also extended to deal with 1D and 2D problems of stability analysis. Geometric stiffness matrix is developed to evaluate critical buckling load of beam, truss, frame and plate problems. Results are found in good agreement with the available solutions.

For carrying out the above work, Pre-, Main-and Post-processors are developed in Visual Basic 6 and Visual Basic.NET. Matlab software is also extensively used for numerical and graphical processing. Moment contours and deflection profiles are also plotted to facilitate easy interpretation of results and to make the whole exercise of finding the solution as attractive as possible.

THE MAHARAJA SAYAJIRAO UNIVERSITY OF BARODA

CERTIFICATE

This is to certify that the thesis entitled **DEVELOPMENT OF INTEGRATED FORCE METHOD FOR THE ANALYSIS OF FRAMED AND CONTINUUM STRUCTURES** *submitted by* **Shri Ganpat Shankerrao Doiphode** *represents his original work which was carried out by him at Applied Mechanics Department, Faculty of Technology and Engineering, The M. S. University of Baroda, Vadodara under my guidance and supervision for the award of the Degree of Doctor of Philosophy in Civil Engineering.*

The matter presented in this thesis has not been submitted anywhere else for the award of any other degree.

Dr. S. C. Patodi
Research Guide
Former Professor of Structural Engg.

Vadodara
September 2012

Dr. I. I. Pandya
Head
Applied Mechanics Department

Prof. A. N. Misra
Dean
Faculty of Technology and Engineering
The M. S. University of Baroda. Vadodara – 390001

ACKNOWLEDGEMENT

I would like to thank God Almighty without whose blessings, no work can get completed. Next, I would like to thank my parents. who have always allowed me to pursue the career of my liking.

I take this opportunity to express my sincere heartfelt gratitude to my guide **Dr. S. C. Patodi**, Former Professor of Structural Engineering, Department of Applied Mechanics, Faculty of Technology and Engineering, MSU, Baroda for his knowledgeable guidance, invaluable suggestions and affectionate care with which he has directed the work and for all the pains he has taken extending his timely cooperation and inspiring encouragement whenever circumstances compelled to give up hope and yield to despair. Words are just insufficient to express my feelings at this stage.

I would like to thank Dr. I. I. Pandya, Head of Applied Mechanics Department, Faculty of Tech & Engg, M. S. University of Baroda, for providing the computational facilities.

Special thanks are also due to all my colleagues in the department who have constantly kept up my morale and helped in boosting my pace of working towards the end of my research work.

I would like to thank my family members who have constantly kept up my spirit while working on the current thesis. Finally, my sincere thanks to all those who have helped me in achieving different milestones at different stages of my life, the current work being one of them.

G. S. Doiphode

September 2012.

NOTATIONS AND ABREVIATIONS

A	Cross sectional area
AR	Aspect ratio
a	Size of element in x - direction
[A]	Co-ordinate matrix
Bccs	Boundary compatibility conditions
b	Size of element in y - direction
$b_{\overline{12}}$, $b_{\overline{23}}, b_{13}$	Coefficients corresponding to equilibrium matrix
[B]	Equilibrium matrix
$[B_e]$	Basic element equilibrium matrix
$[B_F]$	Matrix based coordinates of forward end
CCmatrix	Product of [C] and [G] matrices in Matlab
CCs	Compatibility conditions
$c_{\overline{12}}$, $c_{\overline{23}}$, c_{13}	Coefficients corresponding to equilibrium matrix
[C]	Compatibility matrix
DDRs	Displacement deformation relations
DR	Displacement ratio
ddof	Displacement degrees of freedom
[D]	Material matrix
$[D]_{DIFM}$	Dual matrix
$[D_{ps}]$	Material matrix for plane strain material
E	Modulus of elasticity
EEs	Equilibrium equations
E_x , E_y	Modulus of elasticity along x -and -y directions
FDRs	Force deformation relations
fdof	Force degrees of freedom
{F}	Internal force vector
GJ	Torsional rigidity of member
G_{matrix}	[G] matrix in Matlab

G_{xy}	Shear Modulus in xy plane
[G]	Associated flexibility matrix
$[G_e]$	Elemental flexibility matrix
$[G^e_{ps}]$	Elemental flexibility matrix for plane strain
I	Moment of inertia
[J]	Coefficient matrix equals to $[S^{-1}]^T$
$[K_g]$	Geometric stiffness matrix
$[K_{ge}]$	Geometric element stiffness matrix
L	Length of member
L_1, L_2, L_3	Shape function in area coordinate system
[L]	Operator matrix
MJGmatrix	Product of [M], [J] and [G] matrices
M_{DNLM}	Direct Nodal Lumping Mass
M_{xx}, M_{yy}	Moments along x - and y- directions
M_{xy}	Torsional moment in x-y plane
m	Number of displacement degrees of freedom
m_o	Lumped mass
$[M_C]$	Consistent mass matrix
$[M_L]$	Lumped Mass Matrix or Mmatrix in Matlab
N_x, N_y	In-plane forces along x- and y- directions
n	Number of force degrees of freedom
[N]	Shape function matrix
P_{cr}	Critical buckling load
Pmatrix	{P} vector in Matlab
{P}	Load vector
$q_o(r, \theta)$	Transverse loading on circular plate
r	Radius in polar coordinate system
$\bar{r}$, $\bar{x}$, $\bar{y}$	Coefficients corresponding to respective directions
Sinv	$[S]^{-1}$
[S]	Global equilibrium matrix
$[S_b]$	Stability matrix
[Smatrix]	[S] matrix in Matlab

T	Torque in the member
t	Thickness of element
U	Internal strain energy
u, v	Nodal displacements along x- and y- directions
[Y]	Stress interpolation function matrix
z.cMatrix	[C] matrix
z.cTransposeB	Product of [C] and $[B]^T$
[Z]	Strain linking matrix
$\bar{\alpha}$	Coefficient corresponding to angle α
{β}	Deformation vector
δ_L	Extension in the member
{δ}	Displacement Vector
ε	Strain
θ_x, θ_y	Rotations along x – and- y directions
[λ]	Transformation matrix
ϕ_{xy}	Polynomial function
σ_r	Radial stress in curved member
σ_x , σ_y	Stresses in x – and- y directions
σ_θ	Tangential stress in curved member
τ_{xy}	Shear stress in xy plane
γ	Poisson's ratio
ω_{11}	Frequency of first mode
ω_{IFM}	IFM based frequency
ω_{exact}	Exact Value of frequency

CONTENTS

CHAPTER 1

INTRODUCTION

1.1 MODERN METHODS OF STRUCTURAL ANALYSIS

Structural analysis is concerned with the determination of the forces (stresses) and displacements (strains) experienced by the structure under the loading. The basic requirements to be satisfied by any method of analysis are the equilibrium between the internal stresses and the external loads, the compatibility of displacements at the joints and the satisfaction of the prescribed force or deformation conditions at the supports.

Till 1930s the only means of solving indeterminate structures was to formulate simultaneous equations using slope deflection equation or its variation. Without the relaxation method or computers which were not known till then, the engineer had a formidable task solving large number of these equations and indeterminate structures. The moment distribution method of Hardy Cross in 1930 provided a spurt and the industry saw a great leap. Still many structures were beyond this method. Rigorous methods for solution of multistoried buildings with sway, for example, though within reach involved a formidable task. Air craft structures posed an even more difficult task.

It was only after World War II that computers came into the field. It demanded a fresh look at the more "Exact" methods of computation in favour of iteration method such as moment distribution method. With the development of the digital computer, matrix notations and algebra came to be used in the organization of calculations. By formulating the tools of matrix structural analysis in a mathematically consistent fashion, the analyst achieves a systematic approach that is convenient for automatic computation.

Matrix methods afford the structural analysis of a class of problems to be represented as a series of algebraic operations on matrices representing force and displacement groups, and thus permits programming in terms of these symbols that are to be substituted later by numerical values. Thus by changing numerical data, solutions of structures of different geometry and loading can readily be obtained. Therefore one can develop general program for a given class of structures such as pin jointed frames, plane frames, etc, without any specific geometry of the structure or loading in view. From the practical view point of design office, the power of such computations is immense as general purpose program once written out and tested together with a powerful computational services, it can be used to solve a variety of problems.

Basically there are two different types of matrix methods to analyse structures, namely force and displacement methods. These are frequently called compatibility method and equilibrium method respectively. Also the alternative names, flexibility method and stiffness method are equally popular. The names force and displacement methods are based on the features that certain force or displacement variables finally determine the solution. When forces are used as variable, equilibrium is satisfied first and the requirement of compatibility are used as conditions to determine the unknown force variables. When displacements are chosen as variables, the compatibility requirements are satisfied first and the requirements of equilibrium form the necessary equations to determine the displacement variables.

Any approach of analysis, be it a force or displacement approach, breaks up the total effort in to a number of steps. The analysis proceeds from the part to whole. The analysis procedure essentially consists of two steps: (i) Member analysis, and (ii) Structural analysis. The first step is the study of the member itself as an independent body. Properties such as stiffness or flexibility, which are required in the second stage are determined. The second step then consists of the analysis of the assembly of such members. In the first step certain displacement or force parameters are taken as unknowns and the member analysis is formulated in terms of these unknowns. The second step determines the set of all such unknowns. This step involves the knowledge of structural connectivity and from the computational viewpoint is a major issue.

As mentioned above both flexibility and stiffness methods lead to a set of simultaneous equations to be solved. The number of unknowns in the flexibility method is generally smaller. However there is some arbitrariness in the force approach in the sense that the analyst has certain freedom to choose the set of force unknowns. The displacement approach, however, is more definite in its steps (without any alternatives) and thus becomes easier from the point of view of developing general purpose programs. Some analysts on the other hand, consider this a disadvantage and prefer the flexibility method since it offers freedom in the selection of the redundants.

In practice one is required to analyse not only skeletal structures but also continuum structures such as plate and shell structures. In continuum structures the material is continuously distributed over the whole structure and not concentrated along discrete lines constituting individual members. In other words in a continuum structure, which may be a surface structure (two dimensional structure) or solid structure (three dimensional structure), there are no physically recognizable members or joints. If the stiffness method is to be adopted for the analysis of such structures it is obviously necessary to imagine that the structure is assembled form discrete elements called finite elements connected at discrete joints called as nodes. It is of course necessary to ensure that the structure assembled from the finite elements satisfies the basic equilibrium and compatibility requirements.

An important feature of the finite element method (FEM), which sets it apart from other approximate numerical methods, is its ability to formulate solutions for each element before putting them together to represent the entire problem. In essence, a complex problem reduces to considering a series of greatly simplified problems. Another advantage of the FEM is the variety of ways in which one can formulate the properties of individual elements. Out of the various approaches available to formulate the element properties, direct approach and variational approach are generally used for structural engineering problems. While direct approach can be used to derive properties of simple (1D) elements i.e. members of framed structures, the variational approach can be used to derive properties of both simple (1D) and sophisticated (2D and 3D) element shapes.

The variational approach relies on the calculus of variations and involves minimization of a function. For problems of solid mechanics, the functional turns out to be the potential energy, the complimentary energy or some derivatives of these, such as the Reissner variational principle. Following are the variational approaches available for deriving the element properties.

The displacement based FEM employs the principle of minimum potential energy analogous to the principle of virtual displacement. A displacement function is assumed for each element. Compatibility of displacement within the element and across the interelement boundaries has to be satisfied.

In the force based FE approach, principle of minimum complimentary energy analogous to the principle of virtual forces is employed. An equilibrium field is assumed within the element in such a way so as to satisfy internal equilibrium and continuity of stresses between the elements.

In the mixed method, which is based on the Hellinger-Reissner variational principle, both displacements and stress fields are assumed separately for each element and these have to be continuous within the element and between the elements.

In the hybrid method, stress distributions are assumed within the element and displacement distributions are assumed along the boundaries of the element. Both internal equilibrium and compatibility of displacements along inter element boundaries are to be satisfied.

Among the various approaches discussed above, the displacement formulation is the most popular and widely used for structural problems because of its simplicity and easy applicability to a variety of problems.

1.2 FLEXIBILITY IN FORCE BASED METHOD

The fundamental equation in the matrix form for the flexibility method is $\{D\} = [F]\{A\}$, which states that the displacements can be expressed in terms of actions by the formulation of flexibility matrix [F] that represents displacements due to the unit values of the actions.

For Statically indeterminate structures, the displacements can be obtained by super-imposing those due to loads and those due to redundants for the displacements at the release where the redundants acts, the following equation can be written

$$\{D_Q\} = \{D_{QL}\} + [F]\{Q\} \qquad \dots (1.1)$$

where the column matrix $\{D_Q\}$ represents the actual displacements at the releases that are either prescribed or known, $\{D_{QL}\}$ represents the displacements corresponding to redundants in the released structure due to the actual loads, and [F] is a square flexibility matrix representing the displacements at the releases in the released structure due to unit values of the redundants $\{Q\}$.

If a computer is available for use, one can combine system analysis and member analysis by analyzing the system through a member approach. Matrix formulation allows to develop the necessary flexibility matrices by first considering the flexibilities of the individual members and then combining these to analyse the system - including members- as a whole. In the formalized version of the flexibility method use is made of transfer matrices in finding $\{D_{QL}\}$ and [F] and compatibility equation is expressed as

$$\{D_Q\} = [A^T_{MQ}][F_M][A_{ML}] + [A^T_{MQ}][F_M][A_{MQ}]\{Q\} \quad \ldots (1.2)$$

Where $\{A_{ML}\}$ and $\{A_{MQ}\}$ are known as transfer matrices and these represent respectively member end action due to actual loading and unit redundants, and $[F_M]$ is known as member flexibility matrix.

There is considerable freedom in the manner one can generate compatibility equations in the formalized approach of flexibility method. Six different approaches which differ mainly in the formulation of $\{D_{QL}\}$ and [F] are discussed below.

1.2.1 Conventional Approach

The approach described by Gere and Weaver **[1]** is called here as conventional approach. In this approach the compatibility equation is used exactly in the manner as given in Eq. (1.2). After selecting the suitable released structure the displacements are obtained with the help of given equation as

$$\{D_{QL}\} = [A^T_{MQ}][F_M][A_{ML}] \quad \text{and,} \quad [F] = [A^T_{MQ}][F_M][A_{MQ}] \quad \ldots (1.3)$$

The determination of transfer matrices $[A_{MQ}]$ and $[A_{ML}]$ requires that the released structure be analysed by statics for unit values of redundants and also for the combined joint loads. For each member the size of $[A_{MQ}]$, $[A_{ML}]$ and $[F_M]$ depends upon the internal actions considered for the member.

These matrices are formed for the whole structure from the submatrices containing the contribution of individual members. The choice of the redundants greatly influence the amount of computational work required for the generation of the transfer matrices and because of the multiple choices in the selection of redundants one can not generate these matrices automatically through the computer. i.e. one has to supply these matrices as input to the computer. Moreover, most of the elements of the flexibility matrix are nonzero in this approach unless the released structure is chosen to ensure localized phenomena.

1.2.2 Connection Matrix Approach

The approach which requires the use of connection matrix in the generation of $\{D_{QL}\}$ and [F] matrices is called here as connection matrix approach **[2]**. The connection matrix is generated by writing the force equilibrium equation at each joint between the applied external forces and the member forces and discarding all rows corresponding to known displacements and thus introducing the boundary conditions. This matrix relates member forces to given joint forces. Therefore it is known as connection matrix [C]. The numbers of excess columns than the numbers in this connection matrix automatically represents the indeterminacy of structure. For the generation of [C] matrix, a translation matrix is required. For the generation of $[A_{MQ}]$ and $[A_{ML}]$ matrces, proper partition is made for these in the connection matrix. Therefore, there is no need to divide the method into two steps i.e. one for the determinate structure and the other for the indeterminate structure. After the selection of redundants, the rearrangement of columns in [C] matrix and $[C_I]$ and $[C_{II}]$ is made. From which $[A_I]$ and $[A_{II}]$ are derived. From $[A_I]$ and $[A_{II}]$ by simply adding zero and identity matrix, one can obtain the $[A_{MQ}]$ and $[A_{ML}]$ matrices.

The proper selection of redundants is important as it effects the stability and accuracy requirements. The redundants are selected such that this matrix becomes sparse, banded and well conditioned. The sparseness can be achieved, if attention is paid to the fact that a force in the direction of one redundant should not introduce displacements in the direction of all other redundants or its effect must be as localized as possible. The banded property which economizes memory space could be achieved by careful selection and numbering of the redundants. A well conditioned matrix can be obtained if redundants cause larger displacements in their own direction than in the direction of the other redundants. In connection matrix method displacements and member end actions are represented in global co-ordinate system. The method of connection matrix is advantageous only when the inverted form of the [C], or alternatively [A_{MQ}] and [A_{ML}] can be set up directly but this is possible only in very few cases.

1.2.3 Subframe Approach

In the subframe approach, given by Shaw **[3]**, for the generation of [A_{MQ}] matrix i.e. Self equilibrating force system matrix the structure is broken up into small subframes. Every subframe is cut at suitable places and redundants are applied. Bending moment diagrams are drawn for all redundants and these diagrams are used to write [A_{MQ}] matrix. This method has advantage that no inversion of connection matrix is required. Small subframe prevents spreading of redundant forces on more members and hence in this approach the effects are localized. For generating [A_{ML}] matrix, the bending moment diagram is drawn for applied loads and then [A_{ML}] matrix which is also known as static equilibrium matrix is formed by noting member end forces from the BMD. Although the technique of subframe is applicable to all structures weather fixed or hinged footed but it requires too many diagrams to be drawn and hence much work is to be done manually before making use of the computer for the analysis.

1.2.4 Rank Force Approach

The most significant difference in the stiffness and flexibility method is that one has to prepare more input data when adopting the flexibility approach.

The additional input consists of manually generating the basic redundant load systems which initially one has to decide whether or not the structure is redundant and, if so, to what degree. A proper set of redundancies must then be selected. For large size problems, these requirements are very time consuming, laborious and prone to error. Argyris in 1962 **[4]** suggested a method of reducing this work by considering redundant subsystems but this method is applicable only for specific structural configuration and cannot be used for general analysis. To remove these deficiencies a research was carried out at the Boeing Company and as a result of this work "the rank technique" was developed. This technique furnishes a method for automatic selection of redundancies in the matrix force method and removes the need to generate the basic and redundant load systems.

An automated structural analysis system which adopts the force approach and incorporates the rank technique was developed by Robinson **[5]** and he called this technique as "The Rank Force Method". An attractive feature of the method is that the structure is systematically and automatically investigated to determine the basic characteristics such as stability and indeterminacy. A consistent set of redundants is automatically isolated and moment diagram are not required unlike subframe approach to generate [A_{MQ}] and [A_{ML}] matrices. However, the method is not simple and because of the complete generalization it involves large size matrices compared to the other methods and that is why it has not become that popular.

1.2.5 Modified Force Approach

A modified force approach was developed by Patel and Patodi **[6]** for the analysis of rigid jointed framed structures. The approach partly follows the subframe approach to retain its advantage of localized phenomena. In this approach a determinate structure is formed in a special manner to generate [A_{ML}] and use is made of small sub frames for the generation of [A_{MQ}] matrix. This approach differs from the subframe approach in the generation of [A_{MQ}]

matrix. The generation of $[B_F]$ matrix is carried out from the coordinates of forward ends of the members. Thus the labour involved in selecting the released structure and drawing BMD for each redundant is completely saved. The approach does not require any selection of redundants. These are automatically selected as in the rank force approach of Robinson. These new ideas for the generation of transfer matrices (A_{ML} and A_{MQ}) were, however restricted to rigid jointed frames with encastre ends only. To accommodate hinged, roller and guided support conditions, a new matrix called as "Hinge Matrix H_M" is required and a modified form of compatibility equation is needed for the calculation of primary unknowns. The main drawback of this approach is that it requires a different treatment for pin jointed structures and hinged, roller and guided support conditions.

1.2.6 Integrated Force Method

In 1973, a new version of the force method, was developed by Patnaik **[7]** and named it "Integrated Force Method (IFM)". It was shown in a comparison with early force methods that the IFM makes the automation as convenient as stiffness method and yet retains the known potential for superior stress field accuracy of finite element models that is associated with force method solution techniques. He also pointed out that with the further development IFM can become robust and versatile formulation to deal with variety of problems.

1.3 SCOPE AND OBJECTIVES OF THE PRESENT WORK

It is clear from the literature review, presented in the next chapter, that there are two alternative formulations available for the calculation of stresses directly i.e. hybrid stress method and the force method. In hybrid method, the inversion of the flexibility matrix is necessary in order to generate the element stiffness matrix, which may become a huge computational burden, especially if higher order stress field is strongly required. On the other hand, the advantageous and attractive part of the force method is that it allows the forces in the element to be considered directly as unknowns, which is very appealing feature to the design engineer as the properties of the members of structures most often depend upon the member forces rather than joint displacements.

In case of framed structures subjected to static loading; only problems of beam, plane truss and plane frame have been attempted using Integrated Force Method. No work has been reported in the literature for the analysis of space truss and space frame structures. One of the objectives of the present work, therefore, is to systematize and generalize the analytical work including the development of program for each type of skeletal structure with pre- and post - processing facilities using VB as programming platform. Also, the aim is to include the effect of secondary stresses caused by initial strains, temperature variation and support disturbances in the compatibility condition through vector of initial deformation.

In the field of dynamic analysis, except for one problem of propped cantilever beam subjected to a lumped mass at the centre, no work has been reported in the literature for the force vibration analysis of different types of framed structures using IFM. Hence in the present work it is planned to extend the IFM to deal with other types of framed structures for force based eigen value analysis considering both Lumped and Consistent mass matrices including development of the program for the same.

It is also clear from the literature survey that Patnaik and his team have developed IFM based formulation for the analysis of isotropic plane stress and plate bending problems. The formulation is based on selection of displacement functions using Langragian/Hermite or generalized polynomial, and proper selection of stress polynomial. One of the objectives of the present work is to extend the IFM to handle two dimensional problems of continuum structures by appropriate selection of stress and displacement fields and to demonstrate its applicability to a variety of plane stress, plane strain and isotropic plate bending problems. It is also proposed to develop the formulation for different types of orthotropic plate problems using suitable displacement and stress polynomial functions.

Further, the objective of present work is to develop a rectangular element formulation with nine force and 12 displacement degrees of freedom to deal with plate bending problems under different inertial properties using both lumped and consistent mass matrices to obtain the natural frequency, internal forces and mode shapes for each frequency value.

The objective of the present work is also to develop a modified integrated force method known as Dual Integrated Force Method (DIFM) for variety of problems of framed and continuum structures and to compare, where possible, with the known solutions to validate the proposed formulation.

Also, one of the important objectives of the present work is to develop the IFM based formulation in the polar coordinates to handle different types of circular and annular plate bending problems and to validate the formulation by comparing the results with those available in the literature.

It is also one of the objectives of the present work to carry out the buckling analysis of different types of beams, plane trusses and plane frames using Integrated Force Method. It is also planned to develop the necessary formulation to handle plate buckling problems.

1.4 ORGANIZATION OF THE THESIS

Chapter 1 starts with the classification of structures and brief account of various methods of analysis available for the same with due emphasis on flexibility in force based methods. After highlighting the scope and objectives of the present work, it gives brief account of the organization of the thesis.

Chapter 2 is devoted to literature survey. Some of the major developments taken place in the analysis of structure in last eight decades are highlighted with specific reference to matrix methods, finite element method and integrated force method.

In **Chapter 3** concepts, methodology and variational functionals adopted in IFM are discussed in detail. Different issues related to compatibility conditions are also discussed with the emphasis on force compatibility conditions. Derivation of Equlibrium Equations (EE), Compatibility Conditions (CC) and Deformation Displacement Relations (DDR) are discussed. How concatenation of compatibility matrix [C] with basic equilibrium matrix gives global equilibrium matrix [S] is also explained.

A new matrix based method is developed with minor modifications in the IFM based formulation in **Chapter 4** and is named as Dual Integrated Force Method (DIFM). Different types of elements are formulated to facilitate analysis of different types of framed structures.

Different stress functions which are required for solving different types of 2D in-plane problems and out of plane bending problems are discussed in **Chapter 5.** Firstly, a rectangular element (RECT_5F_8D) having 8 displacement and 5 force degrees of freedom is discussed. Next, development of higher order elements by using Airy Stress Functions is explained. Also, force and displacement polynomials for triangular and curved elements are discussed. For plate bending problems, the use of force polynomial with different terms and approximating the displacements using Hermitian formula is discussed.

After highlighting some of the important features of selected environments for the development of the programs in **Chapter 6**, necessary matrices are derived in **Chapter 7** for solving different types of framed structure problems thru IFM. Computer program for plane and space structures are prepared using VB.NET. Results obtained are depicted in terms of internal moments and nodal displacements for different types of framed structures and are compared with the available solutions.

Chapter 8 is devoted to dynamic analysis of various types of framed structures. Matrices required for dynamic analysis are formulated and a program is developed in VB.NET to find natural frequencies and thus the internal forces are calculated in addition to nodal displacements. IFM results are compared with the available solutions.

In **Chapter 9** different types of plane stress and plane strain problems are solved. Plate bending problems are also attempted using RECT_9F_12D element under different types of loading and boundary conditions. Linear Independent Unknown Technique is employed to calculate compatibility matrix [C] from the basic equilibrium matrix [B] using Matlab software. Computer program is also developed in VB.NET with input and output modules to handle different types of 2D problems.

Chapter 10 is devoted to free vibration analysis of plate bending problems using IFM formulation. First four natural frequencies are calculated and internal moments and nodal displacements are worked out. The values calculated here for natural frequencies are validated using the standard available solutions in the literature.

After discussing the different types of orthotropy in **Chapter 11** modifications required in the formulation to take care of material orthotropy are presented and a variety of problems of GFRP (Glass Fiber Reinforced Plastic) plate problems are solved under different boundary conditions. Problem of bending of Reinforced Cement Concrete Slab with double sided stiffened beam in only one direction is also attempted.

Dynamic analysis of orthotropic plates is attempted in **Chapter 12**. Values of natural frequencies are calculated using the standard Eigen operators given in Matlab. For the first four frequencies, internal moments and nodal displacements are calculated using the secondary equations.

Various matrices required in IFM are formulated in polar coordinates to deal with axisymmetric plate problems in **Chapter 13**. A new element (CIRC_2F_4D) is developed with two force and four displacement degrees of freedom. A variety of circular and annular plate problems are solved and results are validated by comparing with the known solutions.

Chapter 14 is aimed at development of IFM for the buckling problems. After deriving the geometric stiffness matrix, different types of problems are solved. It includes beam, truss and frame problems in addition to examples of buckling of rectangular plates. Comparison is made with the known solutions to validate the formulation and computer implementation.

Finally, **Chapter 15** highlights the conclusions and contributions of the present work followed by the scope for the future work.

CHAPTER 2

LITERATURE REVIEW

2.1 HISTORICAL OUTLINE OF MATRIX METHODS

The concept of framework analysis emerged during 1850-1875, due to the large amount of collaborative effort of Maxwell, Castigliano, Mohr and others. The concepts developed by them truly represent the corner stone of methodology of Matrix Structural Analysis (MSA), which did not take proper form and shape for about 80 years. Even overall progress in the development of theory plus analytical techniques was very slow from 1875 to 1920. This was due to grass-root limitation of solving a large number of algebraic equations required to get a large number of unknowns. The structures of primary interest in that period were basic pin-jointed and rigid-jointed structures which were mainly studied using truly force approach with force parameter in a member as unknown.

In 1915, Maney in United States presented the method of Slope Deflection expressing the moments in terms of deflections and slopes at the rigid joints of framed structures. A similar idea was put forth by Ostenfield in Denmark. These ideas are considered as forerunners of MSA. Handling large size problems, was considered difficult till 1930 in both the approaches i.e., force and displacement based methods. Structural analysis work geared up, when Hardy cross in 1930 **[8]** introduced a method of Moment Distribution. This method made feasible the solution of different types of problems with various complexities in very easy manner, which were considered otherwise quite difficult using other methods. Thus moment distribution method become a strong staple in structural analysis for next 25 years.

In 1943, Courant **[9]** addressed the topic of theoretical and applied mathematics. The equivalence between boundary value problems of partial differential equations on the one hand and problems of the calculus of variations on the other hand was the main theme of the discussion in the paper. Variational formulations are fundamentally simpler to use for approximating extremum problems for different practical applications. Thus, methods emerged which could not fail to attract engineers and physicists; after all, the minimum principles of mechanics are more suggestive than the differential equations. Great successes achieved in applications were soon followed by further progress in the understanding of the theoretical background.

In the year 1945, Southwell **[10]** explained how the relaxation method helps in converting the higher order mathematically complex problems to lower degree by different numerical techniques. He also suggested a new approach for stress calculation in frameworks by the method of relaxation.

Later on digital computer appeared first in early 1950's, but there real significance to both theory and practice did not become widely apparent immediately. Some of the researchers attempted codification of well established frame work analysis procedures in a format suited to the digital computer, which is known as today's matrix format. Later on Argyris, Kelsey, Turner, Clough and Martin combined the concept of frame work analysis and continuum analysis, which resulted in the complete procedure in matrix format.

The use of the so-called matrix methods is nothing but reformulation of existing methods and principles in matrix form, at the same time introducing a generalization of such methods because of the capabilities of computer techniques. The flexibility method is a generalization of the consistent deformation method that enables the engineer not only to include all possible types and directions of loads but also to calculate at the same time all the desired internal forces, reactions, and displacements. Displacements that are needed in this method are usually calculated by the unit load method. The stiffness method is a generalization and extension of the well-known slope deflection method that makes it possible to account not only for the effect of bending moment but for all types of deformations, such as those caused by shear, axial forces, twisting moments, and so forth. In addition, this method allows the determination of all member end actions and reactions at the same time.

Jenen and Dill in 1944 **[11]**, Kempner in 1945 **[12]**, Benscoster in 1946 **[13]**, Plunkett in 1949 **[14]** and Falkenheiner in 1953 **[15]** were the first to develop the basic principles of the matrix force method, with particular reference to standard orthogonal stringer sheet assemblies under loads. Falkenheiner's paper included the clear statement on the importance of self equilibrating load systems as redundancies and also the derivation of the flexibility matrix for the assembled structure. The principle of least work was used in the derivation of the governing equations. Falkenheiner's original contribution was nevertheless, more advanced than much of the subsequent work on the force method.

The first person to apply the force method to the swept wing aircraft structure appears to have been Levy **[16]**. In 1947 Levy idealized the swept wing into an assemblage of simple structural units to which the force method could be applied. The idealization was more realistic than those previously used. Subsequently, Langefors in 1952 **[17]** gave another independent account of the force method based on minimum strain energy approach. Wehle and Lansing in 1952 **[18]** in their paper described clearly the basic force method and the flexibility matrix of an assembled structure. The discussion of redundancies was rather vague with no mention of self equilibrating load systems. They tabulated the flexibilities of various components and suggested to derive matrices by writing down and solving formally all the equilibrium equations with some of the internal forces taken as redundancies. Lang and Bisplinghoff in 1951 **[19]** obtained a matrix formulation of a strain energy analysis for a sweptback wing given previously by Levy **[16]**. Langefors in 1955 **[20]** discussed very briefly the relative merits of the force and displacement methods. He used the matrix formulation of the unit load therein but in a more restricted form, and considered it as following from strain energy or Kron's work **[21]**. Also, he proposed to solve highly redundant systems by splitting them into a number of subsystems, each of which is to be analysed separately.

A recent modern exposition of the elements of the force method was given by Denke in 1954 **[22]**, who effectively derived the matrix formulation of the unit load theorem, which he quoted to arise from Maxwell-Mohr approach. He selected as redundancies forces at hypothetical cuts by inversion of equilibrium conditions. Denke argued that since the strains in the Maxwell-Mohr relations may be due to any cause, the incorporation of thermal strains or other initial strains is straightforward. Denke also showed how the procedure could be applied to certain non-linear cases **[23]**.

The literature on the displacement method is not so extensive. For aeronautical structures the first semi-matrix scheme was proposed by Levy 1953 **[24]**. He considered as components the sparse and ribs for bending and the cells for torsion, the stiffness of which were obtained by inversion of the flexibilities.

In 1956, Livesley **[25]** gave the concept of automated structural design of structural frames in which he considered the problem of finding the lightest structural frame of given geometrical form. Following the development of geometrical analogue and an iterative method of solution, an analytic technique was presented which gave an exact solution to the problem discussed in the paper. A brief description of program developed was included and it was shown that with slight modification in the developed program it can be used to determine the collapse load of a given frame.

These initial contributions were followed by the work of Argyris in 1954 **[26]** and Argyris and Kelsey in 1957 **[27]**. Argyris and Kelsey subsequently **[28]** in their book gave a very general formulation of matrix structural theory based on the fundamental energy principles of elasticity. Although the work emphasized the force method, it also showed that the displacements, rather than internal forces, could be chosen as the primary unknowns of the structural problems. This alternative choice of unknown than brought the stiffness matrix into the formulation. In this early work the stiffness matrix was not calculated directly as in present days work, but rather was obtained by inverting the more familiar flexibility matrix at that time. Hence the stiffness method was considered as an alternative choice for solving the equations generated from the force method. Thus, Argyris and Kelsey presented a format unification of force and displacement methods using dual energy theorems. Although practical application of the duality proved ephemeral, this work systematized the concept of assembly of structural system equations form elemental components.

A mixed force displacement method was proposed by Klein in 1957 **[29-30]** who presented the force displacement relations for all elements. The two groups of equations were solved jointly for the forces and displacements. The matrix formulation appeared only in the final equations. Since both forces and displacement were considered as unknowns, the number of unknowns were inevitably very high. Kron in 1959 **[31]** developed a theory to solve the complex structures by dividing the structure into number of subsystems which could be analysed separately. These subsystems when joined by links represented the static redundancies of assembly of subsystems.

The above publications carried the traditional force and displacement methods to an advanced stage of development. Although the method thereby became applicable to wide classes of structural problems, it took several years for this work to become widely known outside the aircraft industry.

In 1966, Bazant **[32]** gave complete account of theoretical and historical development in analysis of framed structures. He also demonstrated that matrix analysis of framed structures consists of two major methods in which stiffness method is an extension of slope deflection method and flexibility method is an extension of consistent deformation approach.

In 1968, Prezemieniecki **[33]** explained in detail the background required for deriving the stiffness and flexibility properties of all types of structural elements. He suggested various procedures for deriving the above properties using theorems based on virtual work and complementary work principles. He included various examples in his book to validate the above structural properties. The concept of substructure technique was also given in a systematic way for stiffness as well as flexibility matrix based approaches. The book also contains sufficient details regarding the dynamic analysis that is applicable to framed structures using various mass inertial properties. Using stiffness and flexibilities properties, different types of problems of vibration analysis are attempted. Concept of critical damping is also explained in detail accompanied with solution of problems based on damped structural system. Different aspects of nonlinear structural analysis are explained with reference to bar and beam elements using geometric stiffness matrix. Various important aspects of large deflection analysis using matrix force method are also given in his book in a systematic manner.

2.2 FINITE ELEMENT METHODIZATION

The finite element method had its beginning in the area of structural analysis. The first developments were in the aircraft industry, where researchers were striving to model the thin membranes of the fuselage and wing of a jet liner. Membrane elements were used in conjunction with the already established beam and frame elements. A classic paper by Turner, Clough, Martin and Topp appeared in the Journal of Aeronautical Science in 1956 **[34]**. This marked the beginning of the analysis of large complex structural systems. In 1960 Clough **[35]** coined the term "Finite Element Method" which was developed as an extension to established structural analysis techniques. This development is often noted as the beginning of modern finite element analysis.

Two earlier classical papers in mid 1950, by Argyris & Kelsey **[26]** and Turner et al. **[34]** merged the initial concepts of discretized frame analysis and continuum analysis and kicked off the explosive developments in the finite element methods. Following the linear state formulation for each finite element, extension for practical applications have continued to include various field problems.

Additional papers concerned with the basic theory appeared during the 1960s. For example, Melosh's doctoral work in 1963 led to a paper **[36]** placing the finite element method on the principle of minimum potential energy. Also in 1963, Besseling **[37]** presented the analogy between the matrix equation of structural analysis and continuous field equations of elasticity. The question of upper and lower bound was discussed by Venbeke **[38]** in a basic paper that introduced the alternative possibility of defining stress or equilibrium elements based on the principle of minimum complementary energy.

Other papers further demonstrated the rich theoretical base offered by the variational principles for defining finite elements. Jones in 1964 **[39]** pointed out the advantages that could be secured by using Reissner's general variational principle. This led to the development of mixed-element, which depends on assumed displacement and stresses. However, the conditions to be satisfied by these assumed functions are considerably less stringent than those required when seeking a displacement i.e. compatible or stress i.e. equilibrium element. Hence this approach is quite useful when certain complex elements are to be derived.

A variant of Jones theory was also published in 1964 by Pian **[40]**. In this also, element specifications are defined in terms of both displacements and stresses. However, the variation formulation was in terms of both the minimum potential and complementary energy principles. Again the added flexibility led to advantages of particular value for certain complex cases. This approach has led to what is now known as the hybrid element.

Establishing the finite element on the variational principles led to advances that would have otherwise been impossible. This also permitted questions concerned with boundness and the convergence to be discussed with rigor. In addition, the variational formulation permitted a unified approach to be used for determining generalized nodal forces for surface tractions, body forces, thermal gradients, inertia forces and so on. Hence, it became possible to express fully the general elasticity problem in finite element terms. From a conceptual point of view the new theoretical foundations permitted the physical element to be replaced by its mathematical equivalent. The element thus could now be visualized as a small region of space within which the unknown function was to be prescribed in a simple manner. Moreover, the conditions to be met in choosing the function could be stated with certainty. This immediately lifted the method outside the borders of solid mechanics.

These ideas and the underlying theory became widely known with the publication of the text by Zienkewicz and Cheung **[41]**. This was the first comprehensive treatment of the subject, and the text had a far-reaching influence on subsequent developments.

In structural dynamics, three developments are of particular interest. The first was by Archer **[42]**, who in 1963 showed how to correctly determine the mass matrix for distributed mass systems, when using the finite element method. This made the mass matrix consistent with the determination of the stiffness matrix and generalized nodal forces in the stiffness equation. Two years later Guyan **[43]** pointed out how the mass matrix could be reduced in a manner consistent with the reduction of the assemblage stiffness matrix. Such reductions are invariably used in dynamics calculation since they permit the eigen value problem to be condensed to a numerically suitable size. Then Hurty in 1965 **[44]** presented a technique for undertaking the dynamic analysis of higher order structural systems. This procedure is based on using natural modes of vibration of structural subsystems, calculated in terms of displacement mode functions for the various components.

The period from 1965 may be considered as golden age of finite element developments. Any attempt at cataloging this vast literature would require many pages. A brief account of some of the summary papers along with papers which are related to present work is given below.

In 1966, Burton **[45]** handled a two-dimensional problem of equilibrium of a perfectly elastic body with coupling stresses criteria. The general form of solution of the problem in the case of an orthotropic medium indicated that there exists a close analogy between the equations governing the behavior of a plane rectangular lattice composed of rigidly interconnected elastic beams, and the general set of equations of the two-dimensional couple-stress theory for certain orthotropic bodies.

In 1968, Fellipa and Clough **[46]** formulated a fully compatible general quadrilateral plate bending element. The element was assembled from four partially constrained linear curvature compatible triangles, arranged such that no mid-side nodes occur on the external edges of the quadrilateral; thus, the resulting element had only 12 DOF. Also they described the modifications required for incorporating shear distortion. Results were presented for static analyses with and without shear distortion, plate vibration and plate buckling studies; all performed with this quadrilateral element. They claimed it to be the most efficient bending element.

Gallagher in 1969 presented detailed account of contributions to two important subject areas. One treats plates and shells **[47]** while the other discusses the finite element method's contributions to problems of stability analysis **[48]**.

In 1972, Neale, et. al. **[49]** formulated triangular and rectangular elements, both based on the hybrid formulation for the analysis of plate bending problems. Static and dynamic analyses were performed for which results were presented to prove superiority of the hybrid elements over the equivalent displacement elements. Results were also compared with those obtained with elements having strain degrees of freedom.

In 1975, Kikuchi **[50]**, studied convergence rates of the ACM non-conforming scheme for thin plate bending elements with the shape of the domain as rectangular and the exact solution being sufficiently smooth. Error bounds of moments and deflection and error of eigen value were found of the order of square of the maximum mesh size. This result was also confirmed by numerical experimentation.

Also in 1975, Anderson **[51]** formulated a Mixed isoparametric element for the Saint-Venant torsion problem of laminated and anisotropic bars. Both triangular and quadrilateral elements were considered for the analysis purpose. The "generalized" element stiffness matrix was obtained by using a modified form of the Hellinger - Reissner mixed variational principle. The working of the mixed iso-parametric elements was demonstrated by means of numerical examples.

In 1978, Gupta and Rao **[52]** developed the stiffness and mass matrices for a twisted beam element having linearly varying breadth and depth. The angle of twist was assumed to vary linearly along the length of the beam. The effects of shear deformation and rotary inertia were considered in deriving the elemental matrices. The first four natural frequencies and mode shapes were calculated for cantilever beams of various depth and breadth taper ratios at different angles of twist. The results were compared with those available in literature.

In 1985, Gallagher and Ding **[53]** attempted diversified application of force method in optimization process, based on the size variable-independence of the equilibrium equations in matrix format. A rational reduced basis reanalysis and approximate reanalysis methods were presented. Because the static redundancy of pin-jointed structures is often low, and the decomposed coefficient matrix is known, these techniques can be advantageous in the structural optimization process. Several truss structures were studied for the purpose of analysis and validation of results.

In 1993, Argyris and Tenek **[54]** formulated a three-nodes layered triangular element constrained to comply with an assumed linear direct strain distribution across its thickness and including transverse shear deformation for bending analysis of plates. The concepts of the natural mode and matrix displacement methods together with decomposition and lumping ideas based on appropriate kinematic idealizations were combined for stiffness matrix derivation.

In 1993, Wilson **[55]**, summarized the evolution of computational and numerical methods for the static and dynamic analysis of finite element systems. The majority of the material discussed was based on the reflection of the personal experience of the author's working for the past 35 years. It was the opinion of the author, that at the present time, the finite element method is far from being completely automated for complex structures.

In 1993, Kosmatka and Friedman **[56]** developed an improved two noded beam element with appropriate stiffness, mass, and consistent matrices. Timoshenko beam element was developed based upon Hamilton's principle. Cubic and quadratic polynomials were used for the transverse and rotational displacements respectively, where the polynomials were made interdependent by requiring them to satisfy the two homogeneous differential equations associated with Timoshenko's beam theory. Numerical results were presented to show that for short beam subjected to complex distributed loading the proposed element predicts shear and moment resultants and natural frequencies better than the existing Timoshenko beam elements.

In 1996, Gupta and Meek **[57]** presented summary of the works of several eminent authors and research workers associated with the invention of the analysis technique now referred to as the Finite Element Method. It is believed to be an accurate record of the historical sequence of published papers on FEM in the international literature. The complete development of the ideas, which led to the method of analysis in which the field equations of mathematical physics are approximated over simple regions (triangles, quadrilaterals, tetrahedrons, etc.), and then assembled together so that equilibrium or continuity is satisfied at the interconnecting nodal points, are presented in interesting manner by the authors.

In 1998, Fellipa and Park **[58]** presented variational framework for the development of partitioned solution algorithms in structural mechanics. This framework was obtained by decomposing the discrete virtual work of unassembled structure into that of partitioned substructures. New aspects of the formulation were an explicit use of substructural rigid-body mode amplitudes as independent variables and direct construction of rank-sufficient interface compatibility conditions. The resulting discrete variational functional was shown to be variationally complete, thus yielding a full rank solution matrix. Four specializations of the suggested framework were also discussed in detail.

In the year 2000, Fellipa **[59]** gave again a historical background of major contribution made by Collar, Duncan, Argyris and Turner, who really worked hard and shaped the Matrix Based Structural Analysis. He divided the complete development into following three parts: (1) Formative period in which methodological concept were developed, (2) The period in which Matrix methods assumed bewildering complexity in response to conflicting demand and restrictions, and (3) The period in which development of Direct Stiffness Method took place, through which MBSA completely morphed into the present implementation of structural mechanics

In year 2005, Kikuchi **[60]** presented theory and examples of partial approximation as a modification of the displacement based FE analysis. This method needs various types of shape functions for different terms in the potential energy expression to curtail the processes in the standard displacement method. The theory is explained with simple example.

In year 2006, Samuelsson and Zinenkiewicz **[61]** presented a brief history of the development of stiffness method tracing the evolution of the complete method for discrete type of structures i.e. trusses and frames composed of two noded members. Brief description is also given in the same paper for the application of the same method for continuum type of problems, which are modeled by finite difference and finite element methods.

In year 2006, Oztorun **[62]** developed a FE based formulation for the static and dynamic analysis of linear-elastic space structures. He suggested that the finite element method can be efficiently used for the analysis of linear-elastic structures with shear walls. The element considered for the study had six degrees of freedom at each node and an in-plane rotational degree of freedom, which makes it compatible with 3D beam-type finite element model. The rigidity coefficients of the element were determined analytically. The element can be used with facility for the modeling of a shear wall with the connections of slab components. Convergence studies were carried out on actual structures by using several models to check the performance of the rectangular finite elements. Acceptable results were obtained with coarse mesh and reasonable convergence was observed on the models tested.

Kuoa et al. **[63]** in the year 2006, proposed a simple method for deriving the geometric stiffness matrix (GSM) of a three-noded triangular plate element

(TPE). It was found that when the GSM of the element is combined into the global matrix, the structural stiffness matrix becomes symmetric and satisfies both the rigid body requirements and incremental force and moment equilibrium (IFE) conditions. In addition, the advantage is that the derivation of the matrix needs only simple matrix operations without cumbersome non-linear virtual strain energy derivations and tedious numerical integrations.

In 2007, Kaveh et al. **[64]** developed force based methods which were found highly efficient for the generation of sparse and banded null bases and flexibility matrices. However, these methods require special considerations when the support conditions are indeterminate. These considerations with special methods were presented in the paper, which lead to efficient utilization of force method for indeterminate support conditions with no substantial decrease in sparsity.

In 2011, Lang and Yan **[65]** found that finding deformation and internal force of rectangular plate under complex boundary and free corner supported condition is comparatively complicated. Based on the elastic thin plate flexure theory, the rectangular plate with complex boundary was firstly divided into several simple rectangular elements. Introducing the concept of generalized supporting edge in the common boundary, the unknown coefficients of the plate bending functions were determined by satisfying the boundary conditions and thus the geometric compatibility conditions, and the analytic solution of the original plate was finally obtained. It provides a valuable reference for the inner force analysis and engineering design of rectangular plates under various edge and loading conditions.

2.3 EVOLUTION OF INTEGRATED FORCE METHOD

A novel formulation was developed in 1973 by Patnaik **[7]** for the analysis of discrete structures by considering member forces as primary unknowns instead of conventional way of treating the redundants as prime unknowns. He named the method as Integrated Force Method (IFM). Illustrative examples for the determination of forces and displacements were presented for pin-jointed and rigid connected frame structures for various load conditions.

In 1977, Patnaik and Yadagiri **[66]** developed IFM for frequency analysis of spring systems, beams and trusses. In the paper, this concept of eigen analyses was explored considering force mode shape as primary variable. After calculating the frequency, internal forces and moments were calculated using the homogenous equation by substituting each frequency value. Utilizing the force mode shape concept, direct design of structures was attempted by keeping the frequency constrains.

In 1984, Patnaik and Joseph **[67]** developed two schemes for the generation of compatibility conditions for discrete structures which were used in IFM. The first scheme was based on displacement deformation relation and was recommended for the basic analysis of structures. The second scheme was developed using Linear Programming (LP), in which Equilibrium Equations (EEs) were used as constraints and linearized internal strain energy of structure as objective function. Linear programming had advantage of sparsity of coefficient matrix. Both the schemes were included in IFM and thus using the same different problems were attempted

In the year 1986, Patnaik **[68]** compared the IFM with the Standard Force Method (SFM). Standard Force Method was considered as solution technique as a part of IFM for static analysis. IFM bypasses the popular concept of selection of redundant. IFM was applied to twenty bay truss and cantilever plate problems. It has been observed in the paper that the numerical stability of IFM is much better and superior than the SFM.

In the year 1989, Patnaik **[69]** extended the application of Integrated Force Method to structural optimization problems, in which prior knowledge of design variable was necessary. In the paper two methods were presented; first was of Linear Programming and other was based on fully utilized design concept. The Nonlinear Mathematical Programming (NLP) was elegantly removed without disturbing the main aspect which resulted in considerable reduction in computational time.

Nagabhushanam and Patnaik **[70]** in the year 1989 explained the basic methodology of IFM, in which development of equilibrium equation and compatibility conditions were explained in detail. The generation of EEs was very simple, and straightforward while the development of CCs was intricate. They considered both the conditions i.e. field and boundary compatibility conditions with applications to FE based problems. The key feature in the development of compatibility conditions was based on the concept of node determinacy, which was used to reduce the complexity of the displacement deformation relation. It also helped to reduce the mathematical complexity by eliminating the node at intermediate level.

In the year 1990, Patnaik et al. **[71]** applied IFM to plane stress and plate bending problems of deep cantilever beam and clamped plate subjected central point loading. Software named as GIFT was developed based on IFM and displacement based modified approach which was named as IFMD. The same problems were solved by commercially available software "MSc-NASTRAN", which was based on displacement based FE approach and another software "NHOST", which was based on mixed method. IMF with simultaneous emphasis on stress equilibrium and strain compatibility gave acceptable solutions even for coarser discretization.

In the year 1990, Patnaik and Satish **[72]** used the Boundary Compatibility Conditions (BCCs) that geared the force method and brought up IFM upto the level of stiffness method. Based on the new concept and theoretical development of BCCs novel formulation, various problems of static and dynamic analysis of framed and continuum structures were attempted which provided quite accurate results.

In the year 1996, Kaljevic et al. **[73]** explored the possibility of finding secondary stresses in beams and circular annulus by using IFM. The initial strain was directly incorporated into the compatibility conditions using vector of initial deformation. Two problems were successfully attempted i.e. support settlement in two span continuous beam and thermal strain effect in circular annulus.

In the year 1996, Kaljevic et al. **[74]** developed triangular and quadrilateral elements with necessary displacements and stress fields. Elements were developed by discretizing the potential and complimentary strain energies. The displacement field was approximated using the Lagrange's, Hermitian's and generalized coordinate systems. The stress field was approximated by using the complete polynomial of correct order. Airy's stress theory was explored for its constant, linear, quadratic and cubic stress fields which were used in IFM. The resulting matrices were also checked for equations of equilibrium and conditions of compatibility. The elements were insensitive to the orientation from local to global axis. Elements having large number of unknowns in the stress field increased the size of matrices and finally to work out the solution became cumbersome. Thus, in the same paper different schemes were discussed to reduce the number of unknowns without disturbing the accuracy. For comparison purpose, quadratic, linear and constant fileds were assumed. Various examples were considered to validate the developed library of 2D elements. The results were checked and verified with the available solutions.

A two dimensional beam element having eight displacement degrees of freedom and five force degrees of freedom was used in the year 1997, by Patnaik et al. **[75]** for frequency analysis by using IFM and IFMD. Two lumping mass were considered at extreme nodes for which force mode shape based frequency analysis was carried out. From the frequency vectors, internal forces and mode shapes were worked out. Solutions for frequencies and force mode shapes were compared with the standard displacement based eigen value analysis and analytical solutions.

In the year 2004, Patnaik et al. **[76]** gave an account of theoretical development with various numerical examples of framed and continuum structures. It was just a summary of the work done by Patnaik and his team on IFM till that year.

In the year 2005, Dhananjaya et al. **[77]** presented a formulation of a 2D element based on Mindlin - Reissner theory with element having 12 displacement degrees of freedom and 9 force degrees of freedom using IFM. The performance of the element was checked by solving different benchmark problems. Results for deflection and internal moments were checked by the exact solution and displacement based FE method; a good agreement was indicated.

Also, in 2005, Dhananjaya et al. **[78]** extended IFM to laminated composite plate bending problems. A quadrilateral element having total 20 displacement degrees of freedom and 16 force degrees of freedom was developed including shear deformation theory. Simply supported plates under central point load and uniformly distributed loading were considered for the analysis purpose. Different discretization schemes were used for checking the accuracy and convergence with reference to an exact solution.

In the year 2005, Dhananjaya et al.**[79]** also developed a general purpose program for automatic generation of the equilibrium and flexibility matrices which were based on Kirchoff's and Mindlin-Reissner's plate bending theory. Using the equilibrium and flexibility matrices for triangular, rectangular and quadrilateral elements, different types of thin and thick plate bending problems were attempted by using the IFM formulation. The performance of the developed element was evaluated for its accuracy and convergence and it was found satisfactory.

Further, in the year 2005, Dhananjaya et al. **[80]** developed closed form equilibrium and flexibility matrices for four noded and eight noded plate bending elements. The necessary matrices were developed by discretizing the different strain energies required. Numerous standard plate bending problems were solved to get results for deflections and internal moments; a good agreement was indicated, with the available solutions.

In the year 2006, Patnaik and Pai **[81]** formulated the boundary compatibility conditions in the incomplete form of Beltarmi's Michells formulation. It is now recognized as "Completed Beltrami's Michell's Formulation (CBMF)". Using this concept different problems of circular boundary with mixed boundary conditions were efficiently attempted.

Recently, in the year 2010, Dhananjaya et al. **[82]** developed closed form solutions for equilibrium and flexibility matrices by using Mindlin-Reissner's theory of plate bending based on IFM. The rectangular element with 12 displacement degrees of freedom and 9 force degrees of freedom was developed. Large scale plate bending problems were attempted using IFM and results for deflection and moments were calculated and compared with the available small deflection theory.

CHAPTER 3

INTEGRATED FORCE METHOD AND ITS FORMULATION

3.1 PREAMBLE

Navier's table problem (1785) perhaps was the initiator of the analysis of indeterminate structural problems. Navier wanted to calculate the four reactions induced at the foot of the table, but he had only three standard equilibrium conditions. The problem was one degree statically indeterminate. He developed one additional condition i.e. deformational compatibility condition simply by indirect approach, as the available compatibility formulation was just insufficient, incomplete or adhoc in nature to deal with the structural mechanics and Theory of Elasticity.

Despite of immature and insufficient compatibility condition, the development in structural mechanics continued but only through indirect approach and methods were categorized as either displacement method or force method. In the displacement method, the possible nodal displacements are constrained and by applying unit displacement in the direction of constrained displacement stiffness coefficients are worked out. Thus, by considering primary unknowns as displacements, first of all unknown joint displacements are calculated and then the secondary unknowns such as reaction and internal forces are calculated. In the force method, on the other hand, the complete structure is made statically determinate and displacements are calculated in the direction of redundants due to loading. By applying unit force in the direction of primary unknowns, the flexibility coefficients are calculated. Using governing compatibility equations, primary unknowns i.e, redundants, reactions and then other internal actions are calculated.

Infact, the names stiffness method and flexibility method are more diffuse name for the displacement and force methods, respectively. Generally speaking these apply when stiffness and flexibility matrices, respectively, are important part of the modeling and solution process.

Now, the whole pretext regarding immature development of compatibility condition methods for structural mechanics and theory of elasticity can be understood with the help of a Pie diagram which is shown in **Fig. 3.1.**

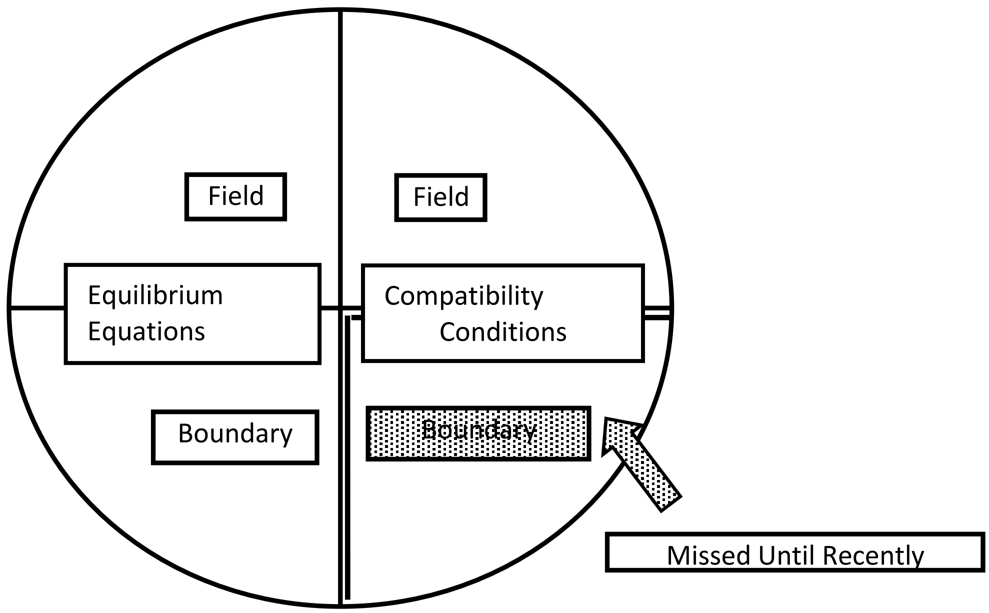


Fig. 3.1 Equilibrium Equations and Compatibility Conditions

It is a fact that all the indirect methods discussed above were just developed by using the three quarter of the pie chart **Fig. 3.1** i.e. field and boundary part of equilibrium equations and field part of compatibility conditions. The remaining component was developed by Patnaik **[7]**, by considering one additional condition which is known as force compatibility condition and Beltrami-Michell's formulation was converted to Complete Beltrami Mitchell's formulation. Thus, Structural Mechanics and Theory of Elasticity become full fledge computational tool for solving the various framed and continuum structure problems.

Using the additional condition in terms of forces with boundary compatibility condition, a new force method was evolved named as Integrated Force Method (IFM), which can complement all the available indirect methods. The IFM is a relatively young approach for the analysis of indeterminate structures, which makes use of both the fundamental equations i.e. equilibrium equations (EEs) and compatibility conditions (CCs). Formerly there was certain degree of asymmetry in the development and utilization of these two concepts. The underlying principle behind the EEs is force balancing likewise the compliance of forces and initial deformations are achieved through the CCs. Augmentation of these CCs with system EEs leads to IFM. It is possible by taking into account of these relations to obtain a complete system of equations which must be satisfied by stress components and thus the way is open for direct determination of forces without solving for components of displacements. In IFM all the internal forces and reactions are directly treated as primary unknowns unlike the displacements and redundants in well-known stiffness and flexibility methods respectively.

3.2 VARIATIONAL FUNCTIONALS FOR IFM

The IFM is one of the five formulations of structural mechanics, where others are Flexibility Method (FM), Displacement Method (DM), Mixed Method (MM) and Total Method (TM). The Pie chart depicted in **Fig. 3.1** shows the role of three parts i.e. Field, Boundary equilibrium conditions and Field compatibility condition in the formulation. Using the stationary condition of variational formulations for the IFM yields the equilibrium equations and compatibility conditions as well as force and displacement boundary conditions. Also, it yields a new set of boundary conditions which was identified as the boundary compatibility conditions before few years. It opened a new thrust for theory of elasticity based problems by considering force boundary condition rather than displacement boundary condition**.** In literature, it has been named as Completed Beltrami-Mitchell Formulation (CBMF), or known as IFM for Theory of Elasticity.

For a two dimensional elasticity problem, the variational functional $\pi_s(\sigma, u, \sigma^e)$ of the IFM is obtained by adding the three expressions as follows.

$$\pi_s = A + B - W \qquad \dots (3.1)$$

In which the variables of the functional π, for variational purpose are the displacements u and redundant stresses σ^e. The BMF has two EEs and one CC in the field expressed in term of stresses. The variational variables of functional π_s^{BMF} are two displacements u and v and one redundant stress function which can be considered as Airy Stress Function ϕ that gives,

$$\sigma_x^e = \frac{\partial^2_\phi}{\partial y^2}\text{-V}; \quad \sigma_y^e = \frac{\partial^2_\phi}{\partial x^2}-\text{V}; \quad \tau_{xy}^e = -\frac{\partial^2_\phi}{\partial x \partial y} \qquad \dots (3.2)$$

The internal strain energy functional represents stresses and strains which are treated independently in explicit form. It is expressed as

$$\text{A}(\sigma, u) = \text{h}\int_s [\sigma_x \frac{\partial u}{\partial x} + \sigma_y \frac{\partial v}{\partial y} + \tau_{xy}(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y})]dxdy \qquad \dots (3.3)$$

The functional $\text{B}(\beta, R)$ gives the compatibility conditions of the structural deformation in two dimensional elasticity. Thus the functional $\text{B}(\epsilon, \sigma^e)$ can be written in the form as

$$\text{B}(\epsilon, \sigma^e) = \text{h}\int_s [\epsilon_x \sigma_x^e + \epsilon_y \sigma_y^e + \gamma_{xy}\tau_{xy}^e]dxdy \qquad \dots (3.4)$$

This $\text{B}(\epsilon, \sigma^e)$ is the complementary strain energy function in which the strains ϵ and redundant stresses σ^e are treated independently. For IFM, as per stress strain law, the strain is converted into stresses. So, the functional for a two dimensional elasticity problem can be written as

$$\text{B}(\sigma, \sigma^e) = \text{h}\int_s [\frac{\sigma_x - \mu\sigma_y}{E}\sigma_x^e + \frac{\sigma_y - \mu\sigma_x}{E}\sigma_y^e + (\frac{1+\mu}{E})\gamma_{xy}\tau_{xy}^e]dxdy \qquad \dots (3.5)$$

The potential of the external force W(P, u) has following three components;

$$W(P,u) = h\int_s [B_x u + B_y v]dxdy + \int_{L_1}[\dot{P}_x u + \dot{P}_y v]dL_1 + \int_{L_2}[P_x\dot{u} + P_y\dot{v}]dL_2$$

$$= I_1 + I_2 + I_3 \qquad \ldots (3.6)$$

The first integral (I_1) is for body forces B_x and B_y in the field S, second integral (I_2) is for the portion of boundary L_1 on which the external loading $\dot{P}_x$ and $\dot{P}_y$ is acting and remaining integral (I_3) is for the boundary L_2 on which displacements $\dot{u}$ and $\dot{v}$ are prescribed as shown in **Fig. 3.2.**

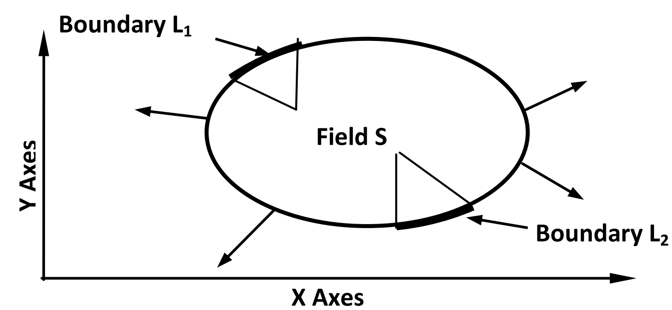


Fig. 3.2 Boundary Compatibility Conditions in BVP

Substituting Eqs. (3.3) to (3.6) in Eq.(3.1) after substituting Eq. (3.2) in Eq. (3.3) gives

$$\pi_S^{BMF} = h\int_s [\sigma_x\frac{\partial u}{\partial x} + \sigma_y\frac{\partial v}{\partial y} + \tau_{xy}(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y})]dxdy + h\int_s [\frac{\sigma_x - \mu\sigma_y}{E}\sigma_x^e + \frac{\sigma_y - \mu\sigma_x}{E}\sigma_y^e + (\frac{1+\mu}{E})\gamma_{xy}\tau_{xy}^e]dxdy - h\int_s [B_x u + B_y v]dxdy - \int_{L_1}[\dot{P}_x u + \dot{P}_y v]dL_1 - \int_{L_2}[P_x\dot{u} + P_y\dot{v}]dL_2$$

$$\ldots (3.7)$$

For the stationary conditions of the VF, in which u, v and ϕ are non zero quantities, making their brackets including coefficients equals to zero leads to the following field equations and boundary conditions:

(1) The field equilibrium equation are

$$\frac{\partial\sigma_x}{\partial x} + \frac{\partial\tau_{xy}}{\partial y} + B_x = 0 \text{ and } \frac{\partial\sigma_y}{\partial y} + \frac{\partial\tau_{xy}}{\partial x} + B_y = 0 \qquad \ldots (3.8)$$

(2) The field compatibility conditions are

$$\frac{\partial^2}{\partial x^2}(\sigma_y - \mu\sigma_x) + \frac{\partial^2}{\partial y^2}(\sigma_x - \mu\sigma_y) - 2(1+\mu)\frac{\partial^{2}\tau_{xy}}{\partial x \partial y} = 0 \quad \dots (3.9)$$

The Eq. (3.9) is CC which can be simplified by using equilibrium equation (3.8) and relation between body forces and its potential $B_x = \frac{\partial u}{\partial x}$, $B_y = \frac{\partial V}{\partial y}$ as

$$\nabla^2(\sigma_x + \sigma_y) + (1+\mu)\nabla^2 V = 0 \quad \dots (3.10)$$

Along the boundary, where forces are prescribed as $\dot{P}_x$ and $\dot{P}_y$, stress boundary conditions are considered which gives

$$\sigma_x n_x + \tau_{xy} n_y = \dot{P}_x \text{ and } \sigma_y n_y + \tau_{xy} n_x = \dot{P}_y \quad \dots (3.11)$$

$$\frac{\partial}{\partial x}(\sigma_y - \mu\sigma_x)n_x + \frac{\partial}{\partial y}(\sigma_x - \mu\sigma_y)n_y - (1+\mu)\left(\frac{\partial\tau_{xy}}{\partial x}n_y + \frac{\partial\tau_{xy}}{\partial y}n_x\right) = 0 \quad \dots (3.12)$$

Here Eq. (3.11) is known as classical stress boundary condition and the boundary condition given by Eq. (3.12) is identified as the novel boundary compatibility condition. Due to which Beltarmi-Michell's Formulation is known as Completed Beltarmi-Michell's Formulation (CBMF) which is IFM procedure for Theory of Elasticity.

3.3 IMPORTANCE OF STRAIN FORMULATION

The strain formulation in theory of elasticity and development of compatibility condition in structural mechanics have neither understood properly and neither utilized in past. Due to this shortcoming, the important development in the direction of methods of analysis for framed and continuum structures got stuck up and diverted to indirect solution techniques. Also, because of this direct stress formulation, which calculates stresses and strains in the structures and continuum could not be developed properly. Using indirect methods, it has been calculated using mathematical differentiation of displacement function, which was developed through an approximate interpolation function. After understanding the importance of strain formulation in terms of Boundary Compatibility Conditions (BCCs) and it has been derived using variational formulation and also verified using mathematical form of integral theorem. Navier's method in elasticity and displacement method in structural mechanics have to take care of this additional condition as an extra step typically at inter-elemental boundaries in discrete element models.

Thus the completion of strain formulation led to revival of direct force calculation method in addition to availability of indirect method, known as IFM and IFMD which are applicable for structural mechanics based framed types problem and Completed Beltarmi-Michells Formulation (CBMF) in theory of elasticity which is based on strain formulation is used to solve 2D plane elasticity problems. Through CBMF now one can attempt problems of direct stress calculation, displacement calculation, mixed boundary value problem with the other major limitations and loop holes of classical force method also being removed; which was in past applicable to only stress boundary condition. The researchers found highest fidelity response by using both the new methods even with coarser FE models for different types of problems. **Table 3.1[83]** shows the different methods of structural mechanics and theory of elasticity with and without use of compatibility condition.

Table 3.1 Various Methods and Associated Variational Formulations

Various Methods		**Primary Variables**		**Variational Formulation**
Elasticity	Structures	Elasticity	Structures	
Completed Beltrami-Mitchell Formulation (CBMF)	Integrated Force Method (IFM)	Stresses	Forces	IFM Variational Formulation
	EEs & CCs Enforced			
Airy Formulation (AF)	Redundant Force Method	Stress Function	Redundants	Complementary Energy
	Field CCs Enforced			

Navier's Formulation (NF)	Stiffness Method Boundary Compatibility Non compliant	Displacements	Deflection	Potential Energy
Hybrid Method (HM)	Reissner Method Boundary Compatibility Non compliant	Stresses and Displacements	Forces and Deflection	Reissner Formulation
Total Formulation (TF)	Washizu Method Boundary Compatibility Non compliant	Stresses, Strains and Displacements	Forces, Deformations and Deflections	Washizu Functional

3.4 BASIC RELATIONS OF INTEGRATED FORCE METHOD

The concept of equilibrium of forces and compatibility of deformations are fundamental to analysis of framed structures. The equilibrium equations are written in terms of forces, which can be axial forces, shear forces, bending moments and twisting moments. The compatibility conditions are expressed in terms of deformations, which can be elongations, deflections and curvatures. *Hence, it is utmost compulsion to express these developed CCs in terms of forces, so that it can be coupled and concatenated from bottom side in a matrix form with Equlibrium Equations, which are already available in terms of forces.* So, to convert the deformations to forces, two additional sets of equations are required. These are the deformation displacement relations (DDRs) and force displacement relations (FDRs). Thus, the four sets of structural mechanics equations required in IFM analysis are:

1. Equilibrium equations (EEs),
2. Deformation displacement relations (DDRs),
3. Compatibility conditions (CCs), and
4. Force deformation relations (FDRs).

3.4.1 Equilibrium Equations (EEs)

A member as shown in **Fig. 3.3**, subjected to a system of external forces, is said to be in equilibrium, when it remains in a position of rest i.e. when the force resultants in the directions of the reference axes are equal to zero. This way three equations for forces and three for moments are available. Force balancing is the central concept behind the equilibrium equations in structural mechanics.

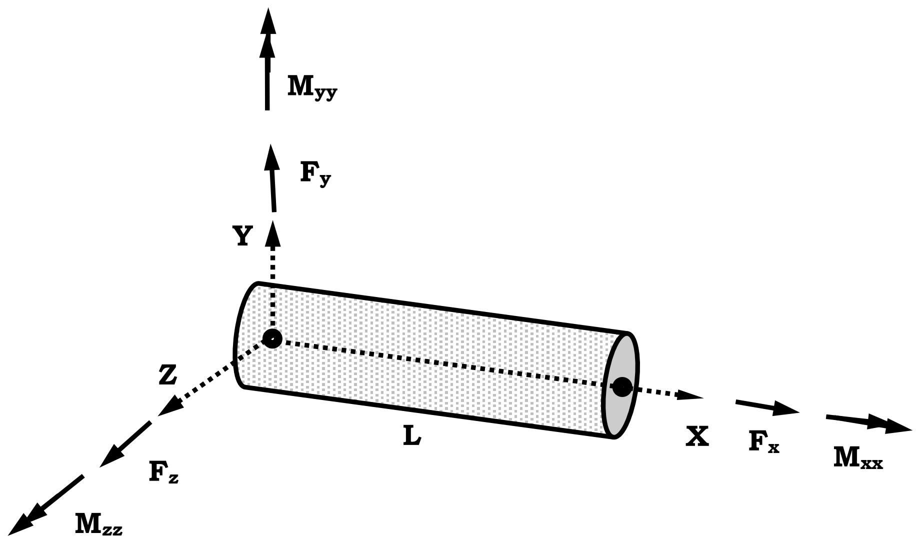


Fig. 3.3 Forces and Moments in Three Directions

Generally equilibrium equations are written by considering either reactions or applicable internal forces as the unknowns. The generation of equilibrium equation is illustrated here with reference to a fixed beam example having length L and flexural rigidity EI, which is subjected to transverse load P as shown in **Fig. 3.4.**

Case 1- External reactions as unknowns

In this case all the applicable external support reactions are treated as unknown forces. Two equilibrium equations for the beam case can be formulated as algebraic summation of forces along y-y axis and moments about z-z axis equals to zero as follows

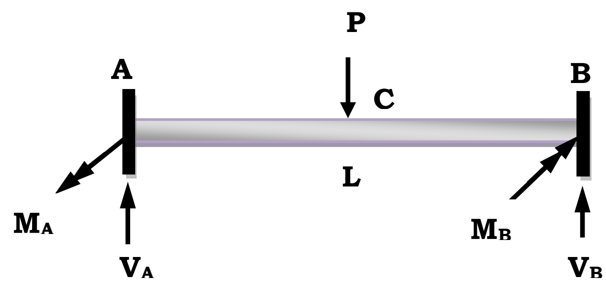


Fig. 3.4 Fixed Beam Example

$\sum F_y = 0$, gives $\quad V_A + V_B - P = 0$ ... (3.13)

$\sum M_{zz} = 0$, gives $\quad M_A + V_B \times L - PL/2 - M_B = 0$ ... (3.14)

Case 2- Internal Moments as unknowns

In this case beam is discretized into two segments i.e. AC and CB. Both the end moments, for each element, are considered as unknowns. Equilibrium equation in this case can be formulated at joint C by considering equilibrium between: (i) External force P and shearing forces of both the members , (ii) Internal moment induced on either side as shown in **Fig. 3.5.**

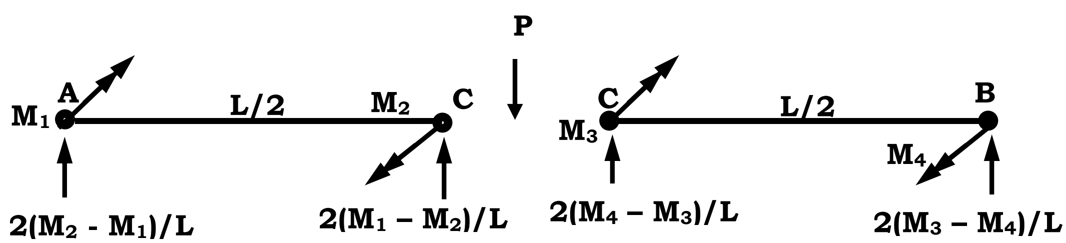


Fig. 3.5 Free Body Diagram of Fixed Beam

Equilibrium Equations are formulated as follows;

$\sum F_y = 0$ at C gives $\quad 2(M_1 - M_2)/L + 2(M_4 - M_3)/L - P = 0$

$\sum M_z = 0$ at C gives $\quad -M_2 + M_3 = 0$ ... (3.15)

3.4.2 Deformation Displacement Relations (DDRs)

Deformation is a change in geometrical shape of the structure due to applied loading. Deformation is often described in terms of strains in mechanics of structures. Displacement on the other hand specifies the position of a point with reference to its original or previous position. The deformation displacement relation (DDR) is an important relation in IFM and is a central component behind both equilibrium and compatibility conditions. The DDR can directly be derived from the equilibrium equations. It depends on the type of material, size and geometry of the member and the forces applied. The deformation is always associated with each type of force variable, i.e. extension in the rod is the deformation due to axial force, bending curvature is the deformation in beam bending, shear deformation is due to shearing forces and twisting angle is the deformation due to twisting action. In the derivation, deformation displacement relation in IFM can be written using the transpose of the equilibrium matrix derived for a particular system. In DDR the system displacements are mapped into the elemental deformations. Symbolically it can be represented as

$$\{\beta\}=[B]^{T}\{d\} \qquad \dots (3.16)$$

where, $\{\beta\}$ are the 'n' elemental deformations in given structure, [B] is equilibrium matrix of size (m x n) and $\{d\}$ are the nodal displacements.

3.4.3 Compatibility Conditions (CCs)

While studying the geometry of the system, the deformation of the small segment of complete structure must be consistent as well as continuous with the overall deformation pattern **(Fig. 3.6)**. It is known as condition of continuity or compatibility.

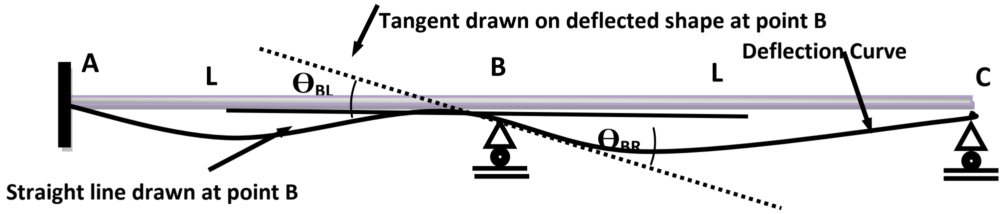


Fig. 3.6 Deflected Shape Showing Slope Compatibility

Whenever structural member deforms due to external loading, at any point of the deflected shape, the displacement function must be continuous, differentiable, and single-valued which takes care of non-distortional and non-overlapping mechanism. The deformational compatibility conditions are necessary for the solution of indeterminate structural problems **[76]**. Also, in IFM sinking of supports and temperature variation can be considered as the initial strain which can be directly accounted through compatibility relations for solution purpose

In IFM, first the DDR is derived and then the elimination of the displacements from the deformation displacement relation is carried out to obtain the compatibility conditions.

3.4.4 Force Deformation Relations (FDRs)

In order to analyse any problem using IFM, force deformation relations are necessary to convert all the compatibility conditions, which are in terms of displacements to independent variables which represent the internal forces for given structural members. The bottom most part of the equilibrium matrix [B] i.e. which represents the equilibrium equations needs this paradigm because upper few rows of [B] are already in terms of forces. Thus, final equation is [B] {F} = {P}. Likewise, the compatibility conditions (CCs) are written in terms of deformations {β} and thus [C] {β} = {0}.

The force deformation relation (FDR) can be obtained from the Hooke's law by simply relating the internal stress developed to the force applied and then deformation to strain induced.

(i) Derivation of FDR for an Axial Rod

For a cylindrical rod having cross sectional area as A and length as L, subjected to an axial force F (**Fig. 3.7)**, the FDR can be written as follows:

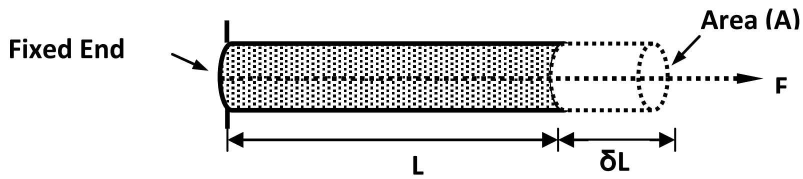


Fig. 3.7 Bar subjected to an Axial Force

As per strength of material relations, $\varepsilon = \dfrac{\delta l}{L}$

In IFM $\epsilon = \frac{\beta}{L}, \ \sigma = \frac{F}{A}$

Now according to Hooke's law stress and strain can be related as

$$\varepsilon = \frac{\sigma}{E} \quad \text{or} \quad \frac{\beta}{L} = \frac{\sigma}{E} \quad \beta = \frac{\sigma L}{E} \quad \text{or} \quad \beta = \left(\frac{L}{AE}\right)F \qquad \dots (3.17)$$

Where $\left(\dfrac{L}{AE}\right)$ is the flexibility coefficient corresponding to an axial force F.

According to Castigliano's theorem, the first derivative of strain energy U with respect to force P is equal to the deformation (**β** in IFM) corresponding to that force P.

$$\beta = \frac{\partial W}{\partial P} = \frac{\partial U}{\partial P} \qquad \dots (3.18)$$

Where strain energy $\quad U = \int_0^L \frac{\sigma\varepsilon}{2} A\, dx$

Replacing σ by F/A and ε = σ/E by F/AE, the strain energy for a uniform bar of area A corresponding to force F can be written as

$$U = \int_0^L \frac{F^2}{2AE}\ dx = \frac{F^2 L}{2AE} \qquad \text{… (3.19)}$$

$$\therefore \text{ Deformation } \quad \beta = \frac{\partial U}{\partial P} = \left(\frac{L}{AE}\right)F \qquad \text{… (3.20)}$$

In indeterminate analysis the energy based derivation for FDR is preferred.

(ii) Derivation of FDR for a Beam Member

Generally shear force and bending moment are the two governing internal forces for the bending deformations in the beams, which are related to each other. However, the strain energy due to shear force as it violates the basic assumption of stress-strain is neglected, only the strain energy due to the moments is taken into account.

From the Castigliano's second theorem,

$$\beta = \frac{\partial W}{\partial P} = \frac{\partial U}{\partial P} = \frac{\partial}{\partial P}\int_0^L \frac{M^2}{2EI}dx \qquad \text{… (3.21)}$$

Where, $U = \int \frac{M^2}{2EI} dx$

To simplify the procedure, one may consider

$$\beta = \int_0^L M\left(\frac{\partial M}{\partial P}\right)\frac{dx}{EI} \qquad \text{… (3.22)}$$

Beam response requires two internal unknown forces i.e. either two end moments or a pair of bending moment and shear force.

The FDR for these two cases is worked out as follows.

Case 1: Consider a beam as shown in **Fig. 3.8** with two end moments (M_1 and M_2) as unknowns with reactions developed as V_A and V_B.

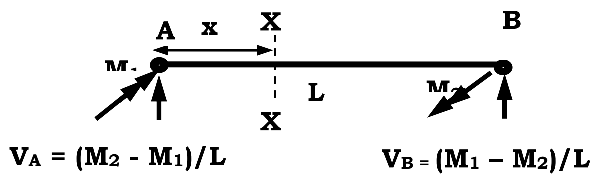


Fig. 3.8 Beam subjected to Moments at ends

Taking moment at any section distance x from A

$$M_{(x)} = V_A x + M_1$$

Substituting the value of V_A from the diagram and differentiating with respect to M_1 gives

$$\frac{\partial M_{(x)}}{\partial M_1} = 1 - \frac{x}{L}$$

So, $\beta_A = \frac{\partial U}{\partial M_1} = \frac{1}{EI}\int_0^L M_{(x)} \frac{\partial M_{(x)}}{\partial M_1} dx = \frac{L}{EI}\left(\frac{M_1}{3} + \frac{M_2}{6}\right)$

Similarly, $\beta_B = \frac{\partial U}{\partial M_2} = \frac{1}{EI}\int_0^L M_{(x)} \frac{\partial M_{(x)}}{\partial M_2} dx = = \frac{L}{EI}\left(\frac{M_2}{3} + \frac{M_1}{6}\right)$ … (3.23)

It can be seen from the above equations that the coefficients associated with the two moment unknowns are the generalized flexibility coefficients. Thus, the two deformations for the beam can be represented in matrix form as:

$$\begin{Bmatrix} \beta_A \\ \beta_B \end{Bmatrix} = \frac{L}{6EI}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}\begin{Bmatrix} M1 \\ M2 \end{Bmatrix} \quad \text{… (3.24)}$$

$$\{\beta\} = [G]\,\{F\}$$

where [G] is the flexibility matrix associated with M_1 and M_2 for the beam.

Case 2: Consider a beam as shown in **Fig. 3.9** with shear force V and bending moment M at a distance L from A as unknowns. As seen above, the FDR for this case can be directly expressed through the flexibility coefficients related to the type of force unknowns.

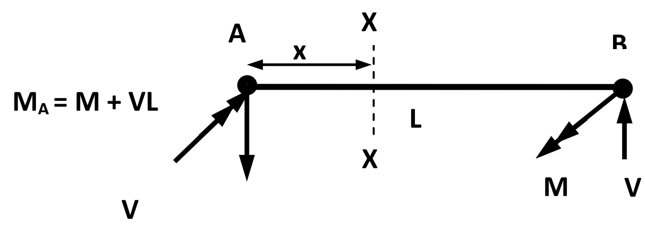


Fig. 3.9 Beam Subjected to Moment and Shear at B end

Following the same procedure, the matrix form for deformation in terms of flexibility coefficients can be written as

$$\begin{Bmatrix} \beta_A \\ \beta_B \end{Bmatrix} = \frac{1}{EI}\begin{bmatrix} L & \frac{L^2}{2} \\ \frac{L^2}{2} & \frac{L^3}{3} \end{bmatrix}\begin{Bmatrix} M \\ V \end{Bmatrix} \qquad \ldots (3.25)$$

$\{\beta\}$ = [G] {F}

where [G] is the flexibility matrix associated with M and V for the beam.

3.5 NULL PROPERTY OF EEs AND CCs

Once the equilibrium between external loading and internal forces or external reactions is satisfied, a rectangular matrix [B] of size m x n is developed with m being the number of equilibrium equations considered along displacement directions and n being the total number of unknowns considered. Thus, r = n – m compatibility conditions are needed. After developing the equilibrium matrix [B], displacement deformation matrix is written as follows.

$$\{\beta\}_{(n\,x\,1)} = [B]^T{}_{(n\,x\,m)}\{\delta\}_{(m\,x\,1)} \qquad \ldots (3.26)$$

Once $\{\beta\}$ is developed, in terms of nodal displacements (θ, δ), one has to develop r number of compatibility conditions by eliminating the nodal displacements from the equations.

Let r = 2, then the problem is 2^0 degree of statically indeterminate. Converting the compatibility conditions in a matrix form gives

$$[C]_{(r\,x\,n)}\{\beta\}_{(n\,x\,1)} = \{0\} \qquad \ldots (3.27)$$

Substituting $\{\beta\}$ from Eq.(3.26) gives,

$$[C]_{(r\,x\,n)}[B]^T{}_{(n\,x\,m)}\{\delta\}_{(m\,x\,1)} = \{0\} \qquad \ldots (3.28)$$

As displacement vector $\{\delta\} \neq \{0\}$, its coefficients matrix $[C]_{(r\,x\,n)}[B]^T{}_{(n\,x\,m)}$ must be zero. Thus,

$$[C]_{(r\,x\,n)}[B]^T{}_{(n\,x\,m)} = \{0\} \qquad \ldots (3.29)$$

or $$[B]_{(m\,x\,n)}[C]^T{}_{(n\,x\,r)} = \{0\} \qquad \ldots (3.30)$$

Which is a must for the correctness of the solution for all the problems and it also confirms that the developed equilibrium matrix [B] and the compatibility matrix [C] are correct.

3.6 IFM SOLUTION PROCEDURE

The complete procedure is explained here with the help of a fixed beam example shown in **Fig. 3.4.**

Step 1: Develop Equilibrium Matrix [B]

By referring **Fig. 3.4** and Eq. (3.15), one can write

$$\begin{bmatrix} \frac{2}{L} & -\frac{2}{L} & -\frac{2}{L} & \frac{2}{L} \\ 0 & 1 & -1 & 0 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} P \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

or $$[B]\{F\} = \{P\} \qquad \ldots (3.31)$$

Step 2: Develop Displacement Deformation Relation $\{\beta\}$

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = \begin{bmatrix} \frac{2}{L} & 0 \\ -\frac{2}{L} & 1 \\ -\frac{2}{L} & -1 \\ \frac{2}{L} & 0 \end{bmatrix} \begin{Bmatrix} \delta \\ \Theta \end{Bmatrix} \quad \ldots (3.32)$$

Which can be written as

$$\{\beta\} = [B]^T\{d\}$$

which can be written in expanded form as follows:

$$\beta_1 = \frac{2}{L}\delta\,,\quad \beta_2 = -\frac{2}{L}\delta + \Theta,\quad \beta_3 = -\frac{2}{L}\delta - \Theta \text{ and } \beta_4 = \frac{2}{L}\delta \quad \ldots (3.33)$$

Step 3: Develop Compatibility Matrix [C]

Let nodal displacements δ and θ, just below the central point load P. To make [B] matrix square, one needs to develop two additional conditions using displacement deformation relations $\{\beta\}$, as r = n - m = 4 – 2 = 2.

So, $\beta_1 + \beta_2 + \beta_3 + \beta_4 = 0$ and $\beta_1 - \beta_4 = 0$

Arranging in matrix form, one can write

$$\begin{bmatrix} 1 & 1 & 1 & 1 \\ 1 & 0 & 0 & -1 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \quad \ldots (3.34)$$

or $[C]\{\beta\} = \{0\}$, where [C] is known as the compatibility matrix which is developed by eliminating the two nodal displacements (δ, Θ) from the displacement deformation relation. Developing coefficients for all the {β} is always a mathematical hurdle in the further development.

Using Eq. (3.17), the null property of all the matrices is checked, which validate the above formulation

$$[C][B]^T = \begin{bmatrix} 1 & 1 & 1 & 1 \\ 1 & 0 & 0 & -1 \end{bmatrix} \begin{bmatrix} \frac{2}{L} & 0 \\ -\frac{2}{L} & 1 \\ -\frac{2}{L} & -1 \\ \frac{2}{L} & 0 \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix} \quad \ldots (3.35)$$

Step 4: Develop Force Deformation Relation

As the problem is having four internal moments as unknowns, considering relative flexibility coefficients and using Eq. (3.4), one can write for AC and CB segments of fixed beam,

$$\begin{Bmatrix} \beta_A \\ \beta_C \end{Bmatrix} = \frac{L}{12EI} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \end{Bmatrix}$$

$$\begin{Bmatrix} \beta_C \\ \beta_B \end{Bmatrix} = \frac{L}{12EI} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{Bmatrix} M_3 \\ M_4 \end{Bmatrix} \quad \ldots (3.36)$$

Substituting Eq. (3.24) into Eq. (3.22), all the DDRs can be converted to FDRs (Force deformation relations).

Thus, one can write

$$\frac{L}{12EI}[\,3M_1 + 3M_2 + 3M_3 + 3M_4] = 0$$

$$\frac{L}{12EI}[\,2M_1 + M_2 - M_3 + 2M_4] = 0$$

Arranging in matrix form,

$$\frac{L}{12EI} \begin{bmatrix} 3 & 3 & 3 & 3 \\ 2 & 1 & -1 & 2 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \quad \ldots (3.37)$$

Step 5: Develop Global Equilibrium Matrix

Concatenating the Eq. (3.25) by substituting related values of parameters in Eq. (3.19), one can have the global equilibrium matrix as

$$\begin{bmatrix} 2 & -2 & 2 & 2 \\ 0 & 1 & -1 & 0 \\ \frac{1}{4} & \frac{1}{4} & \frac{1}{4} & \frac{1}{4} \\ \frac{1}{6} & \frac{1}{12} & -\frac{1}{12} & \frac{1}{6} \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 1 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

Which can be written in the form

$$[S]\{F\} = \{P\} \qquad \text{... (3.38)}$$

Solving the above matrix gives the value of internal moments as

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 0.125 \\ -0.125 \\ -0.125 \\ 0.125 \end{Bmatrix} \text{kN-m} \qquad \text{... (3.39)}$$

Which are matching with the exact solution of moments in a fixed beam at supports and at a mid span under a central point load.

Step 6: Calculate Nodal Displacements

The equation for nodal displacements is given by

$$\{\delta\} = [J][G]\{M\}$$

where [J] = m rows from top of $[[S]^{-1}]^T$ matrix and is of size 2 x 4, [G] consists of flexibility coefficients calculated from Eq. (3.24) of $\{\beta\}$ matrix as

$$\begin{bmatrix} \frac{1}{6} & \frac{1}{12} & 0 & 0 \\ \frac{1}{12} & \frac{1}{6} & 0 & 0 \\ 0 & 0 & \frac{1}{6} & \frac{1}{12} \\ 0 & 0 & \frac{1}{12} & \frac{1}{6} \end{bmatrix}$$

The solution obtained from Eq. (3.39) is as follows:

w = 0.0052mm and θ = 0.0 radians, which is matching with the exact solution of deflection of a fixed beam under the central point load as $\frac{PL^3}{192EI}$ = 0.005208mm.

3.7 IFM FOR FRAMED STRUCTURES

In the IFM any structure is designated as "structure (n, m)" where n and m are the force and displacement degrees of freedom respectively.[1] The n component of force vector {F} must satisfy m equilibrium equations along displacement directions, with r = (n - m) being the number of compatibility conditions. For framed structures, the equilibrium equations can be symbolized as

$$[B]\{F\} = \{P\} \quad \ldots (3.40)$$

The EEs represent a relation between the internal forces {F} and external loads {P}. The internal forces {F} are the prime variables of equilibrium. The equilibrium matrix [B] is always rectangular for statically indeterminate structures, where n > m, and is a sparse matrix for large scale problems. The development of [B] matrix is very simple and straight forward.

The work done by external loading {P} of the structure by considering the nodal displacements {X} of the structure can be written as

$$W = \frac{1}{2}\{P\}^T\{X\} = \frac{1}{2}[P_1X_1 + P_2X_2 + P_3X_3 + \ldots\ldots\ldots P_mX_m] \quad \ldots (3.41)$$

The internal strain energy of the structure can also be written considering the deformation of the elements as follows.

$$IE = \frac{1}{2}\{F\}^T\{\beta\} = = \frac{1}{2}[F_1\beta_1 + F_2\beta_2 + F_3\beta_3 + \ldots\ldots\ldots F_n\beta_n] \quad \ldots (3.42)$$

In which, {β} represents the vector of generalized internal deformations of the elements developed due to the straining.

According to the energy conservation theorem, the internal energy (IE) stored in the structure is equal to the work done (W) by the external loads {P}.

$$IE = W \quad \text{or} \quad \frac{1}{2}\{F\}^T\{\beta\} = \frac{1}{2}\{P\}^T\{\delta\} \quad \ldots (3.43)$$

Substituting the value of {P} from Eq. (3.40) in Eq. (3.43), one can eliminate $\{F\}^T$ from the above equations. Thus, the equation becomes

$$\{\beta\} = \{B\}^T\{\delta\} \qquad \ldots (3.44)$$

The above deformation displacement relation represents n deformations expressed in terms of m displacements, which leads to (n – m) constraints on the deformations of the elements. The constraint on deformations are called compatibility conditions and are expressed through a compatibility matrix [C] and a generalized internal deformation vector {β} which can be written as

$$[C]\{\beta\} = 0 \qquad \ldots (3.45)$$

Now as per IFM these CCs are required to be augmented along with the system equilibrium equations that are already in terms of primal variables {F} i.e. internal forces. Therefore, it is required to express this compatibility matrix in terms of primal variables {F}. Noting that {β} = [G]{F}, one can write

$$[C]\{\beta\} = [C][G]\{F\} \qquad \ldots (3.46)$$

where [G] is concatenated flexibility matrix.

Combining Eqs. (3.40) and (3.46), the coupled equations of IFM can be written as

$$\left[\frac{[B]}{[C][G]}\right]\{F\} = \left\{\frac{\{P\}}{\{\partial R\}}\right\} \quad \text{or} \quad [S]\,\{F\} = \{P\} \qquad \ldots (3.47)$$

Equation (3.47) is known as the basic equation of IFM. In which, [S] is the global equilibrium equation matrix of size (n x n) which consists of two components The upper part [S] is known as [B] matrix, which is a sparse matrix of size (m x n) and is developed through basic equilibrium equations. Bottom part [C][G] is known as the compatibility matrix which is of size (r x n), where r = n – m. {F} is unknown vector of internal force of size (n x 1), which depends upon the type of problem. {P} is the vector of external loading of size (n x 1), in which {∂R} is the vector related to the secondary effect to be concatenated with respect to nodal displacement in {P} matrix. The solution of Eq. (3.47) can be obtained by inverting the [S] matrix. However, before inverting normalization of major elements with respect to upper components has to be carried out. This is required, because [C][G] always gives components that are of much less value compared to the components of [B] matrix.

The nodal displacements {δ} can be worked out using the following relation:

$$\{\delta\} = [J][G]\{F\} \qquad \dots (3.48)$$

where [J] = m rows from top of $[[S]^{-1}]^{T}$ of size (m x n), [G] = flexibility coefficients calculated from Eqs. (3.24) or (3.25) and {F} is the vector calculated from using Eq. (3.47).

3.8 IFM FOR CONTINUUM STRUCTURES

In reality, a physical system has three dimensional domain. Practical situation, however, may have geometry and loading condition such that a three dimensional problem may be idealized as one or two dimensional problem. Two dimensional simplifications implies that one may disregard one of the coordinate axes in these problems, for instance z axis and consider that the whole phenomena takes place in xy plane. Four common situation of 2D simplification are; (i) Plane strain problems, (ii) Plane stress problems (iii) Plate bending problems and (iv) Axisymmetric problems.

Problems involving long body whose geometry and loading do not vary significantly in the longitudinal direction are referred to as plane strain problems (**Fig. 3.10).** Examples of this type are long strip footing, retaining wall, dam and long underground tunnel. In a plane strain problem, the strain normal to the plane and loading is assumed to be zero. Thus, only in-plane strains ϵ_x, ϵ_y and γ_{xy} are nonzero whereas ϵ_z , ν_{yz} and γ_{zx} are zero. In contrast to the plane strain condition, in which longitudinal dimension in the z-direction is large compared to the z and y dimensions, the plane stress condition is characterized by very small dimension in the z-direction.

A thin plate loaded in its own plane is the well known example of plane stress approximation. In a plane stress situation, in-plane stresses i.e. σ_x, σ_y and τ_{xy} are non-zero and other stresses are zero. In plate bending problem, however, a thin plate is subjected to lateral load instead of inplane loading and in such cases moment-curvature relationship is considered for the analysis for the analysis of plate instead of stress-strain relationship. When the geometry, boundary condition, material properties and loading are identical with respect to axis of symmetry, the three dimensional problem can be reduced to an analogous two dimensional problem which is characterized as an axi-symmetric problem **(Fig. 3.10).** Typical examples, where the axisymmetric analysis may be sufficient are pressure vessels, water tanks etc.

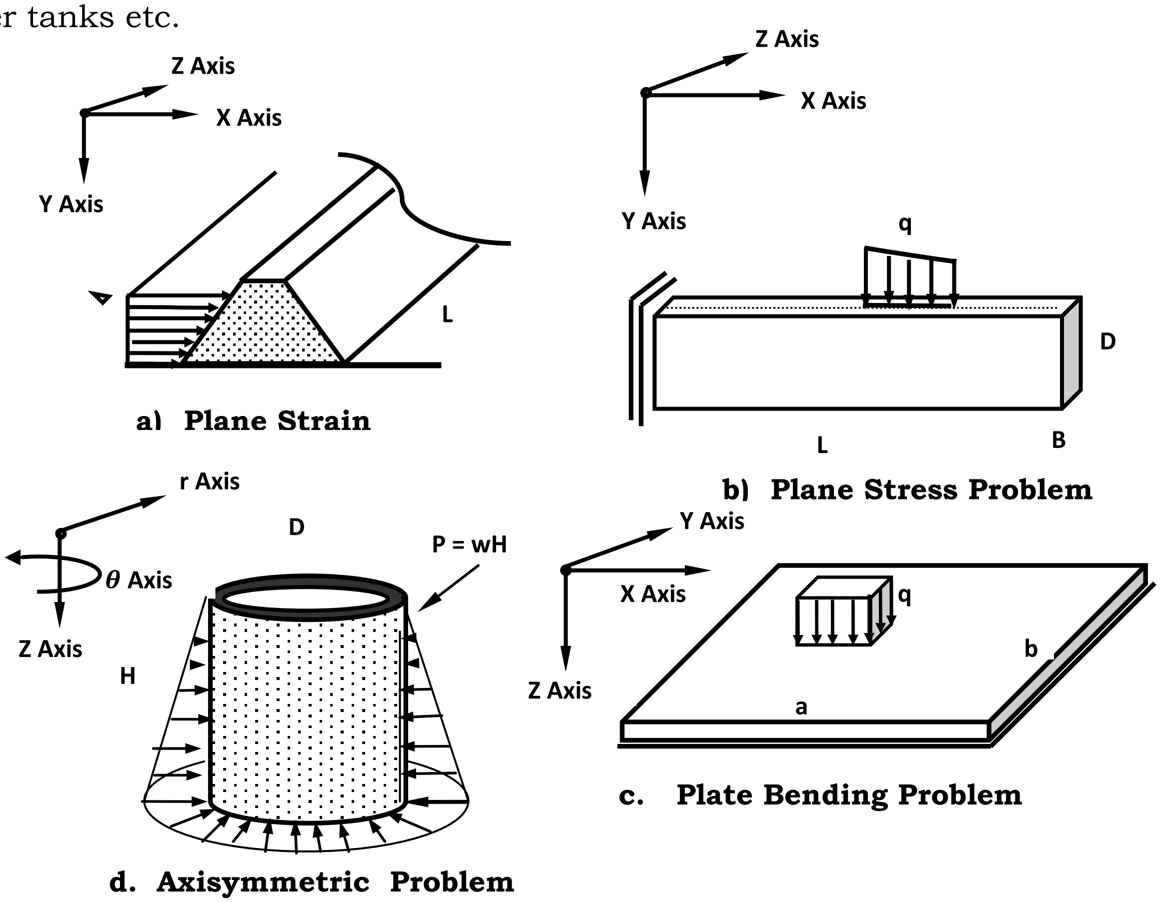


Fig. 3.10 Different Types of 2D Problems

The basic equations of IFM for solving 2D continuum type of problems remain the same. Here also primarily internal forces {F} are worked out first and then utilizing the secondary equations the nodal displacements are calculated. The complete formulation consists of derivation of different types of element properties based on the discretization of different strain energies.

3.9 FORMULATION FOR PLANE STRESS/STRAIN PROBLEMS

The continuum problem is discretized into finite number of elements with 'n' and 'm' force and displacement degrees of freedom respectively. The governing equations are obtained by coupling the m equilibrium equations and r = n – m compatibility conditions. The equilibrium equations are expressed as

$$[B]\{F\} = \{P\}$$

with 'r' compatibility conditions as

$$([C][G]\{F\}= \{\delta R\})$$

The basic governing IFM equation for analysis is expressed as

$$\left[\frac{[B]}{[C][G]}\right]\{F\} = \left\{\frac{\{P\}}{\{\partial R\}}\right\} \quad \text{or } [S]\ \{F\}=\{P\}$$

From forces {F}, the nodal displacements can be calculated using the following formula;

$$\{\delta\} = [J]\ \{[G]\ \{F\} + \{\beta^0\}\} \qquad \text{… (3.49)}$$

where [J] = m top rows of matrix $[[S]^{-1}]^T$.

The complete formulation requires the following 3 matrices:

1. The equilibrium matrix [B], which works as a link between internal forces and external loads.
2. The compatibility matrix [C], which governs the deformations by [C]{β}.
3. The global flexibility matrix [G], which relates deformations to forces.

3.9.1 Formulation of Equlibrium Matrix [B]

The 'EE' written in terms of forces at grid points of finite-element model represents the vectorial summation of 'n' internal forces {F} and 'm' external loads {P}. The nodal EE in matrix notation gives a banded rectangular matrix [B] of size m x n. The variational functional is evaluated as a portion of IFM functional which yields the basic elemental equilibrium matrix [B^e] in explicit form.

$$U^e = \int_S \left\{N_x \left(\frac{\partial u}{\partial x}\right) + N_y \left(\frac{\partial v}{\partial y}\right) + N_{xy} \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right\} dxdy = \int [N]^T\{\epsilon\}\, ds \qquad \ldots (3.50)$$

Where Nx, Ny, Nxy are the in-plane internal forces and $\epsilon_x = \frac{\partial u}{\partial x}$, $\epsilon_y = \frac{\partial v}{\partial y}$ *and* $v_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}$ represents the respective strains.

Consider four-noded, 8 ddof rectangular in-plane element of thickness t with dimensions 2a x 2b along x - and y- axes as shown in **Fig. 3.11.**

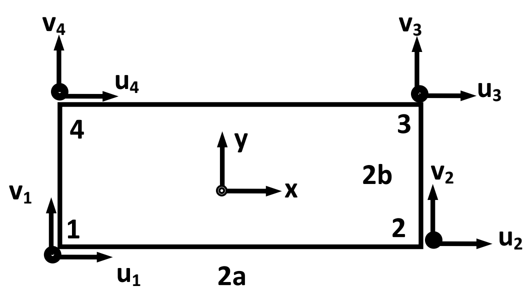


Fig. 3.11 Rectangular Element with Nodal Displacements

For the rectangular membrane element, the force field is chosen in terms of five independent forces as;

$$[F]^T = \{ F_1\ F_2\ F_3\ F_4\ F_5\}^T \qquad \ldots (3.51)$$

Here the distribution of internal forces in terms of unknowns is considered as follows:

$$N_x = F_1 + F_2\frac{y}{b}, \qquad N_y = F_3 + F_4\frac{x}{a} \text{ and } N_{xy} = F_5 \qquad \dots (3.52)$$

Although the variation of normal forces is linear, the shear is constant. The displacement field should satisfy the continuity condition and the selected forces should satisfy the mandatory requirement. Displacement interpolation functions for generalized element are as follows:

$$u_{(x,\,y)} = \frac{1}{4}\begin{Bmatrix} \left(1-\frac{x}{a}\right)\left(1-\frac{y}{b}\right)u_1 + \left(1+\frac{x}{a}\right)\left(1-\frac{y}{b}\right)u_2 \\ +\left(1+\frac{x}{a}\right)\left(1+\frac{y}{b}\right)u_3 + \left(1-\frac{x}{a}\right)\left(1+\frac{y}{b}\right)u_4 \end{Bmatrix} \qquad \dots (3.53)$$

$$v_{(x,\,y)} = \frac{1}{4}\begin{Bmatrix} \left(1-\frac{x}{a}\right)\left(1-\frac{y}{b}\right)v_1 + \left(1+\frac{x}{a}\right)\left(1-\frac{y}{b}\right)v_2 \\ +\left(1+\frac{x}{a}\right)\left(1+\frac{y}{b}\right)v_3 + \left(1-\frac{x}{a}\right)\left(1+\frac{y}{b}\right)v_4 \end{Bmatrix} \qquad \dots (3.54)$$

Where u_1, v_1 ----- u_4, v_4 are eight nodal displacements as shown in **Fig. 3.11.** Substituting Eqs. (3.53) and (3.54) in the Eq. (3.50) and rearranging all force and displacement functions properly, one can obtain the elemental equilibrium matrix as follows

$$U^e = \{\delta\}^T[B^e]\{F\}$$

$$\text{where } [B^e] = \int_s [Z]^T[Y]\, ds \qquad \dots (3.55)$$

Here [Z] = [L][N], [L] is the differential operator matrix, [N] is the displacement interpolation function matrix and [Y] is the force interpolation function matrix. Substituting and integrating yields the following non-symmetrical equilibrium matrix $[B^e]$ for the element, which represents the displacements of m rows from u_1 to v_4 in increasing order from top to bottom and n columns represents forces from F_1 to F_5 in increasing order from left to right.

$$[B^e] = \begin{bmatrix} -b & \frac{b}{3} & 0 & 0 & -a \\ 0 & 0 & -a & \frac{a}{3} & -b \\ b & \frac{b}{3} & 0 & 0 & -a \\ 0 & 0 & -a & -\frac{a}{3} & b \\ b & \frac{b}{3} & 0 & 0 & a \\ 0 & 0 & a & \frac{a}{3} & b \\ -b & -\frac{b}{3} & 0 & 0 & a \\ 0 & 0 & a & -\frac{a}{3} & -b \end{bmatrix} \quad \ldots (3.56)$$

3.9.2 Element Flexibility Matrix [G^e]

The basic elemental flexibility matrix is obtained by discretizing the complementary strain energy which gives

$$[G^e] = \int_s [Y]^T [D][Y]\, dxdy \quad \ldots (3.57)$$

where [Y] is the force interpolation function matrix, which is developed from Eq.(3.40) and [D] is material property matrix. Substituting values in Eq. (3.44) and integrating within the domain (2a x 2b) with the origin at the center of the element yields the symmetrical flexibility matrix [G^e] as follows;

$$[G^e] = \frac{4ab}{Et} \begin{bmatrix} 1 & 0 & -\nu & 0 & 0 \\ 0 & \frac{1}{3} & 0 & 0 & 0 \\ -\nu & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{3} & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix} \quad \ldots (3.58)$$

3.9.3 Global Compatibility Matrix

The compatibility matrix is obtained from the deformation displacement relation ($\{\beta\} = [B]^T\{X\}$). In DDR all the deformations are expressed in terms of all the possible nodal displacements and the 'r' compatibility conditions are developed in terms of internal forces F_1,------ F_{2n}, where '2n' indicates the total number of internal forces in a given problem. So, global compatibility matrix [C] can be evaluated by multiplying the global coefficients of {β} of complete matrix (r x n) under consideration by global flexibility matrix, which is developed by putting all elemental flexibility matrix at diagonal position as per the numbering pattern of each element. One can check the null property of the matrix as per Eq. (3.30) for its mathematical validity.

3.10 FORMULATION FOR PLATE BENDING PROBLEMS

The procedure discussed above remains the same except the change in formulation of element matrices required for the solution of bending problem.

3.10.1 Formulation of Equlibrium Matrix [B^e]

The 'EE' written in terms of forces at grid points of discrete model represents the vectorial summation of 'n' internal forces {F} and 'm' external loads {P}. The nodal EE gives banded rectangular matrix [B] of size m x n. The variational functional is evaluated as a portion of IFM functional which yields the basic elemental equilibrium matrix [B^e] in explicit form.

$$U^e = \int \left\{ M_x \left(\frac{\partial^2 w}{\partial x^2}\right) + M_y \left(\frac{\partial^2 w}{\partial y^2}\right) + 2M_{xy}\left(\frac{\partial^2 w}{\partial x\,\partial y}\right)\right\} dxdy$$

$$= \int [M]^T\{\epsilon\}\, dxdy \qquad \dots (3.59)$$

Where M_x, M_y and M_{xy} are the internal moments and $\{\epsilon\}^T = \left(\frac{\partial^2 w}{\partial x^2}, \frac{\partial^2 w}{\partial y^2} \text{ and } 2\frac{\partial^2 w}{\partial x\,\partial y}\right)$ represent the corresponding curvature terms.

Consider a four-noded, 12 ddof (w_1 to θ_{y4} with three degrees of freedom at each node) rectangular element of thickness t with dimensions as 2a x 2b along the x and y axes. The force field is chosen in terms of nine independent forces as;

$$\{F\} = [F_1, F_2 \ldots\ldots F_9]^T \quad \ldots (3.60)$$

Relations between internal moments and independent forces are written as

$$M_x = F_1 + F_2x + F_3y + F_4xy$$

$$M_y = F_5 + F_6x + F_7y + F_8xy$$

$$M_{xy} = F_9$$

Arranging above equations in matrix form,

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & xy & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & x & y & xy & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \{F_e\} \quad \ldots (3.61)$$

or $\{M\} = [Y]\{F_e\}$, where $\{F_e\} = [F_1, F_2, F_3 \ldots\ldots\ldots\ldots F_9]^T$

The variation of above forces is considered bilinear along both the directions. The displacement fields satisfy the continuity condition and the selected forces satisfy the mandatory requirement.

The Hermitian Interpolation function for the lateral displacement for rectangular plate bending element is as follows:

$$w(x,y) = N_1(x,y)w_1 + N_1(x',y)\Theta_{x1} + N_1(x,y')\Theta_{y1} \ldots\ldots\ldots + N_4(x,y')\Theta_{y4} \quad \ldots (3.62)$$

Where, $N_1(x,y) = N_{1(x)}N_{1(y)}$, $N_1(x',y) = N_{1'(x)}N_{1(y)}$ and $N_1(x,y') = N_{1(x)}N_{1'(y)}$ and so on. Also,

$$N_{1(x)} = \frac{x^3 - 3a^2x + 2a^3}{4a^3}, \quad N_{2(y)} = \frac{y^3 + 3b^2y + 2b^3}{4b^3}, \quad N_1'(x) = \frac{x^3 - ax^2 - a^2x + a^3}{4a^2}$$

$N_1'(y) = \dfrac{y^3 - by^2 - b^2y + b^3}{4b^2}$, $N_2'(y) = \dfrac{y^3 + by^2 - b^2y - b^3}{4b^2}$ .... etc are associated with the nodal displacements w_1, Θ_{x1} -----------Θ_{y4} as shown in **Fig. 3.11**

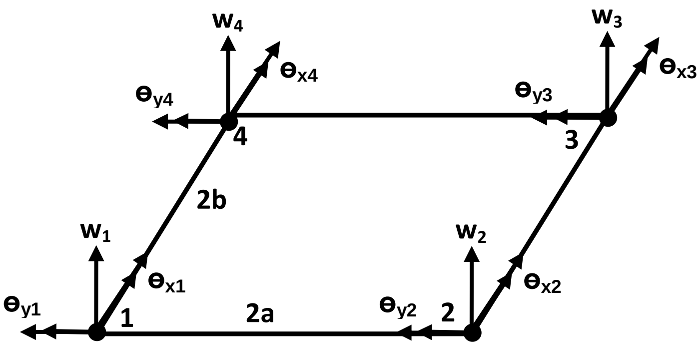


Fig. 3.11 Nodal Displacements

By arranging all force and displacement functions properly, one can discretize the Eq. (3.46) to obtain the elemental equilibrium matrix as follows.

$$U^e = \{\delta\}^T[B^e]\{F\}$$

Where $[B^e] = \int_s [Z]^T[Y]\, ds$ and $[Z] = [L][N]$

where [L] is the differential operator matrix, [N] is the displacement interpolation function matrix and [Y] is the matrix of force interpolation function. Substituting and integrating yields the following equilibrium matrix $[B^e]$ of size 12 x 9. Here the row from top to bottom represents the displacement components (X_1 to X_{12}), while column from left to right represents the independent unknowns (F_1 to F_9).

$$[B^e] = \begin{bmatrix} 0 & b & 0 & \frac{-2b^2}{5} & 0 & 0 & a & \frac{-2a^2}{5} & -2 \\ 0 & \frac{b^2}{3} & 0 & \frac{-b^3}{15} & -a & \frac{2a^2}{5} & ab & \frac{-2a^2b}{5} & 0 \\ b & -ab & \frac{-2b^2}{5} & \frac{-2a^2b}{5} & 0 & 0 & \frac{-a^2}{3} & \frac{a^3}{15} & 0 \\ 0 & b & 0 & \frac{-2b^2}{5} & 0 & 0 & -a & \frac{-2a^2}{5} & 0 \\ 0 & -\frac{b^2}{3} & 0 & \frac{-b^3}{15} & a & \frac{-2a^2}{5} & ab & \frac{-2a^2b}{5} & 0 \\ -b & -ab & \frac{-2b^2}{5} & \frac{-2a^2b}{5} & 0 & 0 & \frac{a^2}{3} & \frac{a^3}{15} & 0 \\ 0 & -b & 0 & \frac{-2b^2}{5} & 0 & 0 & -a & \frac{-2a^2}{5} & -2 \\ 0 & \frac{b^2}{3} & 0 & \frac{b^2}{5} & a & \frac{2a^2}{5} & ab & \frac{-2a^2b}{5} & 0 \\ -b & -ab & \frac{-2b^2}{5} & \frac{-2a^2b}{5} & 0 & 0 & \frac{-a^2}{3} & -\frac{a^3}{15} & 0 \\ 0 & -b & 0 & \frac{-2b^2}{5} & 0 & 0 & 0 & \frac{2a^2}{5} & 2 \\ 0 & -\frac{b^2}{3} & 0 & \frac{-b^3}{15} & -a & \frac{-2a^2}{5} & ab & \frac{-2a^2b}{5} & 0 \\ 0 & -ab & \frac{-2b^2}{5} & \frac{-2a^2b}{5} & 0 & 0 & \frac{a^2}{3} & \frac{a^3}{15} & 0 \end{bmatrix} \quad \ldots (3.63)$$

3.10.2 Formulation of Element Flexibility Matrix [G^e]

The element flexibility matrix is obtained by discretizing the complementary strain energy; which gives

$$[G^e] = \int_s [Y]^T [D][Y] \, dxdy$$

where [Y] is the force interpolation function matrix and [D] is the material property matrix. Integrating within the domain (2a x 2b), with origin at center of the element, yields the symmetrical flexibility matrix [G^e] as follows;

$$[G^e] = \frac{48ab}{Et^3}\begin{bmatrix} 1 & 0 & 0 & 0 & -\nu & 0 & 0 & 0 & 0 \\ 0 & \frac{a^2}{3} & 0 & 0 & 0 & -\nu\frac{a^2}{3} & 0 & 0 & 0 \\ 0 & 0 & \frac{b^2}{3} & 0 & 0 & 0 & -\nu\frac{b^2}{3} & 0 & 0 \\ 0 & 0 & 0 & \frac{a^2b^2}{9} & 0 & 0 & 0 & -\nu\frac{a^2b^2}{9} & 0 \\ -\nu & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & -\nu\frac{a^2}{3} & 0 & 0 & 0 & \frac{a^2}{3} & 0 & 0 & 0 \\ 0 & 0 & -\nu\frac{b^2}{3} & 0 & 0 & 0 & \frac{b^2}{3} & 0 & 0 \\ 0 & 0 & 0 & -\nu\frac{a^2b^2}{9} & 0 & 0 & 0 & \frac{a^2b^2}{9} & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix} \quad \ldots (3.64)$$

3.10.3 Global Compatibility Matrix

The compatibility matrix is obtained from the deformation displacement relation ($\{\beta\} = [B]^T\{\delta\}$). In DDR the deformations are expressed in terms of all the possible nodal displacements and the 'r' compatibility conditions are developed in terms of internal forces i.e.,F_1,------ F_{2n}, where '2n' are the total number of internal forces in a given problem. The global compatibility matrix [C] can be evaluated by multiplying the global coefficients of {β} of complete matrix (r x n) by the global flexibility matrix; which is generated by putting all the elemental flexibility matrices at diagonal position as per the numbering of each element. Before using the global compatibility matrix for further calculations, one has to check the null property of the matrix to ensure the mathematical validity of the developed matrices.

CHAPTER 4

DUAL INTEGRATED FORCE METHOD

4.1 GENERAL REMARKS

In the present chapter, formulation for Dual Integrated Force Method (DIFM) is developed from the basic equations of IFM. IFM has two fundamental equations, in which the first one calculates the independent unknowns (F_1, F_2,.....F_n), which represents the internal forces or moments and the other one evaluates the nodal displacements based on the first set of developed equations. Like IFM, DIFM also has two sets of equations, in which first set of equations are considered as primary equations, which are symmetrical and are used to calculate the global nodal displacements, while all the necessary internal forces developed due to external forces in terms of independent unknowns are calculated using the second set of equations.

The DIFM is analogous to the available well known Stiffness Method. The DIFM has two set of equations, one for calculating the nodal displacements and other for evaluating the internal forces, which are based on independent unknowns. The stiffness method for framed structures and the Displacement based FE method for continuum structures on the other hand consist of only one set of equations to calculate the global nodal displacements. Stresses are then calculated by differentiating the displacement function, which can be the source of error. In the present chapter, after giving the basic equations of DIFM, equilibrium ant other matrices are derived for different types of framed structures.

4.2 BASIC THOERY OF DIFM

The complete set of DIFM equations are developed here from the following IFM equations derived in the previous chapter

Equilibrium Equations (EE): $[B]\{F\} = \{P\}$ ... (4.1)

Compatibility Conditions (CC): $[C]\{\beta\} = \{0\}$ ... (4.2)

Force Deformation Relation: $\{\beta\} = [G]\{F\}$ ... (4.3)

Displacement Deformation Relation: $\{\beta\} = [B]^T\{\delta\}$ ... (4.4)

Equating Eqs. (4.3) and (4.4), one can write

$$[G]\{F\} = [B]^T\{\delta\} \quad \text{... (4.5)}$$

Thus, $\{F\} = [G]^{-1}[B]^T\{\delta\}$ ... (4.6)

Multiplying by [B] on both sides of Eq. (4.6), one gets

$$[B]\{F\} = [B][G]^{-1}[B]^T\{\delta\} = \{P\} \quad \text{... (4.7)}$$

Rewriting the above equations with the size of matrices,

$$[[B]_{m \times n} [G]^{-1}_{n \times n} [B]^T_{n \times m}]\{\delta\}_{m \times 1} = \{P\}_{m \times 1} \quad \text{... (4.8)}$$

or $[D]_{difm} \{\delta\} = \{P\}$ ... (4.9)

Where, $[D]_{difm} = [B][G]^{-1}[B]^T$ and $\{P\}_{difm} = \{P\}$ ... (4.10)

Once displacements are known from the Eq. (4.9), the force displacement relation given in Eq. (4.6) is used to calculate the forces. Eq. (4.9) is the primary equation, and the Eq. (4.10) is the secondary equation of DIFM.

In DIFM, the dual matrix $[D]_{difm}$ is assembled from the element matrices in a manner quite similar to the direct stiffness method of assembly. The assembly of the dual matrix $[D]_{difm}$ is carried out by considering individual element equilibrium matrix $[B_e]$ and element flexibility matrix $[G_e]$ which for each element can be worked out as follows:

$$[D]_{difm(e)}^1 = [B_e^1][G_e^1]^{-1}[B_e^1]^T \quad \text{... (4.11)}$$

The procedure to get Dual Matrix $[D]_{difm}$ for the different types of structural elements is explained below.

4.3 FORMULATION FOR AN AXIAL ELEMENT

Consider an axial element of length L and axial rigidity AE, for the derivation of an element Equilibrium Matrix $[B_e]$ and Flexibility Matrix $[G_e]$. It is one of the elements of the total assembly, having three elements with varying geometrical properties as shown in **Fig. 4.1**.

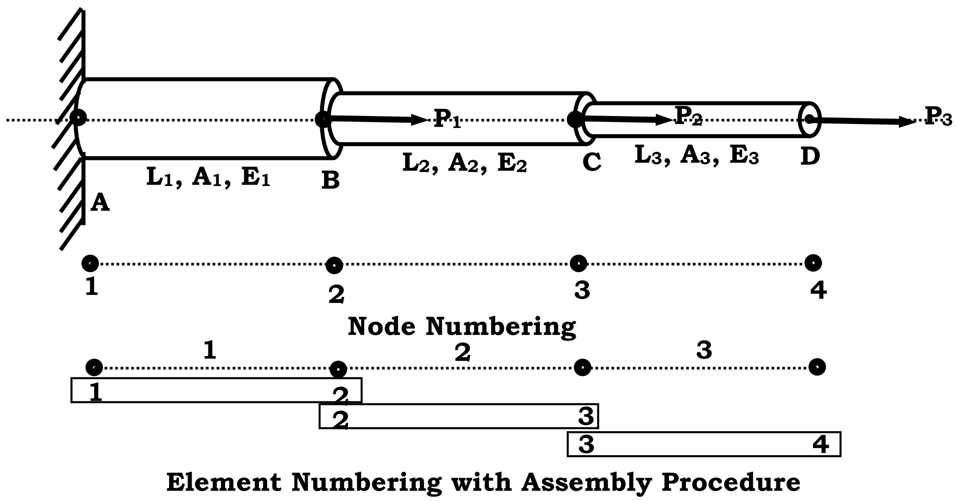


Fig. 4.1 Non-Prismatic Bar Problem

The displacement field for a typical element shown in **Fig. 4.2** is approximated by using the nodal displacements u_1 and u_2 in the given cartesian coordinate system (x, y). The force in the element is {F} associated with an axial stress {σ} for the cross sectional area A.

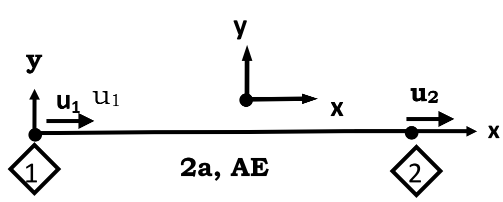


Fig. 4.2 Basic Axial Element

The displacement and stress fields for the element can be written as

$$u = u_1 + \frac{u_2 - u_1}{2a} x \qquad \text{... (4.12)}$$

$$\sigma = \frac{F}{A} \qquad \text{... (4.13)}$$

Eqs. (4.12) and (4.13) can be rewritten using the interpolation function as follows:

$$u = N_1 u_1 + N_2 u_2 \qquad \text{... (4.14)}$$

and $$\sigma = \frac{F}{A} = \left[\frac{1}{A}\right]\{F\} = [Y]\{F\} \qquad \text{... (4.15)}$$

Where $N_1 = \frac{1}{2}(1 - \frac{x}{a})$ and $N_2 = \frac{1}{2}(1 + \frac{x}{a})$ are the displacement shape functions and [Y] is the stress interpolation function. Now the strain- displacement relationship for an axial element can be written as

$$\{\varepsilon\} = \left\{\frac{\partial u}{\partial x}\right\} = [L][N]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{bmatrix} -\frac{1}{2a} & \frac{1}{2a} \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = [Z]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \qquad \text{.... (4.16)}$$

Where $[Z] = [L][N] = [Z]\begin{bmatrix} -\frac{1}{2a} & \frac{1}{2a} \end{bmatrix}$ ... (4.17)

is the strain-displacement linking matrix for an axial element, and [L] = Single order differential operator.

The element equilibrium matrix for an axial element is written as

$$[B_e] = \int_V [Z]^T [Y]\, dV \qquad \text{... (4.18)}$$

Substituting [Z] from Eq. (4.17) and [Y] from Eq. (4.15) in Eq. (4.18) and integrating over the limits (-a, a) gives $[B_e]$ matrix of size (2 x 1) as follows:

$$[B_e] = \begin{bmatrix} -1 \\ 1 \end{bmatrix} \qquad \ldots (4.19)$$

The element flexibility is obtained from

$$[G_e] = \int_{-a}^{a} [Y]^T [D][Y] dV \qquad \ldots (4.20)$$

Considering [D] = 1/E,

$$[G_e] = \int_{-a}^{a} [Y]^T [D][Y]\, dV = \int_{-a}^{a} [A]\left[\frac{1}{E}\right]\left[\frac{1}{A}\right] A dx = \left[\frac{2a}{AE}\right] \qquad \ldots (4.21)$$

4.4 FORMULATION FOR A BEAM ELEMENT

A two span continuous beam is depicted in **Fig. 4.3.** A typical element from the same is shown in **Fig. 4.4.**

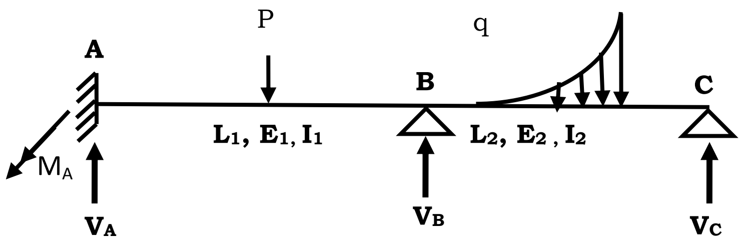


Fig. 4.3 A Continuous Beam Problem

The displacement field for the element AB is written by considering the origin at the center of the element in the cartesian coordinate system.

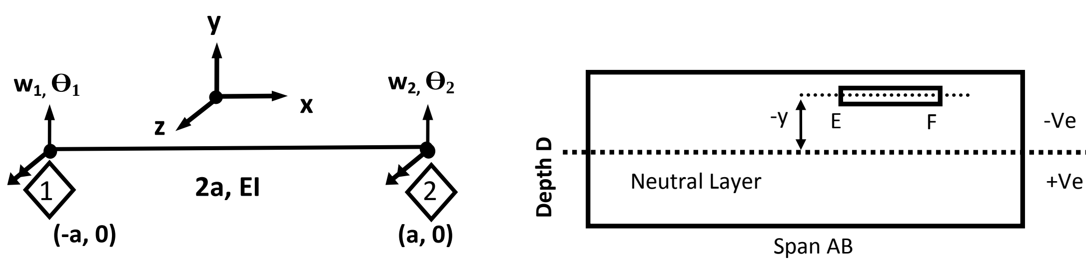


Fig. 4.4 Beam Element

The shape functions, using the generalized coordinate system, corresponding to the four displacement degrees of freedom are as follows

$$N_{1(w)} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3}\,,\quad N_{2(\Theta)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2}\,,$$

$$N_{3(w)} = \frac{1}{2} + \frac{3x}{4a} - \frac{x^3}{4a^3} \text{ and } N_{4(\Theta)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2} \qquad \text{... (4.22)}$$

Now, the deflection w for the beam element can be expressed as

$$w_{(x,y)} = [N]\{\delta\} \qquad \text{... (4.23)}$$

where, [N] = is the shape function matrix of size 1 x 4 and {δ} is the nodal displacement vector corresponding to the nodal displacements at nodes 1 and 2 respectively.

Consider a small layer EF at a distance y from the neutral axis as shown in **Fig. 4.4.** A contraction of EF layer, which causes lateral displacement u of point E or F in horizontal direction can be written as

$$u = -y\tan\Theta = -y\,\frac{dw}{dx} \qquad \text{... (4.24)}$$

So, strain in the layer EF can be calculated from Eq. (4.24) as

$$\varepsilon_x = \frac{\partial u}{\partial x} = -y\,\frac{\partial^2 w}{\partial x^2} \quad = -y[L][N] = -y[Z_1] = [Z] \qquad \text{... (4.25)}$$

Substituting Eq. (4.22) in Eq. (4.25), the strain linking matrix for beam member can written in the form as

$$[Z] = -y\begin{bmatrix} \frac{\partial^2 N_1}{\partial x^2} \\ \frac{\partial^2 N_2}{\partial x^2} \\ \frac{\partial^2 N_3}{\partial x^2} \\ \frac{\partial^2 N_4}{\partial x^2} \end{bmatrix} \qquad \text{... (4.26)}$$

As per the theory of simple bending, the compressive bending stress σ_{bc} can be written in terms of moment and resisting moment of inertia I as

$$\sigma_{bc} = -\frac{My}{I} \qquad \text{... (4.27)}$$

Now the internal moment M can be expressed in terms of independent unknowns {F_1, F_2) as

$$M = F_1 + F_2 \frac{x}{a} \quad \ldots (4.28)$$

Substituting Eq. (4.28) into Eq. (4.27), one can write

$$[M] = [Y]\{F\} = -\frac{y}{I}\left[1 \quad \frac{x}{a}\right]\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \quad \ldots (4.29)$$

The element equilibrium matrix [B_e] can be calculated by using Eq. (4.18) as

$$[B_e] = \int_{-a}^{+a} -y \begin{bmatrix} \frac{\partial^2 N_1}{\partial x^2} \\ \frac{\partial^2 N_2}{\partial x^2} \\ \frac{\partial^2 N_3}{\partial x^2} \\ \frac{\partial^2 N_4}{\partial x^2} \end{bmatrix} \left(-\frac{y}{I}\left[1 \quad \frac{x}{a}\right]\right) dA\, dx \quad \ldots (4.30)$$

Substituting, I = y^2dA and integrating gives the basic equilibrium matrix as

$$[B_e] = \begin{bmatrix} 0 & \frac{1}{a} \\ -1 & 1 \\ 0 & -\frac{1}{a} \\ 1 & 1 \end{bmatrix} \quad \ldots (4.31)$$

The flexibility matrix for the beam element is obtained by substituting [Y] and [D] in Eq. (4.20) as follows;

$$[G_e] = \int_{-a}^{+a} [Y]^T [D][Y]\, dAdx \quad \ldots (4.32)$$

$$= \int_{-a}^{+a} -\frac{y}{I} \begin{bmatrix} 1 \\ \frac{x}{a} \end{bmatrix} \left[\frac{1}{E}\right] \left(-\frac{y}{I}\left[1 \quad \frac{x}{a}\right]\right) dAdx \quad \ldots (4.33)$$

Substituting I = y^2dA, the element flexibility matrix for beam member can be written as

$$[G_e] = \frac{1}{EI}\begin{bmatrix} 2a & 0 \\ 0 & \frac{2a}{3} \end{bmatrix} \quad \ldots (4.34)$$

4.5 FORMULATION FOR A PLANE TRUSS ELEMENT

Consider a pin jointed structure having three members with AE and L as shown in **Fig. 4.5.**

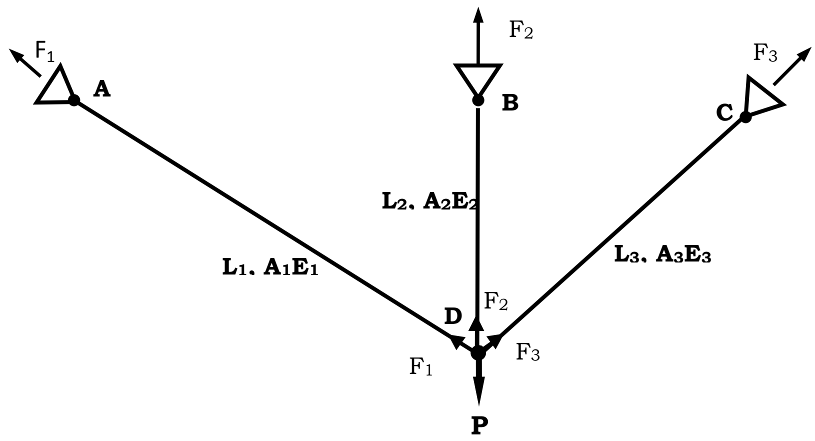


Fig. 4.5 Forces in Three Wire Suspension Problem

The basic matrices are the same as for an axial rod. However, a transformation is required from local to global axis as depicted in **Fig. 4.6.**

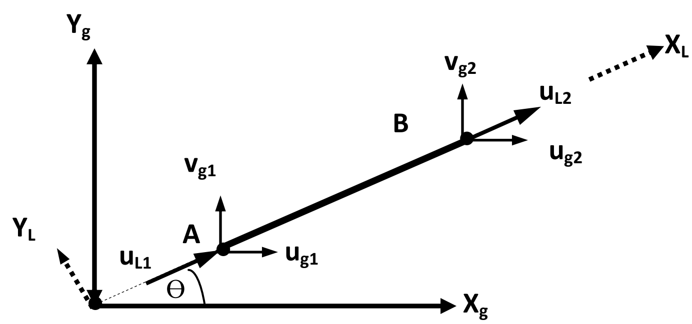


Fig. 4.6 Nodal Displacements in Global and Local Coordinate Systems

If l and m are the direction cosines of the member AB, the local nodal displacements u_{L1} and u_{L2} can be expressed in terms of global nodal displacements u_{g1} to v_{g2} as follows;

$$\begin{Bmatrix} u_{L1} \\ u_{L2} \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix} \begin{Bmatrix} u_{g1} \\ v_{g1} \\ u_{g2} \\ v_{g2} \end{Bmatrix} = [\lambda]\{d_g\} \qquad \dots (4.35)$$

The displacement u in terms of (u_{g1}, v_{g1}, u_{g2}, v_{g2}) can be written by using the relation

$$\{u\} = [N]\begin{Bmatrix} u_{L1} \\ u_{L2} \end{Bmatrix} \quad \dots (4.36)$$

Substituting Eq. (4.35) into Eqn. (4.36)

$$\{u\} = [N][\lambda]\{d_g\} \quad \dots (4.37)$$

Now strain can written as

$$\{\varepsilon\} = \left\{\frac{\partial u}{\partial x}\right\} = [L][N][\lambda]\{d_g\}$$

Considering $[Z] = \{\lambda\}[L][N]$ ... (4.38)

the element equilibrium matrix in global direction will be

$$[B_{e(global)}] = \int [Z]^T \{Y\}\, dV \quad \dots (4.39)$$

Substituting [Z] from Eq. (4.38) into Eq. (4.39), one gets

$$[B_{e(global)}] = [\lambda^T][B_e] \quad \dots (4.40)$$

Substituting equilibrium matrix $[B_e]$ for bar element in local direction in Eq. (4.40) from Eq. (4.19) $[B^e_{(global)}]$ can be calculated as

$$[B^e_{(global)}] = \begin{bmatrix} -l \\ -m \\ l \\ m \end{bmatrix} \quad \dots (4.41)$$

Finally, the flexibility matrix for the element is obtained by the Eq. (4.21).

4.6 FORMULATION FOR A PLANE FRAME ELEMENT

Consider a plane frame example as shown in **Fig. 4.7**.

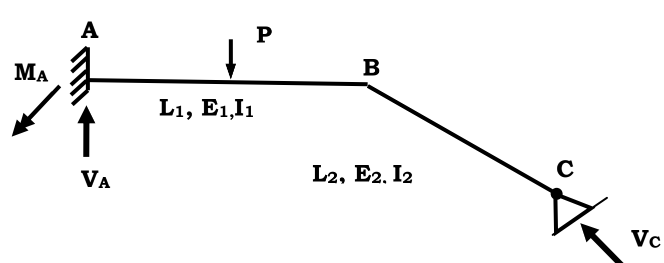


Fig. 4.7 Plane Frame Problem

The displacement field for element AB is approximated by using nodal displacements in the cartesian coordinate system (x, y, z).

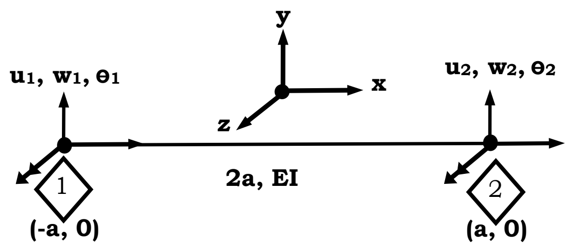


Fig. 4.8 Plane Frame Element

For a plane frame element such as shown in **Fig. 4.8,** the shape functions can be written as combination of previous two cases as follows;

$$N_{1(u)} = \frac{1}{2}\left(1 - \frac{x}{a}\right), N_{2(w)} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3}, N_{3(\Theta)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2},$$

$$N_{4(u)} = \frac{1}{2}\left(1 + \frac{x}{a}\right), N_{5(w)} = \frac{1}{2} + \frac{3x}{4a} - \frac{x^3}{4a^3} \text{ and } N_{6(\Theta)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2} \quad \dots (4.41)$$

Thus, the deflection form for a plane frame element can be written by using the shape functions as

$$w_{(x,y)} = [N]\{\delta\} \quad \dots (4.42)$$

As no interaction between axial and bending effects is assumed, the complete [Z] matrix can be written as

$$[Z] = \begin{bmatrix} \frac{\partial N_{1(u)}}{\partial x} & 0 & 0 & \frac{\partial N_{4(u)}}{\partial x} & 0 & 0 \\ 0 & -y\frac{\partial^2 N_2}{\partial x^2} & -y\frac{\partial^2 N_3}{\partial x^2} & 0 & -y\frac{\partial^2 N_3}{\partial x^2} & -y\frac{\partial^2 N_6}{\partial x^2} \end{bmatrix} \quad \dots (4.43)$$

The stress interpolation function [Y] can also be formed by using appropriate position of corresponding matrix components of the element. Using Eq. (4.15) and Eq. (4.29), it can be written as

$$[Y] = \begin{bmatrix} \frac{1}{A} & 0 & 0 \\ 0 & -\frac{y}{I} & -\frac{y}{I}\left(\frac{x}{a}\right) \end{bmatrix} \quad \dots (4.44)$$

After integration the basic equilibrium matrix [Be] is obtained as

$$[B_e] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & \frac{1}{a} \\ 0 & -1 & 1 \\ -1 & 0 & 0 \\ 0 & 0 & -\frac{1}{a} \\ 0 & 1 & 1 \end{bmatrix} \qquad \ldots (4.45)$$

The flexibility matrix for the beam element is obtained by substituting [Y] from Eq. (4.44) into Eq. (4.20) and substituting [D] = 1/E,

$$[G_e] = \begin{bmatrix} \frac{2a}{AE} & 0 & 0 \\ 0 & \frac{2a}{EI} & 0 \\ 0 & 0 & \frac{2a}{3EI} \end{bmatrix} \qquad \ldots (4.46)$$

4.7 FORMULATION FOR A GRID ELEMENT

A grid structure as shown in **Fig. 4.9** is considered now.

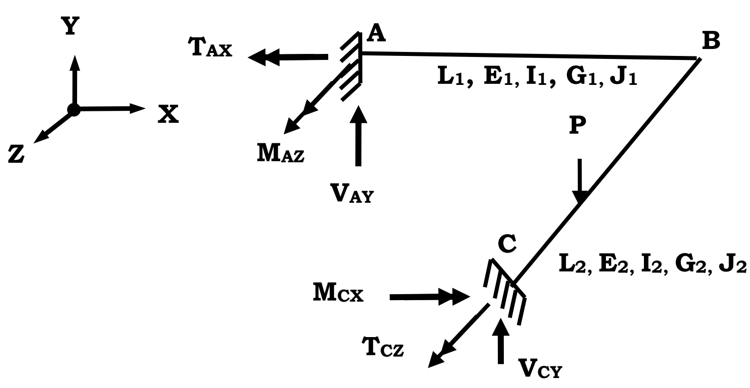


Fig. 4.9 A Grid Structure

The displacement field for a grid member AB (**Fig. 4.10**) is approximated by using the nodal displacements in the cartesian system (x, y, z). The shape functions are the same as last case but an axial component is replaced by torsional one. The shape function for twist is also linearly approximated along the length of member.

$$\Phi = [N_{1(\Phi)} \quad N_{2(\Phi)}] \begin{Bmatrix} \phi_{1(x)} \\ \phi_{2(x)} \end{Bmatrix} \qquad \ldots (4.47)$$

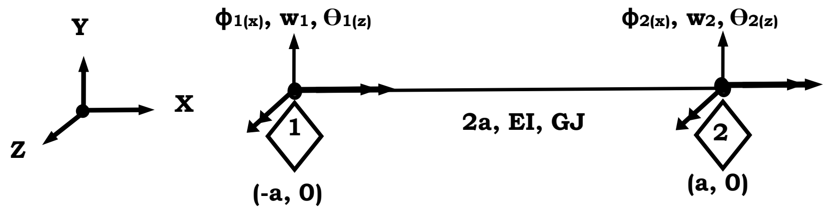


Fig. 4.10 Grid Element

where $\phi_{1(x)}$, and $\phi_{2(x)}$ are the twisting angles at nodes 1 and 2 respectively. The shape functions for a grid element are as follows;

$$N_{1(\Phi)} = \frac{1}{2}\left(1 - \frac{x}{a}\right), N_{2(w)} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3}, N_{3(\Theta)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2},$$

$$N_{4(\Phi)} = \frac{1}{2}\left(1 + \frac{x}{a}\right), N_{5(w)} = \frac{1}{2} + \frac{3x}{4a} - \frac{x^3}{4a^3} \text{ and } N_{6(\Theta)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2} \ldots (4.48)$$

Thus displacement function can be expressed as

$$w_{(x,y)} = [N]\{\delta\} \qquad \ldots (4.49)$$

where [N] is shape function matrix. By separating torsional and bending components, the complete [Z] matrix can be written as

$$[Z] = \begin{bmatrix} r\frac{\partial N_{1(\Phi)}}{\partial x} & 0 & 0 & r\frac{\partial N_{4(\Phi)}}{\partial x} & 0 & 0 \\ 0 & -y\frac{\partial^2 N_2}{\partial x^2} & -y\frac{\partial^2 N_3}{\partial x^2} & 0 & -y\frac{\partial^2 N_3}{\partial x^2} & -y\frac{\partial^2 N_6}{\partial x^2} \end{bmatrix} \qquad \ldots (4.50)$$

The stress interpolation function [Y] also can be formed by using appropriate position corresponding to each component of the matrix. Using Eq. (4.15) and Eq. (4.29), it can be written as

$$[Y] = \begin{bmatrix} \frac{1}{G} & 0 & 0 \\ 0 & -\frac{y}{I} & -\frac{y}{I}\left(\frac{x}{a}\right) \end{bmatrix} \qquad \ldots (4.51)$$

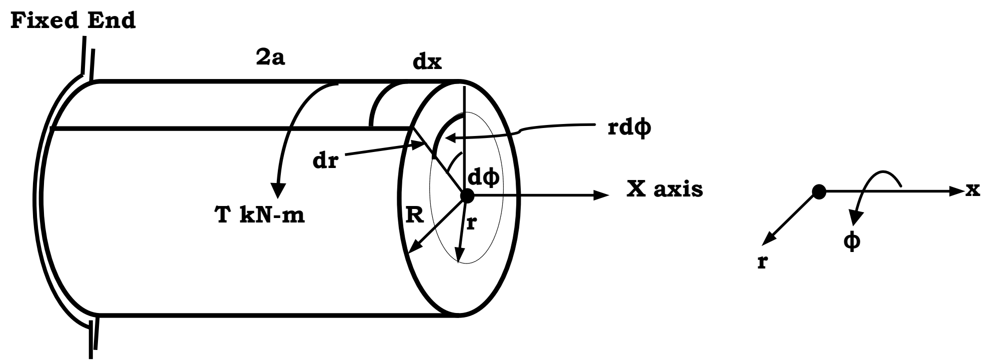


Fig. 4.11 Torsional Deformation of Circular Rod

The element volume of the circular rod **(Fig 4.11)** is $dv = rd\phi \; dr \; dx$ … (4.52)

Where, rdϕ is the circumferential segment at distance r of an angle dϕ, dr is the element length of radius r and dx is the element length along 2a.
Torsional shearing stress (τ) due to torque T can be written in the form

$$\tau = \frac{T\,r}{J} = [Y]\{\overline{T}\} \qquad \dots (4.53)$$

In which [Y] = $\frac{r}{J}$ is considered as stress interpolation function, $\overline{\{T\}}$ = T is the constant torque value along the rod and J is the polar moment of inertia, which can be worked out as

$$J = \int_0^R \int_0^{2\pi} r^3 dr \, d\theta \qquad \dots (4.54)$$

The element equilibrium matrix [Be] can be written by using Eq. (4.18) as

$$[B_e] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & \frac{1}{a} \\ 0 & -1 & 1 \\ -1 & 0 & 0 \\ 0 & 0 & -\frac{1}{a} \\ 0 & 1 & 1 \end{bmatrix} \qquad \dots (4.55)$$

The flexibility matrix [Ge] is obtained by substituting [Y] from Eq. (4.44) and [D] = 1/G; where G is the shear modulus of material.

$$[G_e] = \begin{bmatrix} \frac{2a}{GJ} & 0 & 0 \\ 0 & \frac{2a}{EI} & 0 \\ 0 & 0 & \frac{2a}{3EI} \end{bmatrix} \qquad \ldots (4.56)$$

4.8 FORMULATION FOR A SPACE TRUSS ELEMENT

Consider a pin jointed space truss as shown in **Fig. 4.12**

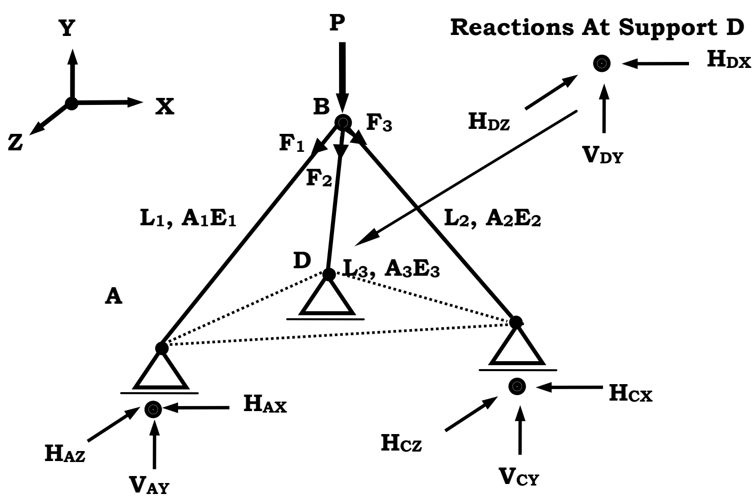


Fig. 4.12 Space Truss Problem

The basic approach for developing various matrices is same as for a plane truss. The transformation from local axes to global axes is carried out referring to **Fig. 4.13.**

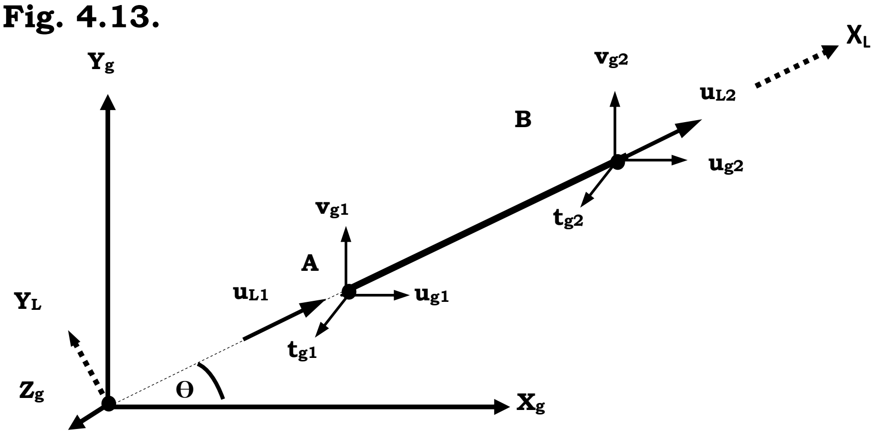


Fig. 4.13 Nodal Displacements in Global and Local Coordinate

As per **Fig. 4.13**, (u_{g1}, v_{g1}, t_{g1}) and (u_{g2}, v_{g2}, t_{g2}) are the global displacements at node A and node B respectively. The bar is oriented at an angle Θ_1, Θ_2 and Θ_3 with respect to global axes (x_g, y_g and z_g). Let l, m and n are the direction cosines of the angles between line AB and global axes respectively. The nodal displacements (u_1, u_2) can be expressed as

$$\begin{Bmatrix} u_{L1} \\ u_{L2} \end{Bmatrix} = \begin{bmatrix} l & m & n & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & n \end{bmatrix} \begin{Bmatrix} u_{g1} \\ v_{g1} \\ t_{g1} \\ u_{g2} \\ v_{g2} \\ t_{g2} \end{Bmatrix} = [\lambda]\{X\} \qquad \ldots (4.57)$$

The displacements can be written as

$$u = [N]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \qquad \ldots (4.58)$$

Substituting nodal displacement vector from Eq. (4.57) into Eq. (4.58)

$$u = [N][\lambda]\{X\}$$

Strain in the truss element is given by

$$\{\varepsilon\} = \left\{\frac{\partial u}{\partial x}\right\} = [L][N][\lambda]\{X\} = [Z]\{X\} \qquad \ldots (4.59)$$

The element equilibrium matrix in global direction is

$$[B_{e(global)}] = \int [Z]^T [Y]\, dV \qquad \ldots (4.60)$$

Substituting [Z] from Eq. (4.59) in Eq. (4.60), one gets

$$[B_{e(global)}] = [\lambda^T][B_e] \qquad \ldots (4.61)$$

Substituting equilibrium matrix [B_e] for bar element in Eq. (4.61) from Eq. (4.18), [$B^e_{(global)}$] can be calculated as

$$[B^e_{(global)}] = \begin{bmatrix} -l \\ -m \\ -n \\ l \\ m \\ n \end{bmatrix} \qquad \ldots (4.62)$$

The flexibility matrix for the bar element is same as given in Eq. (4.21).

4.8 FORMULATION FOR A SPACE FRAME MEMBER

A space frame structure is shown in **Fig. 4.14** with AE, EI, GJ and L of various members.

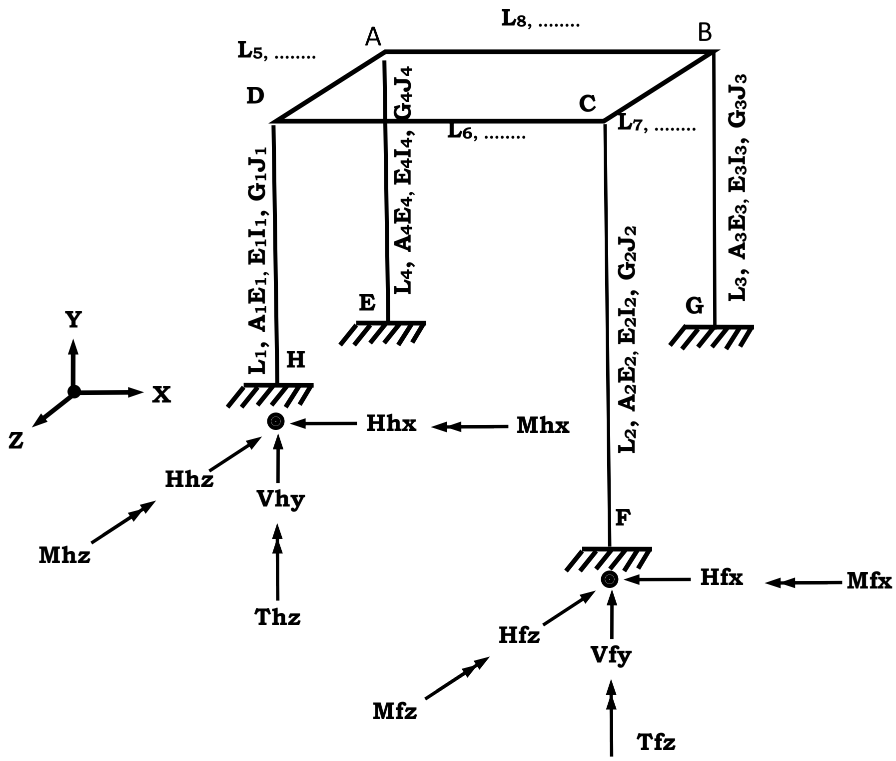


Fig. 4.14 A Typical Space Frame Structure

A typical space frame member has six DOF as shown in **Fig. 4.15.**

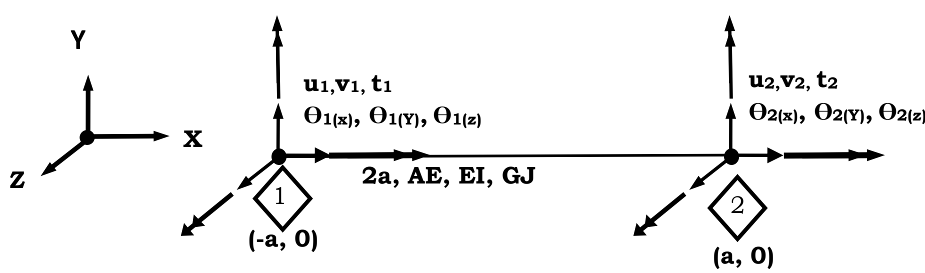


Fig. 4.15 Space Frame Element

The shape functions are written by considering the origin of the coordinates at the centre of the element as follows;

$$N_{1(u)} = \frac{1}{2}\left(1 - \frac{x}{a}\right), \qquad N_{2(wy)} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3}, \qquad N_{3(\boldsymbol{wz})} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3},$$

$$N_{4(\phi)} = \frac{1}{2}\left(1 + \frac{x}{a}\right), \qquad N_{5(\theta\mathbf{y})} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2}, \qquad N_{6(\Theta z)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2}$$

$$N_{7(u)} = \frac{1}{2}\left(1 - \frac{x}{a}\right), \qquad N_{8(wy)} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3}, \qquad N_{9(\boldsymbol{wz})} = \frac{1}{2} - \frac{3x}{4a} + \frac{x^3}{4a^3},$$

$$N_{10(\phi)} = \frac{1}{2}\left(1 + \frac{x}{a}\right), \qquad N_{11(\theta\mathbf{y})} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2}, \qquad N_{12(\Theta z)} = \frac{a}{4} - \frac{x}{4} - \frac{x^2}{4a} + \frac{x^3}{4a^2} \quad \ldots (4.60)$$

The displacement function for a space frame element can be written as

$$w = [N]\{\delta\} \quad \ldots (4.63)$$

By separating axial, torsional and bending components, a complete [Z] matrix can be written in the form as

$$[Z] = \begin{bmatrix} Z_1 & 0 & 0 & 0 & 0 & 0 & Z_7 & 0 & 0 & 0 & 0 & 0 \\ 0 & Z_2 & Z_3 & 0 & 0 & 0 & 0 & Z_8 & Z_9 & 0 & 0 & 0 \\ 0 & 0 & 0 & Z_4 & Z_5 & 0 & 0 & 0 & 0 & Z_{10} & Z_{11} & 0 \\ 0 & 0 & 0 & 0 & 0 & Z_6 & 0 & 0 & 0 & 0 & 0 & Z_{12} \end{bmatrix} \quad \ldots (4.64)$$

The stress interpolation function [Y] matrix can be written as

$$[Y] = \begin{bmatrix} \frac{1}{A} & 0 & 0 & 0 & 0 & 0 \\ 0 & -\frac{y}{I_z} & -\frac{y}{I_z}(\frac{x}{a}) & 0 & 0 & 0 \\ 0 & 0 & 0 & -\frac{z}{I_y} & -\frac{z}{I_y}(\frac{x}{a}) & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G} \end{bmatrix} \quad \ldots (4.65)$$

The element equilibrium matrix [Be] can be written by using Eq. (4.18) as

$$[B_e] = \begin{bmatrix} -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & \frac{1}{a} & 0 & 0 & 0 \\ 0 & -1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{a} & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 \\ 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -\frac{1}{a} & 0 & 0 & 0 \\ 0 & 1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{1}{a} & 0 \\ 0 & 0 & 0 & 1 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \quad \ldots (4.66)$$

The element flexibility matrix for the space frame element is obtained by substituting [Y] from Eq. (4.64) into Eq. (4.20).

$$[G_e] = \begin{bmatrix} \frac{2a}{AE} & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{2a}{EI} & 0 & 0 & 0 & 0 \\ 0 & 0 & \frac{2a}{3EI} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{2a}{AE} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{2a}{3EI} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{2a}{GJ} \end{bmatrix} \quad \ldots (4.67)$$

4.10 SOLUTION STEPS OF DIFM

The solution of a problem by DIFM mainly consists of development of proper Dual Matrix [D_{difm}], which depends upon the element equilibrium [B_e] and flexibility [G_e] matrices. For solving a problem, following steps are required.

1. Generate shape function matrix [N] as per the type of problem.
2. Develop strain linking matrix [Z].
3. Develop force interpolation function matrix [Y] which depends upon the representation of internal stress in member in terms of number of independent force unknowns i.e. $F_1, F_2 \ldots. F_n$.

4. By using Step 2 and Step 3, develop the element equilibrium matrix $[B_e]$ for each element.
5. Develop element flexibility matrix $[G_e]$.
6. Develop dual matrix by using Eq. (4.11).
7. Develop global dual matrix $[D_{difm}]$ by using standard assembly procedure. Remove the rows and columns which correspond to restrained displacements.
8. Find the solution of equations to find the free displacements by using standard solver.
9. Get all the independent force unknowns $\{F_1, F_2 \ldots\ldots F_n.\}$ using Eq. (4.6).

CHAPTER 5

IFM BASED FORMULATION OF 2D ELEMENTS

5.1 IMPORTANCE OF STRESS AND DISPLACEMENT FUNCTION

The accuracy of IFM based solution of different types of problems depends upon the formulation of following three matrices; Basic equilibrium matrix $[B_e]$, Compatibility matrix [C] and Flexibility matrix $[G_e]$, where contribution of displacement and stresses functions plays a key role. Both displacements and stresses are approximated by using properly derived shape functions.

In this chapter, IFM based formulation for rectangular and triangular shape elements is developed. The displacement field within the element is represented using appropriate interpolation formula whereas stress field is aproximated using complete polynomial of proper order where coefficients of force polynomial are independent. They are considered as prime unknowns in IFM based analysis. Each equation of force consisting of stress tensor are generally derived using standard Airy Stress Theory. The element stress polynomial should identically satisfy the equation of equilibrium. The stress field within an element is derived without any reference to the shape and number of kinematic degrees of freedom. Number of independent forces are generally chosen such that $n \geq m - l$, where n represents number of independent forces per element (F_1, F_2……F_n = fdof), m represents number of nodal displacement degrees of freedom per element ($w_1, \theta_{x1}, \theta_{y1 \,\ldots\ldots m}$ = ddof) and l represents number of rigid body modes for the element for hybrid method and number of independent strains present in element for IFM which is equal to 3, i.e. ε_x, ε_y and γ_{xy} for 2D problems.

In IFM based analysis a detection of zero energy mode and suppressing of it is strongly required, which is automatically taken care by approximating stress and displacement functions of appropriate order that produces correct rank of the given equilibrium matrix. The methodology of Pian and Chen **[84]** is found truly ensure the absence of spurious zero energy modes. The expression of internal energy helps in calculating the correct rank of equilibrium matrix, in which zero energy modes correspond to rigid body modes are given by the following formula:

$$[A_c] = \frac{1}{2}[F]^T[B]^T\{\delta\} \qquad \dots (5.1)$$

Where [F] = Number of force degree of freedom for each element, [B] = Elemental equilibrium matrix, {δ} = Displacement degrees of freedom. Thus spurious zero-energy mode can be removed sometimes by constructing totally new element if needed, so that the resulting equilibrium matrix has a rank of $n_r \geq m - 1$ **[85].**

5.2 BASIC LINEAR 2D STRESS FUNCTIONS

Consider a four-noded, 8 ddof (u_1 to v_4) rectangular in-plane element of thickness t with dimensions as 2a x 2b along x and y axes respectively as shown in **Fig. 5.1.**

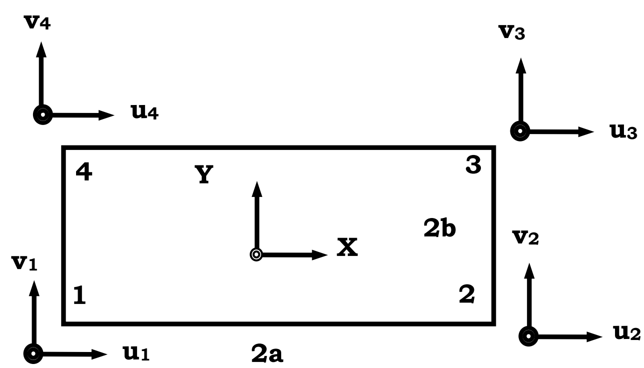


Fig. 5.1 Rectangular Element with Nodal Displacements

For a rectangular membrane element, the force field is chosen in terms of five independent forces as;

$$\{F\} = [F_1\ F_2\ F_3\ F_4\ F_5]^T \qquad \dots (5.2)$$

Here the number of internal forces and their selection totally depend upon formula given in Eq. (5.1). The distribution of internal forces in the element can be considered as follows **(Fig. 5.2)**.

$$N_x = F_1 + F_2\frac{y}{b},\ N_y = F_3 + F_4\frac{x}{a} \text{ and } N_{xy} = F_5 \qquad \dots (5.3)$$

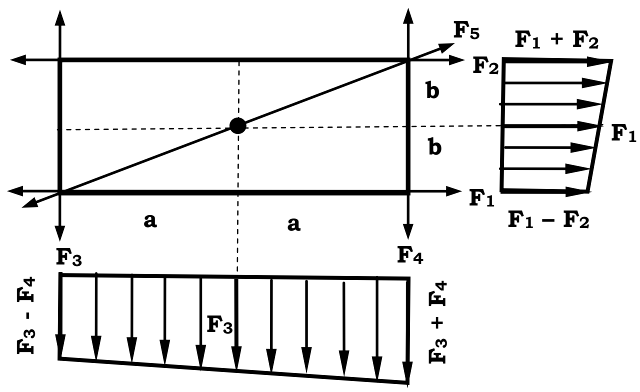


Fig. 5.2 Rectangular Element with Internal Stress Distribution

The structural idealization of a 2D plane rectangular plate may be thought of as being represented by pin-jointed frame work with bars representing sides of rectangular plate and one of their diagonal is such that where adjacent sides of two rectangular plate elements meet **[86].** The same can be approximated by a small plate, which is subjected to axial force in x-x direction and straining takes place in the same direction of an amount $\frac{\gamma\sigma_x}{E}$. The shear stress in diagonal direction converts the rectangular element in rhombus shape. This causes one of the diagonals extent and other contract. Thus, its effect don't interfere the strain produced by normal stress (σ_x and σ_y). Due to this reason its distribution is taken separately and is represented by a constant term as F_5 **[87]**.

The displacement field should satisfy the continuity condition and the selected forces should satisfy the mandatory requirement. Lagrangian interpolation functions are selected to represent displacement field inside the element as follows:

$$u_{(x,y)} = \frac{1}{4}\begin{Bmatrix}\left(1-\frac{x}{a}\right)\left(1-\frac{y}{b}\right)u_1 + \left(1+\frac{x}{a}\right)\left(1-\frac{y}{b}\right)u_2 \\ + \left(1+\frac{x}{a}\right)\left(1+\frac{y}{b}\right)u_3 + \left(1-\frac{x}{a}\right)\left(1+\frac{y}{b}\right)u_4\end{Bmatrix} \quad \ldots (5.4)$$

$$v_{(x,y)} = \frac{1}{4}\begin{Bmatrix}\left(1-\frac{x}{a}\right)\left(1-\frac{y}{b}\right)v_1 + \left(1+\frac{x}{a}\right)\left(1-\frac{y}{b}\right)v_2 \\ + \left(1+\frac{x}{a}\right)\left(1+\frac{y}{b}\right)v_3 + \left(1-\frac{x}{a}\right)\left(1+\frac{y}{b}\right)v_4\end{Bmatrix} \quad \ldots (5.5)$$

where u_1, v_1 -----------v_4 are the nodal displacements as shown in **Fig. 5.1.**

5.3 AIRY STRESS THEORY BASED FUNCTIONS

Airy stress theory is basically based on adding different order of x and y polynomials which correspond to stresses developed in the continuum, which are varying along horizontal and vertical directions from the origin under consideration.

As per the theorem of calculus if the two functions f(x,y) and g(x,y) satisfy

$$\frac{\partial f(x,y)}{\partial x} = \frac{\partial g(x,y)}{\partial y} \quad \ldots (5.4)$$

then, there exists a function A(x,y) such that f(x,y) = $\sigma_x = \frac{\partial A}{\partial y}$ and g(x,y) = $\tau_{xy} = -\frac{\partial A}{\partial x}$ for which an equilibrium equation given below should be satisfied along x-x direction.

$$\frac{\partial \sigma x}{\partial x} + \frac{\partial \tau xy}{\partial y} = 0 \quad \ldots (5.5)$$

If same procedure is followed for the B(x,y), then one can have f(x,y) = $\sigma_y = \frac{\partial B}{\partial x}$ and g(x,y) = $\tau_{xy} = -\frac{\partial B}{\partial y}$ for which an equilibrium equation given below should be satisfied along y-y direction.

$$\frac{\partial \sigma y}{\partial y} + \frac{\partial \tau xy}{\partial x} = 0 \quad \ldots (5.6)$$

Finally, from $\frac{\partial A}{\partial x} = \frac{\partial B}{\partial y}$, one can deduce that there exists a function $\emptyset(x, y)$, which is known as Airy Stress Function, such that A = $\frac{\partial \emptyset}{\partial y}$ and B = $\frac{\partial \emptyset}{\partial x}$, from which three components for stress field can be represented by the stress function **[88]** :

$$\sigma_x = \frac{\partial^2 \emptyset}{\partial y^2}, \; \sigma_y = \frac{\partial^2 \emptyset}{\partial x^2} \text{ and } \tau_{xy} = -\frac{\partial^2 \emptyset}{\partial x \partial x} \quad \ldots (5.7)$$

The Airy stress based polynomial requires a suitable stress field for deriving the stress interpolation matrix [Y], which is directly linked to equilibrium and flexibility matrices. Hence proper selection of polynomial is important to obtain accurate results. A method is developed here which uses an Airy stress function ϕ in terms of complete polynomial. The Airy stress function for a location (x, y) within an element can be written as a complete polynomial of order p as follows;

$$\emptyset(x, y) = \sum_{j=0}^{p} C_j x^{p-j} y^j \quad \ldots (5.8)$$

where C_j with j = 0, 1, 2, 3-------------, p are constants and x, y are the cartesian coordinates of the point in the element for the local coordinate system. The components of stress tensor are obtained using the definitions of the stress function as

$$\sigma_x = \sum_{j=0}^{p-2} C_{j+2} (j+1)(j+2) x^{p-2-j} y^j$$

$$\sigma_y = \sum_{j=0}^{p-2} C_j (p-j)(p-j-1) x^{p-2-j} y^j$$

$$\tau_{xy} = -\sum_{j=0}^{p-2} C_j (j+1)(p-j-1) x^{p-2-j} y^j \quad \ldots (5.9)$$

which can be rewritten as

$$\sigma_x = \sum_{j=0}^{p-2} F_{j+1} x^{(p-j-2)} y^j$$

$$\sigma_y = \sum_{j=0}^{p-2} F_{j+p-1} x^{(p-j-2)} y^j$$

$$\tau_{xy} = -\sum_{j=0}^{p-2} F_{j+2p-1}\, x^{(p-j-2)} y^{j} \qquad \text{... (5.10)}$$

The coefficients of the polynomials in Eq. (5.10) can be considered as elemental forces F_i, with i = 1, 2,......., 3(p+1), and F_i can be expressed in terms of (p+1) constants Cj as

$$F_i = \emptyset(C_0, C_1, \ldots\ldots C_p) \qquad \text{for } i = 1, 2, \ldots\ldots 3(p-1) \qquad \text{... (5.11)}$$

Where $\emptyset$ is the linear function with constants C_j. Thus, not all forces F_i are linearly independent. Final stress field interpolation polynomials can be obtained by eliminating the dependent forces. This results in (p+1) independent forces, when stress function $\emptyset$ is written as a complete polynomial of order (p - 2). The stress function is represented by complete cubic polynomial as follows.

$$\emptyset(x,y) = C_0 x^3 + C_1 x^2 y + C_2 x y^2 + C_3 y^2 \qquad \text{... (5.12)}$$

Substituting p = 3 into Eq. (5.10) yields the following expressions for the stress components:

$$\sigma_x^1 = 2C_2 x + 6C_3 y = \dot{F}_1 x + \dot{F}_2 y$$
$$\sigma_y^1 = 6C_0 x + 2C_1 y = \dot{F}_3 x + \dot{F}_4 y$$
$$\tau_{xy}^1 = -2C_1 x - 2C_2 y = \dot{F}_5 x + \dot{F}_6 y \qquad \text{... (5.13)}$$

Where the superscript 1 denotes the linear terms in the stress polynomial. From Eq. (5.13), it is clear that the six coefficients $\dot{F}_i$ are expressed in terms of four independent constants C_j ; and only four $\dot{F}_i$ forces are linearly independent. By eliminating two forces from Eq. (5.13), one can obtain the linear stress terms as follows:

$$\sigma_x = F_1 x + F_2 y\,, \quad \sigma_y = F_3 x + F_4 y \ \text{ and } \tau_{xy} = -F_4 x - F_1 y \qquad \text{... (5.14)}$$

By substituting p = 2, 3, 4 and 5 into Eq. (5.10) and then eliminating various intermediate terms using above procedure, one can obtain constant, linear quadratic and cubic terms for the stress components as **[89, 90],**

- Constant terms (p = 2), Number of forces (F) = P +1 = 3

  $\sigma_x = F_1, \ \sigma_y = F_2$ and $\tau_{xy} = F_3$ ... (5.15)

- Linear terms (p = 3), Number of forces (F) = P +1 = 4

  $\sigma_x = F_1x + F_2y, \ \sigma_y = F_3x + F_4y$ and $\tau_{xy} = -F_4x - F_1y$ ... (5.16)

- Quadratic terms (p = 4), Number of forces (F) = P +1 = 5

  $\sigma_x = F_1y^2 + F_2xy - F_5 (x^2/2), \ \sigma_y = F_3x^2 - F_5 (y^2/2) + F_4xy$

  and $\tau_{xy} = - F_4 (x^2/2) - F_2 (y^2/2) + F_5xy$ ... (5.17)

- Cubic terms (p = 5), Number of forces (F) = P +1 = 6

  $\sigma_x = (F_3/3)x^3 + 3F_4x^2y + 3F_5xy^2 - F_6y^3$

  $\sigma_y = F_1x^3 + 3F_2x^2y + F_3xy^2 + F_4y^3$

  $\tau_{xy} = - F_2 x^3 - F_3x^2 y - 3F_4xy^2 - F_5y^3$ ... (5.18)

For solving various problems, one can add above equations to develop full polynomial, like full constant terms have total 3 terms in terms of internal forces in Eq. (5.15). The full linear force polynomial can be written as

$\sigma_x = F_1 + F_4x + F_5y$, $\sigma_y = F_2 + F_6x + F_7y$ and $\tau_{xy} = F_3 - F_7x - F_4y$ ... (5.19)

Similarly, full quadratic polynomials can be written as

$\sigma_x = F_1 + F_4x + F_5y + F_8y^2 + F_9xy - F_5 (x^2/2)$

$\sigma_y = F_2 + F_6x + F_7y + F_{10}x^2 - F_{12} (y^2/2) + F_{11}xy$

$\tau_{xy} = F_3 - F_7x - F_4y - F_{11} (x^2/2) - F_9(y^2/2) + F_{12}xy$ ... (5.20)

Table 5.1 gives complete information regarding all affecting parameters for developing higher order stress polynomials.

Table 5.1 Various Parameters for Stress Functions

Type of Polynomial	Numbers of Terms in Ø	Maximum Power of x or y	Total Number of Individual Forces in Polynomial	Total Number of Forces in Full Polynomial
Constant	3	2	3	3
Linear	4	3	4	7
Quadratic	5	4	5	12
Cubic	6	5	6	18

5.4 REDUCTION TECHNIQUE FOR HIGHER ORDER POLYNOMIALS

Higher order elements are preferable for increasing the accuracy but such elements increase the total number of unknowns and thus gives large size of global equilibrium matrix, compatibility matrix and material flexibility matrix. For the last two matrices there is no mathematical complexity. However, the individual element equilibrium matrix is of sparse nature, assembly of the same may raise percentage sparsity to considerable level which may create problems in finding the solution. Thus, it is preferable to reduce the number of independent forces in the stress field representation while preserving the original desired mathematical property of overall problem and related matrices. The compatibility condition can be used to reduce number of independent forces. It is written as follows **[91]**;

$$\nabla^2 (\sigma_x + \sigma_y) = 0 \qquad \dots (5.21)$$

where $\nabla^2 = \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2}$

Applying Eq. (5.21) to cubic representation one may reduce one or two independent forces in the complete polynomial. On the other side, cubic representation may be converted into a quadratic representation by reducing one or two individual force unknowns. It is preferable to apply compatibility conditions to the force polynomial which is of higher order than linear terms as second derivative may not give necessary results for linear and constant representations.

5.5 INTERPOLATION FUNCTIONS FOR INPLANE PROBLEMS

Function used to represent behavior of a field variable within an element are called interpolation functions. These functions are also known as shape functions. Different types of functions could be used as interpolation functions such as polynomials or trignometric functions. As polynomials are easy to manipulate and have recognizable degree of approximation, they are most widely used as interpolation functions. Depending on the problem dimension, polynomials of one, two or three independent variables may be used in the interpolation functions. Starting directly with polynomial, however, requires inversion of generalized coordinate matrix to derive shape function N_i. Such a process can be quite cumbersome for higher order polynomials. Alternatively, shape functions can be easily derived by using Lagrangian formula for any simple or most sophisticated element. However, these functions can be used for C^o degree continuity elements only i.e. where only function and not their derivatives are used as degree of freedom.

The displacement field can be expressed, for a 1D element, as follows:

$$U = \sum_{i=1}^{n} N_i \, u_i \qquad \text{... (5.22)}$$

Where n is number of nodes and interpolation function N_i can be calculated by using the formula:

$$N_i = \prod_{\substack{j=1 \\ J \neq i}}^{n} \frac{x - x_j}{x_i - x_j} \qquad \text{... (5.23)}$$

For example, for 2-noded element, the linear functions for u can be written as

$$u = N_1 u_1 + N_2 u_2$$

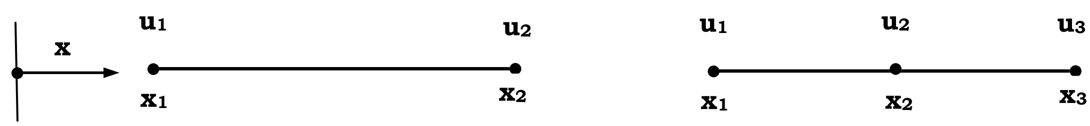
Where N1 = $\frac{x - x_2}{x_1 - x_2}$ and N2 = $\frac{x - x_1}{x_2 - x_1}$

For 3 - noded element having quadratic variation, one can write

$$u = N_1 u_1 + N_2 u_2 + N_3 u_3$$

where N1 = $\frac{x - x_2}{x_1 - x_2}\frac{x - x_3}{x_1 - x_3}$, N2 = $\frac{x - x_1}{x_2 - x_1}\frac{x - x_3}{x_2 - x_3}$ and N3 = $\frac{x - x_1}{x_3 - x_1}\frac{x - x_2}{x_3 - x_3}$

The interpolation functions can be written by considering the origin of coordinates outside the element as shown in **Fig. 5.3** or at the left node of the element or may be at the centre of the element.



a) Two Noded Line Element **b) Three Noded Line Element**

Fig. 5.3 One Dimensional Elements

The procedure can be easily generalized to accomplish shape functions for two and three dimensional elements. Consider a rectangular element in plane stress with nodes at corners as shown in **Fig. 5.4(a).** The displacements in the x-direction u and in y direction v can be written in terms of nodal displacements as

$$u = N_1(x,y)u_1 + N_2(x,y)u_2 + N_3(x,y)u_3 + N_4(x,y)u_4$$

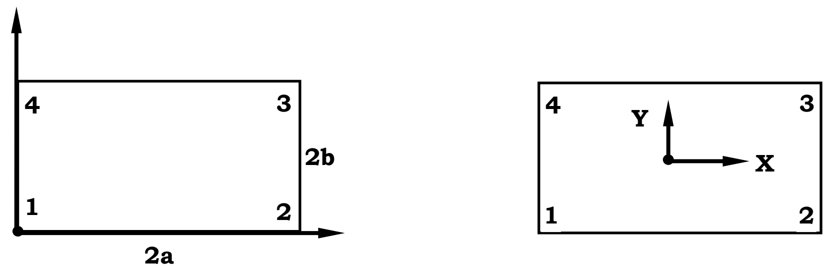
$$v = N_1(x,y)v_1 + N_2(x,y)v_2 + N_3(x,y)v_3 + N_4(x,y)v_4 \quad \text{... (5.24)}$$

where, $N_1(x,y) = N_{1(x)}N_{1(y)}$, $N_2(x,y) = N_{2(x)}N_{1(y)}$

$N_3(x,y) = N_{2(x)}N_{2(y)}$ and $N_4(x,y) = N_{1(x)}N_{2(y)}$

Where $N_{1(x)} = (1 - \frac{x}{2a})$, $N_{2(x)} = \frac{x}{2a}$

$$N_{1(y)} = \left(1 - \frac{y}{2b}\right), \qquad N_{2(y)} = \frac{y}{2b} \quad \text{... (5.25)}$$



a) Origin at Node 1 **b) Origin at Centre**

Fig. 5.4 Two Dimensional Rectangular Element

Considering origin at the centre of the element as shown in **Fig. 5.4(b),** the shape functions along each direction can be written as

$$N_{1(x)} = \frac{1}{2}(1 - \frac{x}{a}) \ , \quad N_{1(y)} = \frac{1}{2}\left(1 - \frac{y}{b}\right),$$

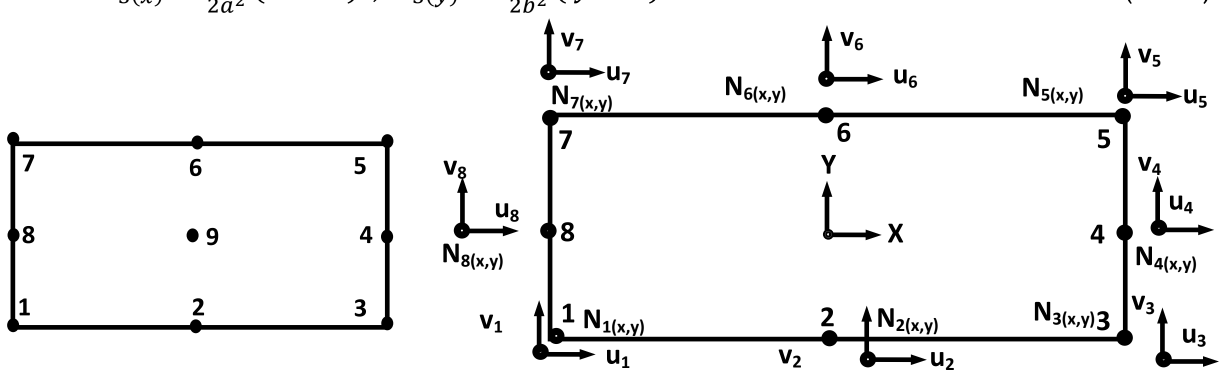
$$N_{2(x)} = \frac{1}{2}(1 + \frac{x}{a}) \ , \text{ and } \ N_{2(y)} = \frac{1}{2}(1 + \frac{y}{b}) \qquad \dots (5.26)$$

5.6 SHAPE FUNCTION FOR 2D QUADRATIC RECTANGULAR ELEMENT

For 3 nodes in each direction, with origin at the centre

$$N_{1(x)} = \frac{x}{2a^2}(x - a) \ , \quad N_{1(y)} = \frac{y}{2b^2}(y - b), \quad N_{2(x)} = (1 - \frac{x^2}{a^2}) \ , \quad N_{2(y)} = (1 - \frac{y^2}{b^2})$$

$$N_{3(x)} = \frac{x}{2a^2}(x + a) \ , N_{3(y)} = \frac{y}{2b^2}(y + b) \qquad \dots (5.27)$$



a) 9-Noded Element **b) 8-Noded Element**

Fig. 5.5 Quadratic Rectangular Element

Hence for 9 noded rectangular element **Fig. 5.5(a)** which belongs to Lagrangian family, the interpolation function will be

$$N_1(x,y) = \frac{x}{2a^2}(x-a)\,\frac{y}{2b^2}(y-b), \quad N_2(x,y) = \left(1-\frac{x^2}{a^2}\right)\frac{y}{2b^2}(y-b),$$

$$N_3(x,y) = \frac{x}{2a^2}(x+a)\frac{y}{2b^2}(y-b), \quad N_4(x,y) = \frac{x}{2a^2}(x+a)\,(1-\frac{y^2}{b^2})$$

$$N_5(x,y) = \frac{x}{2a^2}(x+a)\frac{y}{2b^2}(y-b), \quad N_6(x,y) = (1-\frac{x^2}{a^2})\,\frac{y}{2b^2}(y+b)$$

$$N_7(x,y) = \frac{x}{2a^2}(x-a)\,\frac{y}{2b^2}(y+b), \quad N_8(x,y) = \frac{x}{2a^2}(x-a)\,(1-\frac{y^2}{b^2})$$

and $N_9(x,y) = (1-\frac{x^2}{a^2})\,(1-\frac{y^2}{b^2})$ ... (5.28)

Eliminating the internal node, one can have the interpolation functions for 8 noded element **Fig.5.5(b).** For mid side nodes, shape functions are obtained by considering quadratic variation along 3-noded line and linear variation along 2 noded line. For corner nodes, shape functions are obtained by substracting from linear variation along its sides the half of the shape functions at mid side nodes lying on the sides of the respective corner node. Finally,

$$N_1(x,y) = \frac{1}{4}\left(1-\frac{x}{a}\right)\left(1-\frac{y}{b}\right)\left(-\frac{x}{a}-\frac{y}{b}-1\right), \; N_2(x,y) = \frac{1}{2}(1-\frac{x^2}{a^2})\,\left(1-\frac{y}{b}\right)$$

$$N_3(x,y) = \frac{1}{4}\left(1+\frac{x}{a}\right)\left(1-\frac{y}{b}\right)\left(\frac{x}{a}-\frac{y}{b}-1\right), \; N_4(x,y) = \frac{1}{2}(1+\frac{x}{a})\,\left(1-\frac{y^2}{b^2}\right)$$

$$N_5(x,y) = \frac{1}{4}(1+\frac{x}{a})\,(1+\frac{y}{b})\left(\frac{x}{a}+\frac{y}{b}-1\right) \quad N_6(x,y) = \frac{1}{2}(1-\frac{x^2}{a^2})\,\left(1+\frac{y}{b}\right)$$

$$N_7(x,y) = \frac{1}{4}(1-\frac{x}{a})(1+\frac{y}{b})\,\left(-\frac{x}{a}+\frac{y}{b}-1\right) \text{ and } N_8(x,y) = \frac{1}{2}(1-\frac{x}{a})\,(1-\frac{y^2}{b^2})$$

... (5.29)

Using above shape functions the displacement u in the x direction and v in the y direction can be expressed as

$$u = N_1u_1 + N_2u_2 + N_3u_3 + \dots\dots +N_8u_8 = \sum_{i=1}^{n} N_i\, u_i$$

$$v = N_1v_1 + N_2v_2 + N_3v_3 + \dots\dots\dots +N_8v_8 = \sum_{i=1}^{n} N_i\, v_i \quad \dots (5.30)$$

5.7 IFM BASED IN-PLANE TRIANGULAR ELEMENTS

The major advantage of the triangular type of element is that it can be used to discretize any irregular shape domain. The mathematical expression and related matrices are relatively simpler; hence it is always suited in terms of domain geometry and computational cost.

Following two types of triangular elements are developed in the present work: 1. Basic Triangular Element (TRI_3F_6D) and 2. Higher order Triangular Element (TRI_9F_12D).

5.7.1 Triangular Element (TRI_3F_6D)

Consider a 3-noded triangular element as shown in **Fig. 5.6**. The element is designated as TRI_3F_6D where 3F represents three independent forces (F_1, F_2, F3) and 6D represents total number of displacements u_1,v_1, u_2,v_2 and u_3, v_3 respectively at nodes 1, 2 and 3 of the element.

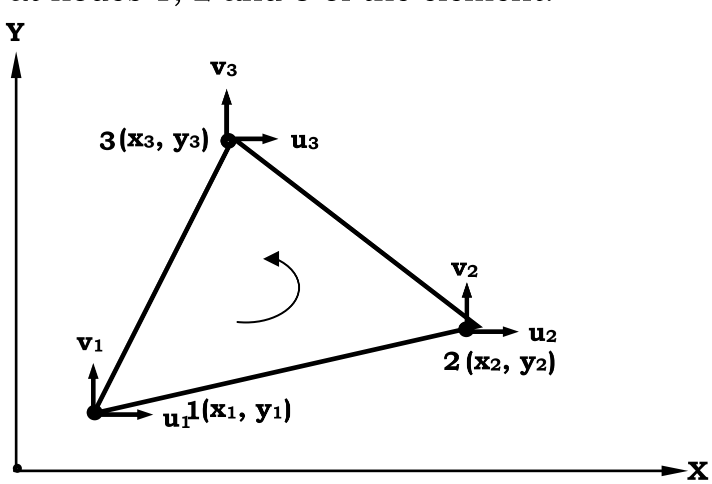


Fig. 5.6 Basic Triangular Element (CST)

Here (x_1, y_1), (x_2, y_2) and (x_3, y_3) are the known nodal coordinates of nodes 1, 2 and 3 respectively. The procedure for deriving basic equilibrium matrix is based on displacement function and appropriate forcing function.

Selecting linear displacement function in u and v direction as

$$u_{(x,y)} = c_1 + c_2x + c_3y$$

$$v_{(x,y)} = c_4 + c_5x + c_6y \qquad \ldots (5.31)$$

The unknown nodal displacement vector {δ} is as follows;

$$\{\delta\} = \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} \qquad \ldots (5.32)$$

Thus writing Eq. (5.31) in matrix form gives

$$\{\phi\} = \begin{Bmatrix} u_{(x,y)} \\ v_{(x,y)} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{Bmatrix} c_1 \\ c_2 \\ c_3 \\ c_4 \\ c_5 \\ c_6 \end{Bmatrix}$$

$$\{\phi\} = [M^*]\{C\} \qquad \ldots (5.33)$$

Considering cartesian set of axes and substituting nodal coordinates one can write in matrix form as

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix} \begin{Bmatrix} c_1 \\ c_2 \\ c_3 \end{Bmatrix} \qquad \ldots (5.34)$$

Calculating value of constants by inverting in terms of nodal coordinates of only $u_{(x,y)}$ in Eq. (5.34) one can have

$$\begin{Bmatrix} c_1 \\ c_2 \\ c_3 \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} \alpha_1 & \alpha_2 & \alpha_3 \\ \beta_1 & \beta_2 & \beta_3 \\ \eta_1 & \eta_2 & \eta_3 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \qquad \text{... (5.35)}$$

In which, $\alpha_1 = x_2y_3 - y_2x_3$, $\alpha_2 = x_3y_1 - y_3x_1$, $\alpha_3 = x_1y_2 - y_1x_2$,

$\beta_1 = y_2 - y_3$, $\beta_2 = y_3 - y_1$, $\beta_3 = y_1 - y_2$,

$\eta_1 = x_3 - x_2$, $\eta_2 = y_1 - y_3$ and $\eta_3 = y_2 - y_1$

While, $2A = x_1(y_2 - y_3) + x_2(y_3 - y_1) + x_3(y_1 - y_2)$ is the determinant of the generalized coordinate matrix given in Eq. (5.34) and is equal to two times the area of the triangle. Thus substituting vector $[c_1\ c_2\ c_3]^T$ from Eq. (5.35) into Eq. (5.34), one gets

$$u_{(x,y)} = \frac{1}{2A}[1 \quad x \quad y]\begin{bmatrix} \alpha_1 & \alpha_2 & \alpha_3 \\ \beta_1 & \beta_2 & \beta_3 \\ \eta_1 & \eta_2 & \eta_3 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \qquad \text{... (5.36)}$$

Multiplying, one gets

$$u_{(x,y)} = \frac{1}{2A}\{(\alpha_1 + \beta_1 x + \eta_1 y)u_1 + (\alpha_2 + \beta_2 x + \eta_2 y)u_2 + (\alpha_3 + \beta_3 x + \eta_3 y)u_3\} \qquad \text{... (5.37)}$$

$$v_{(x,y)} = \frac{1}{2A}\{(\alpha_1 + \beta_1 x + \eta_1 y)v_1 + (\alpha_2 + \beta_2 x + \eta_2 y)v_2 + (\alpha_3 + \beta_3 x + \eta_3 y)v_3\} \qquad \text{... (5.38)}$$

Coefficients of nodal displacements are shape functions which can be written as

$$N_1 = \frac{1}{2A}(\alpha_1 + \beta_1 x + \eta_1 y)$$

$$N_2 = \frac{1}{2A}(\alpha_2 + \beta_2 x + \eta_2 y) \qquad \text{... (5.39)}$$

$$N_3 = \frac{1}{2A}(\alpha_3 + \beta_3 x + \eta_3 y)$$

Thus, one can write

$$u_{(x,y)} = N_1u_1 + N_2u_2 + N_3u_3$$

$$v_{(x,y)} = N_1v_1 + N_2v_2 + N_3v_3 \quad \text{... (5.40)}$$

$$\{\phi\} = \begin{Bmatrix} u_{(x,y)} \\ v_{(x,y)} \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_2 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = [N]\{\delta\} \quad \text{... (5.41)}$$

Where [N] is the shape function matrix which is given by

$$[N] = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \quad \text{... (5.42)}$$

For plane stress/strain problem, strain vector can be written as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial v}{\partial x} + \frac{\partial u}{\partial x} \end{Bmatrix} \quad \text{... (5.43)}$$

Substituting the Eq. (5.41) into (5.43)

$$= \begin{Bmatrix} \frac{\partial}{\partial x}(N_1u_1 + N_2u_2 + N_3u_3) \\ \frac{\partial}{\partial y}(N_1v_1 + N_2v_2 + N_3v_3) \\ \frac{\partial}{\partial x}(N_1v_1 + N_2v_2 + N_3v_3) + \frac{\partial}{\partial y}(N_1u_1 + N_2u_2 + N_3u_3) \end{Bmatrix}$$

$$= \frac{1}{2A} \begin{bmatrix} \beta_1 & 0 & \beta_2 & 0 & \beta_3 & 0 \\ 0 & \eta_1 & 0 & \eta_2 & 0 & \eta_3 \\ \eta_1 & \beta_1 & \eta_2 & \beta_2 & \eta_3 & \beta_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = [Z]\{\delta\} \quad \text{... (5.44)}$$

Where, [Z] is known as strain displacement linking matrix.

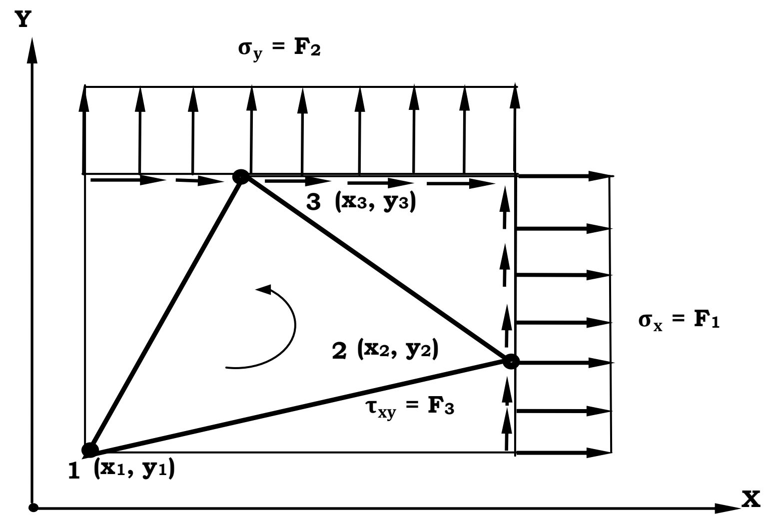


Fig. 5.7 Constant Stress Distribution for Triangular Element

Now number of independent unknowns in forcing function {F} = Total displacement degrees of freedom (ddof) – Number of rigid body modes (3) = 3

So, $\{F\} = [F_1 \; F_2 \; F_3]^T$ … (5.45)

Expressing internal stress distribution of the given element in terms of independent unknowns, one can write

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_2 \end{Bmatrix} = [Y]\{F\} \quad \text{… (5.46)}$$

The equilibrium matrix $[B_e]$ for the given triangular element is given by

$$[B_e] = \int_A [Z]^T [Y] \, dA \quad \text{… (5.47)}$$

Now substituting strain linking matrix [Z] from Eq. (5.44) and [Y] from Eq. (5.46) into above equation. The equilibrium matrix can be written as

$$[B_e] = \frac{1}{2} \begin{bmatrix} \beta_1 & 0 & \eta_1 \\ 0 & \eta_1 & \beta_1 \\ \beta_2 & 0 & \eta_2 \\ 0 & \eta_2 & \beta_2 \\ \beta_3 & 0 & \eta_3 \\ 0 & \eta_3 & \beta_3 \end{bmatrix} \quad \text{… (5.48)}$$

The elemental flexibility matrix for the triangular element can be worked out using the following expression.

$$[G_e] = \int_A [Y]^T [D][Y]dA \qquad \dots (5.49)$$

In which material property matrix [D] is written as

$$[D] = \frac{1}{Et}\begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix} \qquad \dots (5.50)$$

Where E, t and ν are the modulus of elasticity, thickness of plate and Poisson's ratio of plate material respectively.

Substituting material matrix [D] and stress interpolation matrix [Y] in Eq. (5.49), one gets

$$[G_e] = \frac{A}{Et}\begin{bmatrix} 1 & -\nu & 0 \\ -\nu & 1 & 0 \\ 0 & 0 & 2(1+\nu) \end{bmatrix} \qquad \dots (5.51)$$

5.7.2 Higher Order Triangular Element (TRI_9F_12D)

Consider a higher order triangular element with 3 additional nodes at the midpoints of the each side. The displacement function has 3 nodes along each edge of triangle, thus, it can be approximated using parabolic behavior.

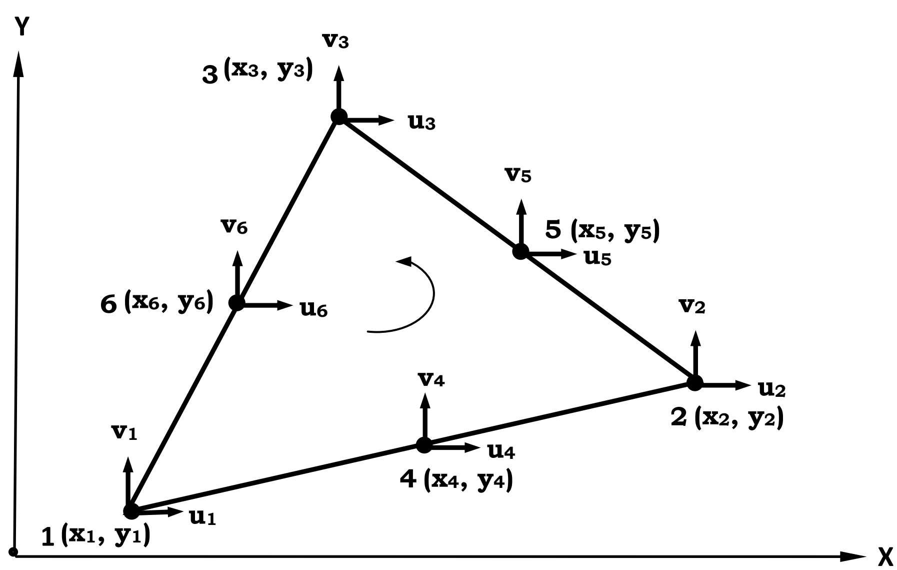


Fig. 5.6 Higher Order Quadratic Triangular Element

Selecting quadratic displacement function for u and v, one can write

$$u_{(x,y)} = c_1 + c_2x + c_3y + c_4x^2 + c_5xy + c_6y^2$$

$$v_{(x,y)} = c_7 + c_8x + c_9y + c_{10}x^2 + c_{11}xy + c_{12}y^2 \qquad \dots (5.52)$$

The unknown nodal displacement vector can be written as

$$\{\delta\} = [\, u_1, v_1 \; \dots\dots u_6, v_6 \,]^T \qquad \dots (5.53)$$

Arranging Eq. (5.52) in matrix form, thus

$$\{\phi\} = \begin{Bmatrix} u_{(x,y)} \\ v_{(x,y)} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & x^2 & xy & y^2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & x^2 & xy & y^2 \end{bmatrix} \begin{Bmatrix} c_1 \\ \cdot \\ \cdot \\ c_{12} \end{Bmatrix}$$

$$\text{or, } \{\phi\} = [M^*]\{C\} \qquad \dots (5.54)$$

Substituting the nodal coordinates, one can have

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix} = \begin{bmatrix} 1 & x_1 & y_1 & {x_1}^2 & x_1y_1 & {y_1}^2 \\ 1 & x_2 & y_2 & {x_2}^2 & x_2y_2 & {y_2}^2 \\ 1 & x_3 & y_3 & {x_3}^2 & x_3y_3 & {y_3}^2 \\ 1 & x_4 & y_4 & {x_4}^2 & x_4y_4 & {y_4}^2 \\ 1 & x_5 & y_5 & {x_5}^2 & x_5y_5 & {y_5}^2 \\ 1 & x_6 & y_6 & {x_6}^2 & x_6y_6 & {y_6}^2 \end{bmatrix} \begin{Bmatrix} c_1 \\ c_2 \\ c_3 \\ c_4 \\ c_5 \\ c_6 \end{Bmatrix} = [M_1]\{C\} \qquad \dots (5.55)$$

Inverting Eq. (5.55) gives

$$\begin{Bmatrix} c_1 \\ c_2 \\ c_3 \\ c_4 \\ c_5 \\ c_6 \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} \alpha_1 & \alpha_2 & \alpha_3 & \alpha_4 & \alpha_5 & \alpha_6 \\ \beta_1 & \beta_2 & \beta_3 & \beta_4 & \beta_5 & \beta_6 \\ \eta_1 & \eta_2 & \eta_3 & \eta_4 & \eta_5 & \eta_5 \\ \zeta_1 & \zeta_2 & \zeta_3 & \zeta_4 & \zeta_5 & \zeta_6 \\ \rho_1 & \rho_2 & \rho_3 & \rho_4 & \rho_5 & \rho_6 \\ \lambda_1 & \lambda_2 & \lambda_3 & \lambda_4 & \lambda_5 & \lambda_6 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix} \qquad \dots (5.56)$$

In which, $\alpha_1, \alpha_2, \dots\dots \lambda_6$ are the elements of generalized coordinate matrix which depends upon values of nodal coordinates. Substituting {C} vector from Eq. (5.56) into Eq.(5.54),

$$u_{(x,y)} = [1 \quad x \quad y \quad x^2 \quad xy \quad y^2]\frac{1}{2A}\begin{bmatrix} \alpha_1 & \alpha_2 & \alpha_3 & \alpha_4 & \alpha_5 & \alpha_6 \\ \beta_1 & \beta_2 & \beta_3 & \beta_4 & \beta_5 & \beta_6 \\ \eta_1 & \eta_2 & \eta_3 & \eta_4 & \eta_5 & \eta_5 \\ \zeta_1 & \zeta_2 & \zeta_3 & \zeta_4 & \zeta_5 & \zeta_6 \\ \rho_1 & \rho_2 & \rho_3 & \rho_4 & \rho_5 & \rho_6 \\ \lambda_1 & \lambda_2 & \lambda_3 & \lambda_4 & \lambda_5 & \lambda_6 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \\ u_6 \end{Bmatrix} \ldots (5.57)$$

or one can write expressions for u and v as

$$u_{(x,y)} = \sum_{i=1}^{6} \frac{1}{2A}\{\alpha_i + \beta_i x + \eta_i y + \zeta_i x^2 + \rho_i xy + \lambda_i y^2\}u_i = \frac{1}{2A}\sum_{i=1}^{6} N_i u_i$$

$$v_{(x,y)} = \sum_{i=1}^{6} \frac{1}{2A}\{\alpha_i + \beta_i x + \eta_i y + \zeta_i x^2 + \rho_i xy + \lambda_i y^2\}u_i = \frac{1}{2A}\sum_{i=1}^{6} N_i v_i$$

In matrix form, the same can be written as

$$\{\phi\} = [N]\{u\} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 & N_5 & 0 & N_6 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 & N_5 & 0 & N_6 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ . \\ . \\ u_6 \\ v_6 \end{Bmatrix}$$

$$\ldots (5.58)$$

Strain vector for a plane stress/strain problem can be written as

$$\{\epsilon\} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \nu_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \end{Bmatrix} =$$

$$\begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 & \frac{\partial N_5}{\partial x} & 0 & \frac{\partial N_6}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} & 0 & \frac{\partial N_5}{\partial y} & 0 & \frac{\partial N_6}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial y} & \frac{\partial N_4}{\partial x} & \frac{\partial N_5}{\partial y} & \frac{\partial N_5}{\partial x} & \frac{\partial N_6}{\partial y} & \frac{\partial N_6}{\partial x} \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ . \\ . \\ u_6 \\ v_6 \end{Bmatrix} = [Z]\{u\}$$

$$= \frac{1}{2A}\begin{bmatrix} \sum_{i=1}^{6}\{\beta_i + 2\zeta_i x + \rho_i y\} & 0 \\ 0 & \sum_{i=1}^{6}\{\eta_i + \rho_i x + 2\lambda_i y\} \\ \sum_{i=1}^{6}\{\eta_i + \rho_i x + 2\lambda_i y\} & \sum_{i=1}^{6}\{\beta_i + 2\zeta_i x + \rho_i y\} \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ . \\ . \\ u_6 \\ v_6 \end{Bmatrix} = [Z]\{u\} \qquad \ldots (5.59)$$

The forcing function {F} with nine number of internal unknowns can be expressed as

So, $\{F\} = [F_1\ F_2\ F_3\ .....F_9]^T$

The internal stress distribution of the given element can be expressed in terms of independent unknowns as

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \\ F_9 \end{Bmatrix} = [Y]\{F\} \qquad \dots (5.60)$$

Where, [Y] is known as stress interpolation matrix.

The distribution of internal stresses is depicted in **Fig 5.7.**

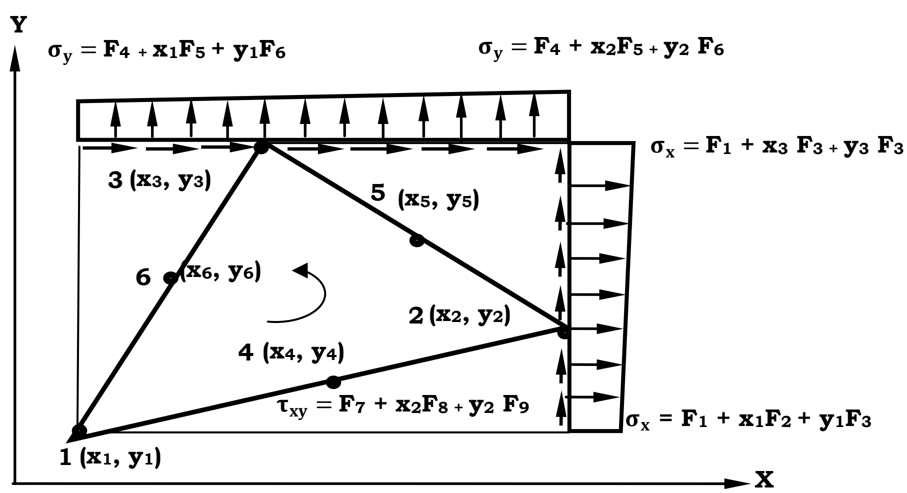


Fig. 5.7 Variation of Internal Stresses along Orthogonal Directions

Now element equilibrium matrix [B_e] is formulated by substituting [Z] matrix from Eq. (5.59) and stress interpolation matrix [Y] from Eq. (5.60) into Eq. (5.47). The elemental flexibility matrix [G_e] is also formulated by substituting necessary matrices in Eq. (5.49)

Instead of using generalized coordinate system, one can use the non-dimensional natural coordinate system, known as are coordinate system used to derive the basic element matrices. Here three coordinates L_1, L_2 and L_3 are used to define the location of any point in the element, in which two of these are independent.

The relation between area coordinate system and cartesian coordinates x, y is can be written as

$$\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix} \begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} \quad \dots (5.61)$$

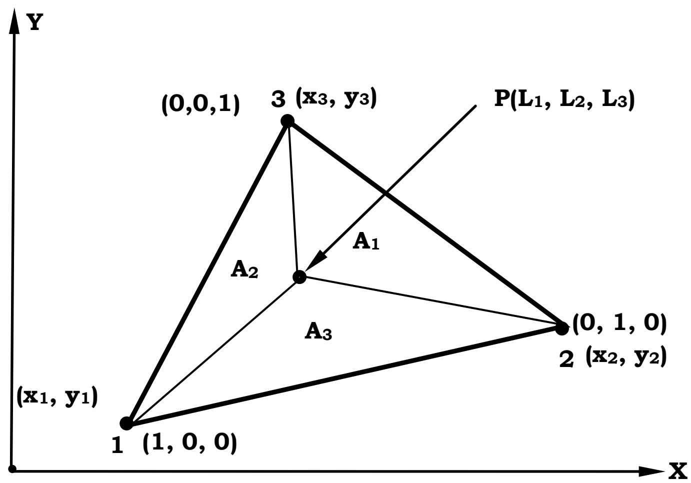


Fig. 5.8 Area Coordinate System for Triangular Element

If A is the area of triangle 1-2-3, and A_1, A_2, and A_3 are the areas of the smaller triangles as shown in **Fig. 5.8**, then the non-dimensional coordinates can be defined as

$L_1 = A_1/A$, $L_2 = A_2/A$ and $L_3 = A_3/A$

Now, inverting the Eq. (5.61), yields

$$\begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} = \frac{1}{2A} \begin{bmatrix} b_1 & c_1 & a_1 \\ b_2 & c_2 & a_2 \\ b_3 & c_3 & a_3 \end{bmatrix} \begin{Bmatrix} x \\ y \\ 1 \end{Bmatrix} \quad \dots (5.62)$$

Where $b_1 = y_2 - y_3$, $c_1 = x_3 - x_2$ and $a_1 = x_2y_3 - x_3y_2$ and others can be written in cyclic order. Here L_1, L_2 and L_3 are calculated from Eq. (5.62), in which differentiation and integration are carried out as follows.

$$\frac{\partial}{\partial x} = \sum_{i=1}^{n} \frac{\partial L_i}{\partial x}\frac{\partial}{\partial L_i} = \sum_{i=1}^{n} \frac{b_i}{2A}\frac{\partial}{\partial L_i} \quad , \frac{\partial}{\partial y} = \sum_{i=1}^{n} \frac{\partial L_i}{\partial y}\frac{\partial}{\partial L_i} = \sum_{i=1}^{n} \frac{a_1}{2A}\frac{\partial}{\partial L_i}$$

$$\text{and} \quad \int_A L_1{}^p L_2{}^q L_3{}^r \, dA = \frac{p!q!r!}{(p+q+r+2)!}2A \qquad \dots (5.63)$$

Using above mathematical operations and relations, one can calculate the equilibrium matrix [Be] and flexibility matrix [Ge] for both Constant Strain Triangular element (TRI_3F_6D) and Higher order quadratic element (TRI_9F_12D) as follows:

- **TRI_3F_6D Triangular Element**

  The values of area coordinates L_1, L_2, and L_3 are found identical to those N_1, N_2 and N_3 for the 3-noded triangular element of cartesian coordinates and therefore one can write

  $N_1 = L_1$, $N_2 = L_2$ and $N_3 = L_3$

$$\{\phi\} = \begin{Bmatrix} u_{(x,y)} \\ v_{(x,y)} \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_2 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = [N]\{\delta\} \qquad \dots (5.64)$$

Differentiating as per Eq.(5.63), the strains can be related to displacements as

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \end{Bmatrix}$$

$$= \frac{1}{2A} \begin{bmatrix} b_1 & 0 & b_2 & 0 & b_3 & 0 \\ 0 & c_1 & 0 & c_2 & 0 & c_3 \\ c_1 & b_1 & c_2 & b_2 & c_3 & b_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = [Z]\{\delta\} \qquad \dots (5.65)$$

The forcing function {F} is taken from Eq. (5.46) and strain displacement linking matrix [Z] is taken from Eq. (5.65) to get the equilibrium matrix [B_e] as per Eq. (5.47) as

$$[Be] = \frac{1}{2}\begin{bmatrix} b_1 & 0 & c_1 \\ 0 & c_1 & b_1 \\ b_2 & 0 & c_2 \\ 0 & c_2 & b_2 \\ b_3 & 0 & c_3 \\ 0 & c_3 & b_3 \end{bmatrix} \qquad \dots (5.66)$$

Similarly [Ge] is obtained which is same as given in Eq. (5.51)

- **TRI_9F_12D Triangular Element**

Considering higher order triangular element with 3 corner nodes and 3 mid side nodes. Using area coordinate system, the six shape functions are obtained using Lagrangian formula as

$$N_1 = L_1(2L_1 - 1)\ , \quad N_2 = L_2(2L_2 - 1)\ , \quad N_3 = L_3(2L_3 - 1),$$
$$N_4 = 4L_1L_2, \quad N_5 = 4L_2L_3 \text{ and} \quad N_6 = 4L_1L_3 \qquad \dots (5.67)$$

By following the same procedure as outlined above from Eq. (5.39) to (5.51) the equilibrium matrix [B_e] and flexibility matrix [G_e] can be calculated as

$$[B_e] = \begin{bmatrix}
-\frac{b_1}{3} & -\frac{b_1}{18}\bar{x} & -\frac{b_1}{18}\bar{y} & 0 & 0 & 0 & -\frac{c_1}{3} & -\frac{c_1}{18}\bar{x} & -\frac{c_1}{18}\bar{y} \\
0 & 0 & 0 & -\frac{c_1}{3} & -\frac{c_1}{18}\bar{x} & -\frac{c_1}{18}\bar{y} & -\frac{b_1}{3} & -\frac{b_1}{18}\bar{x} & -\frac{b_1}{18}\bar{y} \\
-\frac{b_2}{3} & -\frac{b_2}{18}\bar{x} & -\frac{b_2}{18}\bar{y} & 0 & 0 & 0 & -\frac{c_2}{3} & -\frac{c_2}{18}\bar{x} & -\frac{c_2}{18}\bar{y} \\
0 & 0 & 0 & -\frac{c_2}{3} & -\frac{c_2}{18}\bar{x} & -\frac{c_2}{18}\bar{y} & -\frac{b_2}{3} & -\frac{b_2}{18}\bar{x} & -\frac{b_2}{18}\bar{y} \\
-\frac{b_3}{3} & -\frac{b_3}{18}\bar{x} & -\frac{b_3}{18}\bar{y} & 0 & 0 & 0 & -\frac{c_3}{3} & -\frac{c_3}{18}\bar{x} & -\frac{c_3}{18}\bar{y} \\
0 & 0 & 0 & -\frac{c_3}{3} & -\frac{c_3}{18}\bar{x} & -\frac{c_3}{18}\bar{y} & -\frac{b_3}{3} & -\frac{b_3}{18}\bar{x} & -\frac{b_3}{18}\bar{y} \\
\frac{2b_{\overline{12}}}{3} & \frac{2b_{\overline{12}}}{18}\bar{x} & \frac{2b_{\overline{12}}}{18}\bar{y} & 0 & 0 & 0 & \frac{2c_{\overline{12}}}{3} & \frac{2c_{\overline{12}}}{18}\bar{x} & \frac{2c_{\overline{12}}}{18}\bar{y} \\
0 & 0 & 0 & \frac{2c_{\overline{12}}}{3} & \frac{2c_{\overline{12}}}{18}\bar{x} & \frac{2c_{\overline{12}}}{18}\bar{y} & \frac{2b_{\overline{12}}}{3} & \frac{2b_{\overline{12}}}{18}\bar{x} & \frac{2b_{\overline{12}}}{18}\bar{y} \\
\frac{2b_{\overline{23}}}{3} & \frac{2b_{\overline{23}}}{18}\bar{x} & \frac{2b_{\overline{23}}}{18}\bar{y} & 0 & 0 & 0 & \frac{2c_{\overline{23}}}{3} & \frac{2c_{\overline{23}}}{18}\bar{x} & \frac{2c_{\overline{23}}}{18}\bar{y} \\
0 & 0 & 0 & \frac{2c_{\overline{23}}}{3} & \frac{2c_{\overline{23}}}{18}\bar{x} & \frac{2c_{\overline{23}}}{18}\bar{y} & \frac{2b_{\overline{23}}}{3} & \frac{2b_{\overline{23}}}{18}\bar{x} & \frac{2b_{\overline{23}}}{18}\bar{y} \\
\frac{2b_{\overline{13}}}{3} & \frac{2b_{\overline{13}}}{18}\bar{x} & \frac{2b_{\overline{13}}}{18}\bar{y} & 0 & 0 & 0 & \frac{2c_{\overline{13}}}{3} & \frac{2c_{\overline{13}}}{18}\bar{x} & \frac{2c_{\overline{23}}}{18}\bar{y} \\
0 & 0 & 0 & \frac{2c_{\overline{13}}}{3} & \frac{2c_{\overline{13}}}{18} & \frac{2c_{\overline{23}}}{18}\bar{y} & \frac{2b_{\overline{13}}}{3} & \frac{2b_{\overline{13}}}{18}\bar{x} & \frac{2b_{\overline{13}}}{18}\bar{y}
\end{bmatrix} \qquad \dots (5.68)$$

$$
[G_e] = \frac{2A}{Et}\begin{bmatrix}
1 & \frac{\bar{x}}{6} & \frac{\bar{y}}{6} & -\nu & -\frac{\nu\bar{x}}{6} & -\frac{\nu\bar{y}}{6} & 0 & 0 & 0 \\
 & \left(\frac{\bar{x}}{6}\right)^2 & \frac{\overline{xy}}{36} & -\frac{\nu\bar{x}}{6} & -\nu\left(\frac{\bar{x}}{6}\right)^2 & -\nu\left(\frac{\overline{xy}}{36}\right) & 0 & 0 & 0 \\
 & & \left(\frac{\bar{y}}{6}\right)^2 & -\frac{\nu\bar{y}}{6} & -\nu\left(\frac{\overline{xy}}{36}\right) & -\nu\left(\frac{\bar{y}}{6}\right)^2 & 0 & 0 & 0 \\
 & & & 1 & \frac{\bar{x}}{6} & \frac{\bar{y}}{6} & 0 & 0 & 0 \\
 & & & & \left(\frac{\bar{x}}{6}\right)^2 & \frac{\overline{xy}}{36} & 0 & 0 & 0 \\
 & & & & & \left(\frac{\bar{y}}{6}\right)^2 & 0 & 0 & 0 \\
 & & & Sym & & & \bar{\nu} & \frac{\overline{\nu x}}{6} & \frac{\overline{\nu y}}{6} \\
 & & & & & & & \bar{\nu}\left(\frac{\bar{x}}{6}\right)^2 & \bar{\nu}\left(\frac{\overline{xy}}{36}\right) \\
 & & & & & & & & \bar{\nu}\left(\frac{\bar{y}}{6}\right)^2
\end{bmatrix}
$$

… (5.69)

Where $\bar{x} = x_1 + x_2 + x_3$, $\bar{y} = y_1 + y_2 + y_3$, $b_{\overline{12}} = b_1 + b_2$, $b_{\overline{13}} = b_1 + b_3$

$c_{\overline{12}} = c_1 + c_2$, $c_{\overline{23}} = c_2 + c_{3,}$, $c_{\overline{13}} = c_1 + c_3$ and $\bar{\nu} = 2 + 2\nu$.

5.8 IFM BASED IN-PLANE CURVED ELEMENT.

A rectangular element having two opposite edges with curved boundaries as shown in **Fig. 5.9** is approximated using IFM based two dimensional element and named as CURVE_5F_8D for having five independent force degrees of freedom and eight displacement degrees of freedom.

Procedure for deriving the basic equilibrium matrix is the same as all previous elements except coordinate system. The displacement function in u and v are selected as

$$u_{(r,\alpha)} = c_1 + c_2 r + c_3 \alpha + c_4 r\alpha$$

$$v_{(x,y)} = c_5 + c_6 r + c_7 \alpha + c_8 r\alpha \qquad \text{… (5.70)}$$

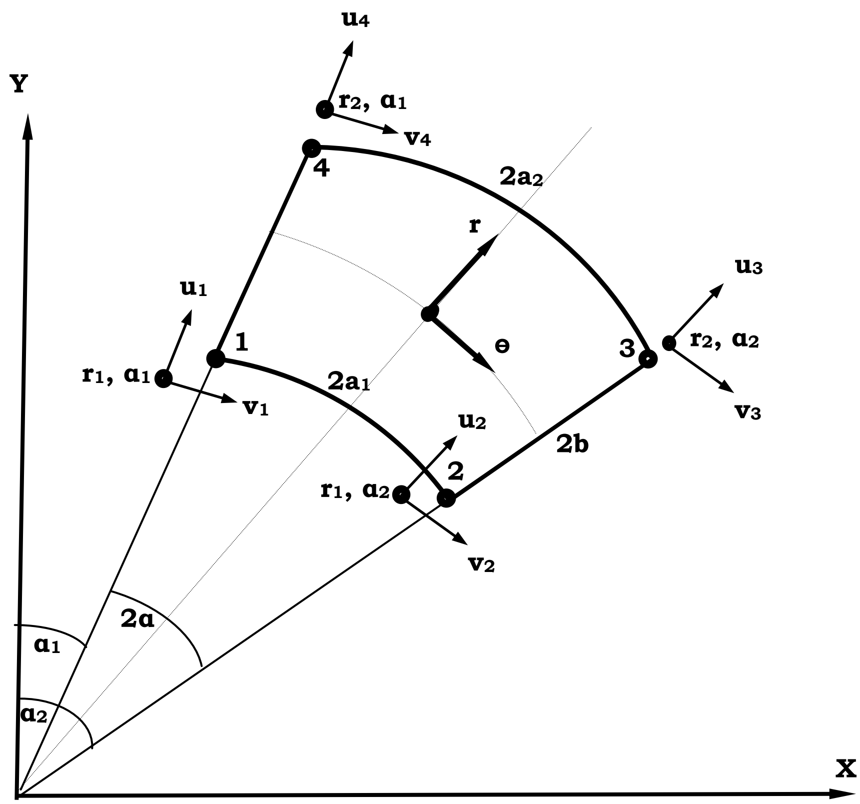


Fig. 5.9 Curved Element

The unknown nodal displacements are

$$\{\delta\} = [u_1 \; v_1 \ldots u_4 \; v_5]^T \qquad \ldots (5.71)$$

Arranging Eq. (5.52) in matrix form, one can write

$$\{\phi\} = \begin{Bmatrix} u_{(x,y)} \\ v_{(x,y)} \end{Bmatrix} = \begin{bmatrix} 1 & r & \alpha & r\alpha & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & r & \alpha & r\alpha \end{bmatrix} \begin{Bmatrix} c_1 \\ \cdot \\ \cdot \\ c_8 \end{Bmatrix}$$

$$\text{So, } \{\emptyset_r\} = [M_r]\{C\} \qquad \ldots (5.72)$$

Substituting the values of nodal coordinates.

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{bmatrix} 1 & r_1 & \alpha_1 & r_1\alpha_1 \\ 1 & r_1 & \alpha_2 & r_1\alpha_2 \\ 1 & r_2 & \alpha_2 & r_2\alpha_2 \\ 1 & r_2 & \alpha_1 & r_2\alpha_1 \end{bmatrix} \begin{Bmatrix} c_1 \\ c_2 \\ c_3 \\ c_4 \end{Bmatrix} = [M_1]\{C\} \qquad \ldots (5.73)$$

By inverting Eq. (5.55), one can have

$$\begin{Bmatrix} c_1 \\ c_2 \\ c_3 \\ c_4 \end{Bmatrix} = \frac{1}{Const}\begin{bmatrix} r_2\alpha_2 & -r_2\alpha_1 & r_1\alpha_1 & r_1\alpha_2 \\ -\alpha_2 & \alpha_1 & -\alpha_1 & \alpha_2 \\ -r_2 & r_2 & -r_1 & r_1 \\ 1 & -1 & 1 & -1 \end{bmatrix}\begin{Bmatrix} \mathrm{u}_1 \\ \mathrm{u}_2 \\ \mathrm{u}_3 \\ \mathrm{u}_4 \end{Bmatrix} \qquad \dots (5.74)$$

Where, $const = -r_2\alpha_1 + r_1\alpha_1 - r_1\alpha_2 + r_2\alpha_2$

Substituting {C} in Eq. (5.72), one can get

$$\{\mathrm{u}_{(\mathrm{r},\theta)}\} = [1 \quad r \quad \alpha \quad r\alpha]\frac{1}{Const}\begin{bmatrix} r_2\alpha_2 & -r_2\alpha_1 & r_1\alpha_1 & r_1\alpha_2 \\ -\alpha_2 & \alpha_1 & -\alpha_1 & \alpha_2 \\ -r_2 & r_2 & -r_1 & r_1 \\ 1 & -1 & 1 & -1 \end{bmatrix}\begin{Bmatrix} \mathrm{u}_1 \\ \mathrm{u}_2 \\ \mathrm{u}_3 \\ \mathrm{u}_4 \end{Bmatrix} \qquad \dots (5.75)$$

Now u and v can be expressed as

$$\mathrm{u} = \frac{1}{Const}\sum_{i=1}^{4} N_i\, u_i \text{ and } \mathrm{v} = \frac{1}{Const}\sum_{i=1}^{4} N_i\, v_i \qquad \dots (5.76)$$

Writing in expanded form, one can write

$$\{\phi\} = \begin{Bmatrix} \mathrm{u}_{(\mathrm{r},\theta)} \\ \mathrm{v}_{(\mathrm{r},\theta)} \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix}\begin{Bmatrix} \mathrm{u}_1 \\ \mathrm{v}_1 \\ \mathrm{u}_2 \\ \mathrm{v}_2 \\ \mathrm{u}_3 \\ \mathrm{v}_3 \\ \mathrm{u}_4 \\ \mathrm{v}_4 \end{Bmatrix} \qquad \dots (5.77)$$

Where $N_1 = \frac{1}{Const}(r_2\alpha_2 + r\alpha_2 - r_2\alpha + r\alpha)$, $N_2 = \frac{1}{Const}(-r_2\alpha_1 + r\alpha_1 + r_2\alpha - r\alpha)$

$$N_3 = \frac{1}{Const}(r_1\alpha_1 - r\alpha_1 - r_1\alpha + r\alpha),\ and\ N_4 = \frac{1}{Const}(-r_1\alpha_2 + r\alpha_2 + r_1\alpha - r\alpha)$$

... (5.78)

The strain vector can be written as

$$\{\epsilon\} = \begin{Bmatrix} \epsilon_r \\ \epsilon_\alpha \\ \nu_{r\alpha} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial u}{\partial r} \\ (\frac{u}{r} + \frac{1}{r}\frac{\partial v}{\partial \alpha}) \\ \frac{1}{2r}(\frac{\partial u}{\partial \alpha} + r\frac{\partial v}{\partial r} - v) \end{Bmatrix} =$$

$$\begin{bmatrix} \frac{\partial N_1}{\partial r} & 0 & \frac{\partial N_2}{\partial r} & 0 & \frac{\partial N_3}{\partial r} & 0 & \frac{\partial N_4}{\partial r} & 0 \\ N_1 & \frac{\partial N_1}{\partial \alpha} & N_2 & \frac{\partial N_2}{\partial \alpha} & N_3 & \frac{\partial N_3}{\partial \alpha} & N_4 & \frac{\partial N_4}{\partial \alpha} \\ \frac{\partial N_1}{\partial \alpha} & r\frac{\partial N_1}{\partial r} - N_1 & \frac{\partial N_2}{\partial \alpha} & r\frac{\partial N_2}{\partial r} - N_2 & \frac{\partial N_3}{\partial \alpha} & r\frac{\partial N_3}{\partial r} - N_3 & \frac{\partial N_4}{\partial \alpha} & r\frac{\partial N_4}{\partial r} - N_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = [Z]\{u\}$$

... (5.79)

The forcing function {F} with five numbers of internal unknowns can be written as

$\{F\}^T = [F_1\ F_2\ F_3\ .....F_5]^T$

Expressing internal stress distribution of the element in terms of independent unknowns, one can write

$$\begin{Bmatrix} \sigma_r \\ \sigma_\alpha \\ \tau_{r\alpha} \end{Bmatrix} = \begin{bmatrix} 1 & r & 0 & 0 & 0 \\ 0 & 0 & 1 & \alpha & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{Bmatrix} = [Y]\{F\} \qquad ... (5.80)$$

Where [Y] is known as stress interpolation matrix in which the variation of both the stresses along r and α is considered as linear.

Substituting strain linking matrix [Z] and stress interpolation function [Y] in the following equation gives the terms of [Be] matrix in which the non-zero terms are as follows;

$$[B_e] = \int_{\alpha_1}^{\alpha_2}\int_{r_1}^{r_2}[Z]^T[Y]r\,dr\,d\alpha \qquad ... (5.81)$$

$B_e(1,1) = \frac{\bar{r}}{4const}\bar{\alpha} - \frac{\alpha_2\bar{r}\,\alpha}{const}$, $\quad B_e(1,2) = \frac{\bar{r}}{6const}\bar{\bar{\alpha}} - \frac{\alpha_2\bar{r}\,\bar{\alpha}}{4const}$,

$B_e(1,3) = -\frac{r_2 b}{const} + \frac{\bar{r}}{2const}\bar{\alpha} - \frac{\bar{r}}{const}\alpha\alpha_2 + \frac{4b\alpha\alpha_2 r_2}{const}$,

$B_e(1,4) = \frac{\bar{r}_*}{2const} - r_2\frac{\overline{r\alpha}}{2const} - \frac{\bar{r}_*}{3const}2\alpha\alpha_2 + \bar{r}\frac{\alpha_2 r_2 2\alpha}{const}$

$B_e(1,5) = -\frac{2r_2 b\alpha}{const} + \frac{4\bar{r}}{4const}\alpha$, $B_e(2,3) = -\frac{4r_2\alpha b}{const}\bar{\alpha} + \frac{\bar{r}\alpha}{const}$, $B_e(2,4) = \frac{2\alpha\bar{r}_*}{3const} - \frac{r_2\bar{r}}{const}\alpha$

$B_e(2,5) = -\frac{2b\alpha r_2\alpha_2}{const} + r_2\frac{b\bar{\alpha}}{2const}$, $B_e(3,1) = -\frac{\overline{r\alpha}}{4const} + \frac{\alpha_1\alpha}{const}$

$B_e(3,2) = -\frac{\bar{r}_*\bar{r}}{6const} + \frac{\alpha_1\overline{r\alpha}}{4const}$, $B_e(3,3) = \frac{br_2\bar{r}}{const} - \frac{\bar{r}\,\bar{\alpha}}{2const} + \frac{b\alpha_1\bar{r}}{2const} - \frac{4b\alpha\alpha_1 r_2}{const}$

$B_e(3,4) = \frac{\bar{r}_*}{6const} + \frac{r_2\overline{r\alpha}}{2const}\bar{\alpha} + \frac{\bar{r}_*}{const}2\alpha - \frac{\alpha\bar{r}r_2}{const}$, $B_e(3,5) = \frac{2\alpha br_2\bar{r}_*}{const} - \frac{\alpha\bar{r}}{2const}$

$B_e(4,3) = \frac{4r_2 b\alpha}{const} - \frac{\alpha\bar{r}}{const}$, $B_e(4,4) = \frac{2\bar{r}_*\bar{r}}{3const} + \frac{2\alpha r_2}{const}$,

$B_e(4,5) = \frac{2\alpha br_2\alpha_1}{const} - \frac{2r_2 b\bar{r}}{4const}\bar{\alpha}$, $B_e(5,1) = \frac{\overline{r\alpha}}{4const} - \frac{\alpha_1\alpha\bar{r}}{const}$

$B_e(5,2) = \frac{\bar{r}\bar{\alpha}_*}{6const} - \frac{b\alpha r_1}{2const} + \frac{4\alpha_1\bar{r}}{4const}\alpha$, $B_e(5,3) = \frac{r_1}{2const} + \frac{\bar{\alpha}\bar{r}}{2const} - \frac{\alpha_1\alpha\bar{r}}{const} + \frac{\alpha b\alpha_1 r_1}{2const}$

$B_e(5,4) = \frac{\bar{r}_*}{6const} + \frac{r_1\bar{\alpha}\bar{r}}{2const} - \frac{\alpha\alpha_1 r_1}{2const} + \frac{\alpha\alpha_1 r_1\bar{r}}{2const}$, $B_e(5,5) = \frac{b\alpha r_1}{const} + \frac{\alpha\bar{r}}{2const}$

$B_e(6,3) = -\frac{4r_2\alpha b}{const}\bar{\alpha} + \frac{\bar{r}\alpha}{const}$, $B_e(6,4) = \frac{2\alpha\bar{r}_*}{3const} - \frac{r_1\bar{r}}{const}\alpha$

$B_e(6,5) = -\frac{2b\alpha r_2\alpha_2}{const} + r_1\frac{b\bar{\alpha}}{2const}$, $B_e(7,1) = -\frac{\overline{r\alpha}}{4const} + \frac{\alpha_2\alpha\bar{r}}{const}$

$B_e(7,2) = -\frac{\bar{r}\bar{\alpha}_*}{6const} + \frac{b\alpha r_1}{2const} + \frac{4\bar{r}}{4const}\alpha$, $B_e(7,3) = \frac{br_1}{const} - \frac{\bar{\alpha}\bar{r}}{2const} + \frac{\alpha_2\alpha\bar{r}}{const} + \frac{\alpha b\alpha_2}{const}$

$B_e(7,4) = \frac{\bar{r}_*}{6const} + \frac{r_1\bar{\alpha}\bar{r}}{2const} - \frac{\alpha\alpha_1 r_1}{2const} + \frac{\alpha\alpha_1 r_1\bar{r}}{2const}$, $B_e(7,5) = \frac{b\alpha r_1}{const} + \frac{\alpha\bar{r}}{2const}$

$B_e(8,3) = \frac{2\alpha br_1}{const} - \frac{\bar{\alpha}\bar{r}}{2const}$ $B_e(8,4) = -\frac{\alpha\bar{r}_*}{6const} + \frac{\alpha r_1\bar{r}}{const}$ $B_e(8,5) = \frac{2b\alpha\alpha_2}{const} - \frac{b\bar{\alpha}}{2const}$

Now the flexibility matrix [Ge] can be worked out by using the formula,

$$[G_e] = \int_{\alpha_1}^{\alpha_2}\int_{r_1}^{r_2}[Y]^T[D][Y]r\,dr\,d\alpha$$

$$[G_e] = \frac{1}{Et}\begin{bmatrix} \frac{2\alpha\bar{r}}{2} & \frac{\bar{r}\alpha}{4} & -\frac{2\alpha\bar{r}}{2} & -\frac{2\alpha\bar{r}_*\gamma}{3} & 0 \\ & \frac{\bar{r}\bar{\alpha}_*}{6} & \gamma\frac{\bar{r}\alpha}{4} & -\gamma\frac{\bar{\alpha}\bar{r}_*}{6} & 0 \\ & & \bar{r}\alpha & \frac{2\alpha\bar{r}_*\gamma}{3} & 0 \\ & sym & & \frac{\bar{\alpha}}{2}({r_2}^4 - {r_1}^4) & 0 \\ & & & & \alpha\overline{\gamma r} \end{bmatrix} \quad \ldots (5.82)$$

Where

$\bar{r} = {r_2}^2 - {r_1}^2, \bar{\alpha} = {\alpha_2}^2 - {\alpha_1}^2, r_2 - r_1 = 2b, \alpha_2 - \alpha_1 = 2\alpha, \bar{r_*} = {r_2}^3 - {r_1}^3,$

$\bar{\gamma} = 2(1+\gamma)$, $a_1 = r_1\alpha$ and $a_2 = (r_1 + 2b)\alpha$

5.9 IFM BASED PLATE BENDING ELEMENTS

5.9.1 RECT_9F_12D Plate Bending Element

Consider a typical rectangular plate element having size as 2a x 2b with its thickness t. The coordinate system and node numbering is as shown in **Fig. 5.7**, in which each node of the element has three displacement degrees of freedom i.e. transverse deflection $w_{(x,y)}$ and two orthogonal rotations (θ_x and θ_y). The displacements at node 1 are w_1, θ_{x1} and θ_{y1} with corresponding internal forces P_1, M_{x1} and M_{y1}. The displacement and internal force vectors are assumed as

$$\{\delta_e\} = [w_1, \theta_{x1}, \theta_{y1}, w_2, \theta_{x2}, \theta_{y2}, w_3, \theta_{x3}, \theta_{y3}, w_4, \theta_{x4}, \theta_{y4}]^T \quad \ldots (5.83)$$

$$\{P_e\} = [P_1\ M_{x1}\ M_{y1}, P_2\ M_{x2}\ M_{y2}, P_3\ M_{x3}\ M_{y3}, P_4\ M_{x4}\ M_{y4}]^T \quad \ldots (5.84)$$

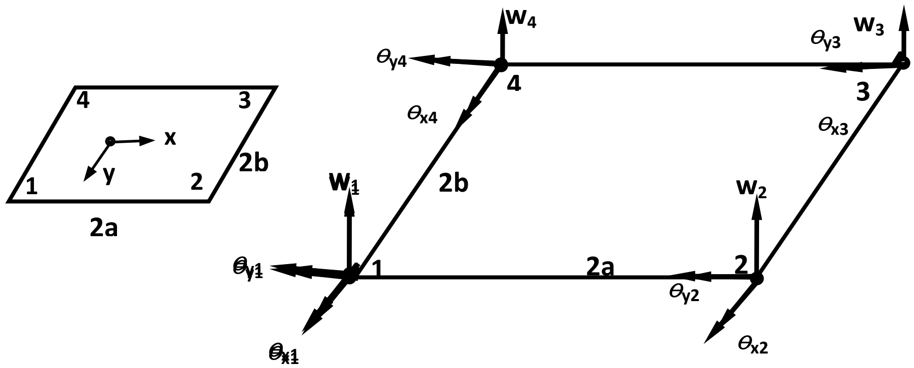


Fig. 5.6 Rectangular Plate Bending element

For a two dimensional rectangular plate bending element having 3 degrees of freedom at each node (w. θ_x, θ_y), the lateral deflection w can be expressed as

$$w_{(x,y)} = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 xy + \alpha_5 x^2 + \alpha_6 y^2 + \alpha_7 x^2 y + \alpha_8 xy^2 + \alpha_9 x^3 + \alpha_{10} y^3 + \alpha_{11} xy^3 + \alpha_{12} xy^3 \quad \ldots (5.85)$$

$$= [A]\{\alpha\}$$

where, [A] is known as function of (x, y) matrix and α_s are the constants.

The above displacement function however may not satisfying an inter-element compatibility along common boundaries (non-conformal element) but with finer discretization pattern it may provide the solution for moments and deflections within acceptable limits.

Referring to Fig. 5.6 and Eq. (5.85) and substituting nodal coordinates one can write

$$\{d\} = [A_1]\{\alpha\} \quad \ldots (5.86)$$

Where $\{d_e\}$ = Vector of displacement degree of freedom of size 12 x 1.

$[A_1]$ = Generalized coordinate matrix of size 12 x 12.

and $\{\alpha\}$ = Vector of constants of size 12 x 1.

Inverting $[A_1]$ using matlab based inverting module and substituting in (Eq.5.85), once can write

$$w_{(x,y)} = [A][A_1]^{-1}\{\delta_e\} = [N]\{\delta_e\} \qquad \ldots (5.87)$$

Or $\quad w_{(x,y)} = [N]\{\delta_e\} = \sum_{i=1}^{4}[N_{i(x,y)w_i} + N_{i(x',y)\theta_{xi}} + N_{i(x'',y'')\theta_{yi}}]$

In which, [N] is known as shape function matrix for rectangular plate bending problem of size (1 x 12). In which first three values corresponding to deflection $w_{(x,y)}$ in vertical direction and rotations θ_x and θ_y along x-x and y-y direction respectively are given below.

$$N_{1(x,y)} = \frac{9}{32} - \frac{9x}{16a} - \frac{13y}{32b} - \frac{x^2}{32a^2} + \frac{11xy}{16ab} + \frac{5x^3}{16a^3} + \frac{x^2y}{32a^2b} + \frac{y^3}{8b^3} - \frac{5x^3y}{16a^3b} - \frac{xy^3}{8ab^3}$$

$$N_{1'(x,,y)} = \frac{a}{8} - \frac{x}{4} - \frac{ay}{8b} - \frac{x^2}{8a} + \frac{xy}{b} + \frac{x^3}{4a^3} + \frac{x^2y}{ab} - \frac{x^3y}{4a^3b}$$

$$N_{1''(x,y)} = \frac{5y}{32} - \frac{x^2}{32a^2} + \frac{3ay}{16a} - \frac{y^2}{8b} + \frac{x^2y}{32a^2} + \frac{xy^2}{8ab} + \frac{y^3}{8b^2} - \frac{x^3y}{16a^3} - \frac{xy^3}{8ab^3} \qquad \ldots (5.88)$$

Now, the strain vector, which consists of curvature terms can be expressed as

$$\{\epsilon\} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \nu_{xy} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial^2 w(x,y)}{\partial x^2} \\ \frac{\partial^2 w(x,y)}{\partial y^2} \\ 2\frac{\partial^2 w(x,y)}{\partial x \partial y} \end{Bmatrix} = \begin{Bmatrix} \frac{\partial^2}{\partial x^2}[N_{1(x,y)}, N_{1(x'y)} \cdots \quad N_{4(x'y)}] \\ \frac{\partial^2}{\partial y^2}[N_{1(x,y)}, N_{1(x'y)} \cdots \quad N_{4(x'y)}] \\ 2\frac{\partial^2}{\partial x \partial y}[[N_{1(x,y)}, N_{1(x'y)} \quad N_{4(x'y)}] \end{Bmatrix} \begin{Bmatrix} w_1 \\ \theta_{x1} \\ \theta_{y1} \\ w_2 \\ \theta_{x2} \\ \theta_{y2} \\ w_3 \\ \theta_{x3} \\ \theta_{y2} \\ w_4 \\ \theta_{x4} \\ \theta_{y4} \end{Bmatrix}$$

$$= [Z]\{\delta_e\} \qquad \ldots (5.89)$$

Where [Z] = [L][N], is known as strain linking matrix and [L] is flexural strain operator.

The forcing function {F} in IFM is selected for bending element based on number of rigid body modes. As each strain component has its own rigid body component, number of independent unknowns in forcing function {F} = Total displacement degrees of freedom (ddof) – Number of rigid body modes (3) = 9

So, $\{F\}^T = [F_1\ F_2\ F_3\ .....F_9]^T$ ... (5.98)

Expressing internal moments in terms of independent unknown, one can write

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & xy & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & x & y & xy & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \\ F_9 \end{Bmatrix} = [Y]\{F\} \qquad \text{... (5.90)}$$

Where, [Y] is known as stress interpolation matrix. The element has bilinear variation for the orthogonal moments M_x and M_y while constant relation for torsional moment M_{xy}.

The behavior of Mx can be understood in orthogonal direction with respect to origin (0, 0) by substituting approximating values at each corner and at midpoint of each edge of the element

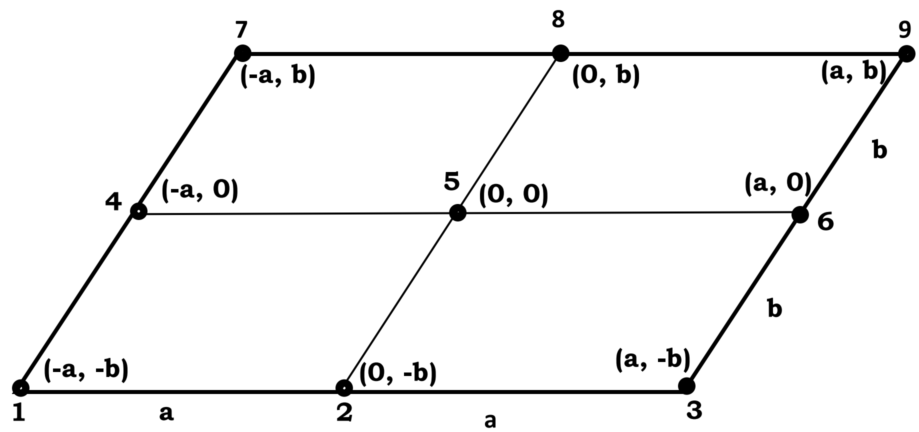


Fig. 5.7 Coordinates for expressing Variation of Mxx

Referring to **Fig 5.7** the values of Mx at each different nodes can be written as

$M_1 = F_1 - aF_2 - bF_3 + abF_4,\quad M_2 = F_1 - bF_3\,,\quad M_3 = F_1 + aF_2 - bF_3 - abF_4,$

$M_4 = F_1 - aF_2\,,\quad M_5 = F_1,\quad M_6 = F_1 + aF_2$

$M_7 = F_1 - aF_2 + bF_3 - abF_4\quad M_8 = F_1 + bF_3$

and $M_9 = F_1 + aF_2 + bF_3 + abF_4$ … (5.91)

Here M_9 has the maximum value compared to all other values.

The equilibrium matrix $[B_e]$ for the given plate bending element is obtained using

$$[B_e] = \int_{-a}^{+a}\int_{-b}^{+b}[Z]^T\,[Y]\,dxdy \qquad \dots (5.92)$$

Substituting flexural strain displacement linking matrix [Z] from Eq. (5.89) and stress interpolation matrix [Y] from Eq. (5.90) and integrating one can get the equilibrium matrix $[B_e]$ as

$$[B_e] = \begin{bmatrix} -\frac{b}{4a} & \frac{5b}{2} & \frac{b^2}{12a} & \frac{-5b^2}{6} & 0 & 0 & a & \frac{a^2}{3} & 2 \\ -b & 2ab & \frac{b^2}{3} & \frac{-2b^2a}{3} & 0 & 0 & 0 & 0 & 0 \\ -\frac{b^2}{4a} & \frac{b^2}{2} & \frac{b^3}{12a} & -\frac{b^2}{6} & -a & \frac{a^2}{3} & ab & -\frac{a^2b}{3} & 0 \\ -\frac{b}{4a} & -\frac{5}{2}b & \frac{b^2}{-12a} & -\frac{5b^2}{6} & 0 & 0 & a & \frac{a^2}{3} & -2 \\ b & 2ab & -\frac{b^2}{3} & \frac{-2b^2a}{3} & 0 & 0 & 0 & 0 & 0 \\ \frac{b^2}{4a} & -\frac{b^2}{2} & -\frac{b^3}{12a} & \frac{b^3}{3} & -a & -\frac{a^2}{3} & ab & \frac{a^2b}{3} & 0 \\ -\frac{b}{4a} & -\frac{3}{2}b & \frac{b^3}{12a} & \frac{-5b^2}{6} & 0 & 0 & -a & -\frac{a^2}{3} & 2 \\ b & 2ab & \frac{b^2}{3} & \frac{2b^2a}{3} & 0 & 0 & 0 & 0 & 0 \\ \frac{b^2}{4a} & -\frac{b^2}{2} & \frac{b^3}{-12a} & \frac{b^2}{6} & a & \frac{a^2}{3} & ab & \frac{a^2b}{3} & 0 \\ -\frac{b}{4a} & \frac{3}{2}b & -\frac{b^3}{12a} & \frac{5b^2}{6} & 0 & 0 & -a & \frac{a^2}{3} & -2 \\ -b & 2ab & -\frac{b^2}{3} & \frac{2b^2a}{3} & 0 & 0 & 0 & 0 & 0 \\ -\frac{b^2}{4a} & -\frac{b^2}{2} & -\frac{b^3}{12a} & -\frac{b^3}{6} & a & -\frac{a^2}{3} & ab & -\frac{a^2b}{3} & 0 \end{bmatrix} \qquad \dots (5.93)$$

Now the flexibility matrix for plate bending element can be obtained by substituting stress interpolation matrix [Y] and material matrix [D] in the Eq. (5.49) as

$$[G_e] = \frac{48ab}{Et^3}\begin{bmatrix} 1 & 0 & 0 & 0 & -\nu & 0 & 0 & 0 & 0 \\ & \frac{a^2}{3} & 0 & 0 & 0 & -\nu\frac{a^2}{3} & 0 & 0 & 0 \\ & & \frac{b^3}{3} & 0 & 0 & 0 & -\nu\frac{b^3}{3} & 0 & 0 \\ & & & \frac{a^2b^3}{9} & 0 & 0 & 0 & -\nu\frac{a^2b^3}{9} & 0 \\ & & & & 1 & 0 & 0 & 0 & 0 \\ & & & & & \frac{a^2}{3} & 0 & & 0 \\ & & Symm & & & & \frac{b^3}{3} & 0 & 0 \\ & & & & & & & \frac{a^2b^3}{9} & 0 \\ & & & & & & & & 2(1+\nu) \end{bmatrix}$$

… (5.94)

5.9.2 RECT_13F_16D Plate Bending Element

When C^1 continuity has to be satisfied, both displacement and its first derivatives are taken as nodal degrees of freedom and Hermitian Interpolation functions can be used. The order of function depends on order of derivatives used for the degrees of freedom. If only the first derivatives are used, then first order function are used.

For a n-noded beam element with w and its first derivative $\frac{\partial w}{\partial x}$ as degree of freedom at each node, the Hermitian function can be written as

$$w = \sum_{i=1}^{n}[\, N_i\,(x)w_i + N_i^{'}(x)\frac{\partial w_i}{\partial x}\,]$$

Where shape function corresponding to w and $\frac{\partial w}{\partial x}$ can be calculated respectively by

$$N_i = \prod_{j=1, j\neq i}^{n} \left(\frac{x - x_j}{x_i - x_j}\right)^2 \left[1 + 2\sum_{j=1}^{n} \frac{x_i - x}{x_i - x_j}\right]$$

$$N'_i = \prod_{j=1, j\neq i}^{n} \left(\frac{x - x_j}{x_i - x_j}\right)^2 (x - x_i) \qquad \ldots (5.95)$$

For a two noded beam element of length 2a and origin of coordinates at the centre of the element, these shape functions are as follows:

$$N_{1(x)} = \frac{x^3 - 3a^3x + 2a^3}{4a^3}$$

$$N_{1(x)} = \frac{-x^3 + 3a^3x + 2a^3}{4a^3}$$

$$N_{1'(x)} = \frac{x^3 - ax^3 - a^3x + a^3}{4a^2}$$

$$N_{1'(x)} = \frac{x^3 + ax^2 - a^2x + a^3}{4a^2} \qquad \ldots (5.96)$$

If product of these shape functions in x and y are used for a two dimensional rectangular element with four corner nodes, one gets 16 shape function in all. The nodal degree of freedom at any node will be w, $\frac{\partial w}{\partial x}$ and their derivatives in y i.e. $\frac{\partial w}{\partial y}$ and $\frac{\partial^2 w}{\partial x \partial y}$ which is shown in **Fig 5.8.**

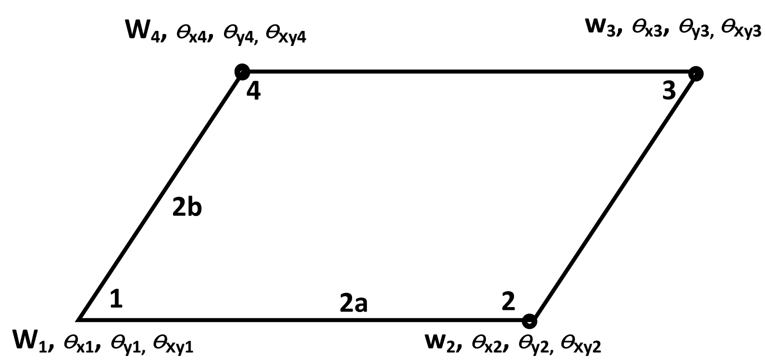


Fig. 5.8 Degrees of Freedom for Higher Order Element

For a four noded rectangular element Hermitian Interpolation function can be written as

$$w = \sum_{i=1}^{n} [\, N_{i(x,y)}\, w_i + N_{i'(x,y)} \frac{\partial w_i}{\partial x} + N_{i''(x,y)} \frac{\partial w_i}{\partial y} + N_{i'''(x,y)} \frac{\partial^2 w}{\partial x \partial y} \qquad \ldots (5.97)$$

where $N_{i(x,y)} = N_{i(x)}\, N_{i(y)}$, $N_{i'(x,y)} = N_{i'(x)}\, N_{i(y)}$, $N_{i''(x,y)} = N_{i(x)}\, N_{i'(y)}$

and $N_{i'''(x,y)} = N_{i'(x)}\, N_{i'(y)}$ … (5.98)

for example, for node 1 the shape functions will be

$$N_{i(x,y)} = \frac{x^3-3a^3x+2a^3}{4a^3}\;\frac{y^3-3b^3y+2b^3}{4b^3}$$

$$N_{1'(x,y)} = \frac{x^3-ax^2-a^2x+a^3}{4a^2}\;\frac{y^3-3b^3y+2b^3}{4b^3}$$

$$N_{i''(x,y)} = \frac{x^3-3a^3x+2a^3}{4a^3}\;\frac{y^3-by^2-b^2y+b^3}{4b^2}$$

$$N_{i'''(x,y)} = \frac{x^3-ax^2-a^2x+a^3}{4a^2}\;\frac{y^3-by^2-b^2y+b^3}{4b^2}$$

Similarly, shape function for the other nodes can be written and finally w can be expressed in terms of these functions as per Eq. (5.97).

Now strains can be related to nodal displacements as

$$\{\epsilon\} = \begin{Bmatrix}\epsilon_x\\ \epsilon_y\\ \nu_{xy}\end{Bmatrix} = \begin{Bmatrix}\frac{\partial^2 w(x,y)}{\partial x^2}\\ \frac{\partial^2 w(x,y)}{\partial y^2}\\ 2\frac{\partial^2 w(x,y)}{\partial x \partial y}\end{Bmatrix} = \begin{Bmatrix}\frac{\partial^2}{\partial x^2}[N_{1(x,y)}, N_{1'(x,y)} \cdots \quad N_{4'''(x,y)}]\\ \frac{\partial^2}{\partial y^2}[N_{1(x,y)}, N_{1'(x,y)} \cdots \quad N_{4'''(x,y)}]\\ 2\frac{\partial^2}{\partial x \partial y}[[N_{1(x,y)}, N_{1'(x,y)} \ldots \quad N_{4'''(x,y)}]\end{Bmatrix}\begin{Bmatrix}w_1\\ \theta_{x1}\\ \theta_{y1}\\ \theta_{xy1}\\ \cdot\\ \cdot\\ \cdot\\ w_4\\ \theta_{x4}\\ \theta_{y4}\\ \theta_{xy5}\end{Bmatrix}$$

$$= [Z]\{\delta_e\}$$ … (5.99)

Where [Z] = [L][N] is known as strain displacement linking matrix and [L] is flexural strain operator.

The forcing function {F} is selected for bending element based on number of rigid body modes. As each and every strain component has its own rigid body component inside, the number of independent unknowns in forcing function {F}

= Total displacement degrees of freedom (ddof) – Number of rigid body modes (3) = 13.

So, $\{F\} = [F_1\ F_2\ F_3\ \ldots\ldots F_{13}]^T$ … (5.100)

Expressing internal stress distribution of the given element in terms of independent unknown, one can write

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & xy & x^2 & y^2 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & xy & x^2 & y^2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ \cdot \\ \cdot \\ F_{13} \end{Bmatrix} = [Y]\{F\}$$

… (5.101)

Where, [Y] is known as stress interpolation matrix, in which x and y are the distances from the centre of element.

The Basic elemental equilibrium matrix $[B_e]$ for the given plate bending element can be obtained by

$$[B_e] = \int_{-a}^{+a} \int_{-b}^{+b} [Z]^T\,[Y]\,dxdy \qquad \ldots (5.101)$$

in which flexural strain linking matrix [Z] is taken from Eq. (5.99) and stress interpolation matrix [Y] from Eq. (5.101) respectively. After integrating, the equilibrium matrix $[B_e]$ is obtained as

$$[B_e] =$$

$$\begin{bmatrix}
0 & b & 0 & \frac{-2b^2}{5} & 0 & 0 & 0 & 0 & a & \frac{-2a^2}{5} & 0 & 0 & 2 \\
-b & 2ab & \frac{2b^2}{5} & \frac{-2b^2a}{3} & -\frac{a^2b}{3} & -\frac{b^3}{3} & 0 & 0 & \frac{a^2}{3} & -\frac{a^3}{15} & 0 & 0 & 0 \\
0 & \frac{b^2}{3} & 0 & -\frac{b^3}{15} & 0 & 0 & a & -\frac{2a^2}{5} & ab & \frac{-2a^2b}{3} & -\frac{a^3}{3} & \frac{-b^2a}{3} & 0 \\
-\frac{b^2}{3} & \frac{b^2a}{3} & \frac{b^3}{15} & \frac{-2a^3b}{15} & \frac{-2b^2a^2}{9} & -\frac{b^4}{15} & -\frac{a^2}{3} & \frac{a^3}{15} & \frac{a^2b}{3} & \frac{-a^3b}{15} & -\frac{a^4}{15} & \frac{-b^2a^2}{9} & 0 \\
0 & -b & \frac{2b^2}{5} & \frac{-2b^2a}{5} & \frac{a^2b}{3} & \frac{b^2}{3} & 0 & 0 & \frac{a^2}{3} & \frac{a^3}{15} & 0 & 0 & 0 \\
b & ab & \frac{2b^3}{5} & \frac{2ab^2}{5} & \frac{a^2b}{3} & \frac{b^3}{3} & 0 & 0 & \frac{a^2b}{3} & \frac{a^3}{15} & 0 & 0 & 0 \\
0 & \frac{b^2}{3} & 0 & \frac{b^3}{15} & 0 & 0 & a & \frac{2a^2}{5} & ab & \frac{2a^3b}{5} & \frac{a^2}{3} & \frac{b^2a}{3} & 0 \\
-\frac{\mathbf{b^2}}{\mathbf{3}} & \frac{-b^2a}{3} & -\frac{b^3}{15} & -\frac{b^3a}{15} & \frac{-a^2b^2}{9} & -\frac{b^4}{15} & -\frac{a^3}{3} & -\frac{a^3}{15} & 0 & -\frac{a^2b}{3} & \frac{-a^3b}{15} & \frac{-b^2a^2}{9} & 0 \\
0 & -b & 0 & -\frac{2b^2}{5} & 0 & 0 & 0 & 0 & 0 & -\frac{2a^3}{5} & 0 & 0 & 2 \\
b & ab & \frac{2b^3}{5} & \frac{2b^2a}{5} & \frac{a^2b}{3} & \frac{b^3}{3} & 0 & 0 & \frac{a^3}{3} & \frac{a^3}{15} & 0 & 0 & 0 \\
0 & \frac{b^2}{3} & 0 & \frac{b^3}{15} & 0 & 0 & a & \frac{2a^2}{5} & ab & \frac{2a^2b}{5} & \frac{a^3}{3} & \frac{a^2b}{3} & 0 \\
-\frac{b^2}{3} & -\frac{ab^2}{3} & -\frac{b^3}{15} & -\frac{ab^3}{15} & \frac{-a^2b^2}{9} & -\frac{b^2}{15} & -\frac{a^3}{3} & -\frac{a^3}{15} & -\frac{a^2b}{3} & -\frac{a^3b}{15} & -\frac{a^4}{15} & \frac{-a^2b^2}{9} & 0 \\
0 & b & 0 & -\frac{2b^2}{5} & 0 & 0 & 0 & 0 & -a & \frac{2a^2}{5} & 0 & 0 & -2 \\
-b & ab & \frac{2b^3}{5} & \frac{2ab^2}{5} & -\frac{a^2b}{3} & \frac{b^3}{3} & 0 & 0 & -\frac{a^2}{3} & \frac{a^3}{15} & 0 & 0 & 0 \\
0 & \frac{b^2}{3} & 0 & -\frac{b^3}{15} & 0 & 0 & a & \frac{2a^2}{5} & ab & \frac{2a^2b}{5} & \frac{a^2}{3} & \frac{b^2a}{3} & 0 \\
-\frac{\mathbf{b^2}}{\mathbf{3}} & \frac{-b^2a}{3} & -\frac{b^3}{15} & -\frac{b^3a}{15} & \frac{-a^2b^2}{9} & -\frac{b^4}{15} & -\frac{a^3}{3} & -\frac{a^3}{15} & \frac{a^2b}{3} & -\frac{a^3b}{15} & \frac{a^4}{15} & \frac{a^2b^2}{9} & 0
\end{bmatrix} \quad \ldots (5.102)$$

The flexibility matrix for this element can be worked out by substituting stress interpolation matrix [Y] and material matrix [D] in the Eq. (5.49), which is found as as below

$$[G_e] = G' \begin{bmatrix} 1 & 0 & 0 & 0 & \frac{a^2}{3} & \frac{b^2}{3} & -\nu & 0 & 0 & 0 & \frac{-\nu a^2}{3} & \frac{-\nu b^2}{3} & 0 \\ & \frac{a^2}{3} & 0 & 0 & 0 & 0 & 0 & \frac{\nu a^2}{3} & 0 & 0 & 0 & 0 & 0 \\ & & \frac{b^2}{3} & 0 & 0 & 0 & 0 & 0 & \frac{\nu b^2}{3} & 0 & 0 & 0 & 0 \\ & & & \frac{a^2 b^2}{9} & 0 & 0 & 0 & 0 & 0 & \frac{\nu a^2 b^2}{9} & 0 & 0 & 0 \\ & & & & \frac{a^4}{5} & \frac{a^2 b^2}{9} & \frac{\nu a^2}{3} & 0 & 0 & 0 & \frac{\nu a^4}{5} & \frac{\nu a^2 b^2}{9} & 0 \\ & & & & & \frac{b^4}{5} & \frac{b^3}{3} & 0 & 0 & 0 & \frac{\nu a^2 b^2}{9} & -\frac{b^4}{5} & 0 \\ & & Symm & & & & 1 & 0 & 0 & 0 & \frac{a^2}{3} & \frac{b^3}{3} & -\nu \\ & & & & & & & \frac{a^2}{3} & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & & \frac{b^2}{3} & 0 & 0 & 0 & 0 \\ & & & & & & & & & \frac{a^2 b^2}{9} & 0 & 0 & 0 \\ & & & & & & & & & & \frac{a^4}{5} & \frac{a^2 b^2}{9} & \frac{\nu a^2}{3} \\ & & & & & & & & & & & \frac{b^4}{5} & \frac{b^3}{3} \\ & & & & & & & & & & & & 2.6\nu \end{bmatrix} \quad \ldots (5.103)$$

Where, $G' = \frac{48ab}{Et^3}$

CHAPTER 6

ENVIRONMENT SELECTED FOR THE DEVELOPMENT

6.1 PREAMBLE

Generally speaking, the programming language is a language code designed to communicate schematically as per user requirement to computer. The instructions are given to computer in terms of various functions, procedures, subroutines and various types of modules. User friendly programming languages with flexible type Graphical User Interface (GUI) based input data mainly succeed in competence of other type of programming language. Least number of input data, successful connection with data bases needed with minimum linking procedure are also important factors for good programming language. All these necessary information are easily developed by the Visual Programming Language, which mainly consists of powerful GUI based input editor. Some of the important features of VB6, VB.NET and Matlab software which are necessary for understanding the procedure adopted for finding the solution are described in the subsequent sections.

6.2 VISUAL BASIC AS PROGRAMMING ENVIROMENT

Visual Basic 6 is designed to provide all the essential information which is needed to create user friendly and sophisticated programs. It provides a graphical environment in which one can visually design the forms and controls using the coding which becomes the building blocks to develop applications. VB's integrated development environment (IDE) is the term commonly used in the programming world to describe the interface which is used to create the applications. It is called integrated, because one can access virtually all of the development tools which are needed from the bunch, called an interface.

6.2.1 General Features

Following are some of the features **(Fig. 6.1)** and inbuilt functions, which are used in the development of program in the present work.

1. **Form**: The most basic object is the form object, which is the visual foundation of any application. It is basically a window that can add different elements in order to create a complete application. Every application is based on some type of form. Some features of form are, border, title bar caption, control menu, minimize and maximize button.
2. **MDI form**: MDI stand for Multiple Document Interface. With the MDI option, all of the IDE windows are contained within a single resizable parent window. One can easily switch between the two modes simultaneously.

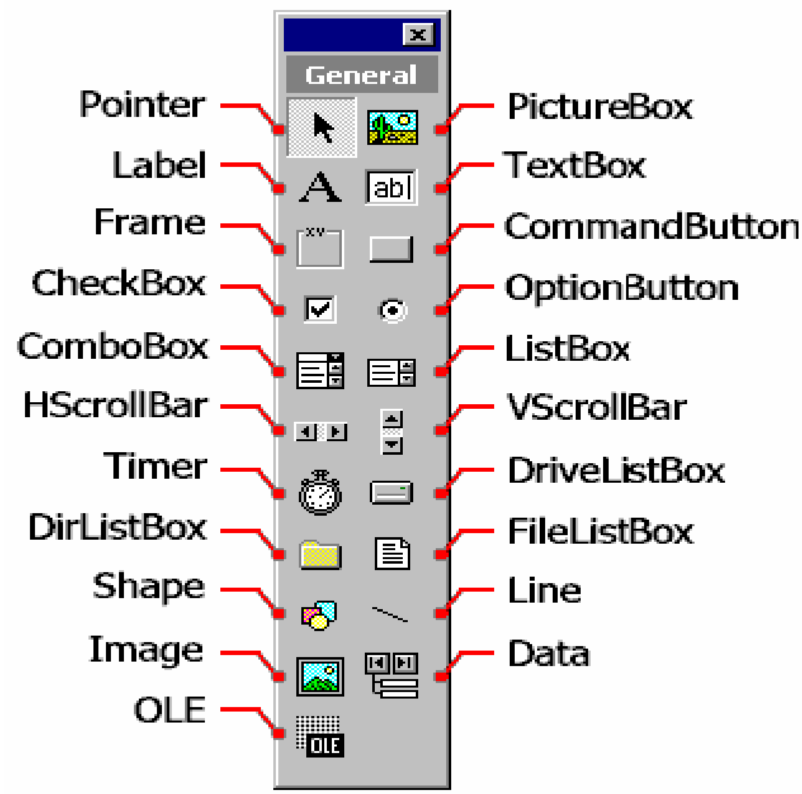


Fig. 6.1 Standard Toolbar

3. **Command button**: It is used to execute the event. One can write a code by clicking on the command button.
4. **Text box**: Each and every VB project involves a text box control. Text boxes are commonly used for accepting user input or for entering data
5. **Label**: A label control is similar to a text box control, in that both display text. The main difference, however, a label displays read-only text, though one can alter the caption as a run-time property.
6. **Image object**: The image control is an equivalent of the picture box control. But unlike the picture control, the image control can't act as a container for other objects, but it is a good choice. It simply displays on the form as picture.
7. **Option button**: Option button control is also called as radio buttons which are used to allow a user to select one, and only one option. Usually option buttons are grouped together within a frame control.
8. **Check box**: A check box control is rather similar to an option button. But, the fundamental difference between check boxes and option button is the check box allows multiple selections in a single form or frame.
9. **Frame control**: This has the effect of grouping the controls such as option buttons and check boxes, so that when the frame is moved, the other controls also move simultaneously.
10. **Timer**: The timer is one of the few controls always hidden at run time, means it doesn't have to find room for it on a form. The timer basically does just one thing; it checks the system clock and acts accordingly.
11. **Immediate window**: Immediate window is used to test the prepared application interactively. From this, one can launch procedures, view and change the values of variables, and evaluate expressions and to print information without any intervention. This technique works for both functions and subroutines.
12. **SSTAB control:** It is like the dividers in a notebook or the labels on a group of file folders. Using an SSTab control, one can define multiple

pages for the same area of a window or dialog box in the application. Using the properties of this control,

- Determine the number of tabs.
- Organize the tabs into more than one row.
- Set the text for each tab.
- Display a graphic on each tab.
- Determine the style of tabs used.
- Set the size of each tab.

13. **Microsoft FlexGrid (MSFlexGrid) control:** It displays and operates on tabular data. It allows complete flexibility to sort, merge, and format tables containing strings and pictures. When bound to a **Data** control, MSFLEXGrid display only read data.

14. **Types of conditional statements:** An application needs a built-in capability to test conditions and take a different course of actions depending on the outcome of the test. Visual Basic provides three decision structures, which are as follows;
    a. If…Then
    b. If…Then…Else
    c. Select Case.

15. **Summary of looping statements**: The general forms of loops used in the present work are:

a) For…Next — For *variable identifier* = *start value* To *end value*

  statement(s) = body of loop

 Next *variable identifier*

b) Do While…Loop — Do While *condition*

statement(s) = body of loop

Loop

c) Do…Loop Until   Do

statement(s) = body of loop

Loop Until condition.

16. **Variable scope:** All procedures, variables, constants, etc. have a particular scope. This scope determines when and from where one can access the procedures, variable, or constants which depends upon where and how one declares it. Following variables are generally required in the development of a program.

   a. Procedure variables: Variables that are only available within the procedure in which they are declared are called local or **procedure variable.** One may declare such a variable using either the Static or Dim keyword.

   b. Module variables: The availability of variables that declared at module level depends upon how they are declared. If one declares variables with Dim or Static keyword then the scope of that variable is within that module. If variables are declared with Public key word then they are available in all procedures.

   c. Global Variables: If one needs variable declared in a module to be accessible within the whole application, one can use Public keyword.

   These variables are then available to every module and procedure within the application.

VB is based upon an event-driven paradigm, in which each feature included within the program is activated only when the user responds to a corresponding object (i.e., an icon, a check box, option button, a menu selection, etc.) within the user interface. The program's response to an action taken by the user is referred to as an event. The group of basic commands that brings about this response is called as event procedure.

6.2.2 Benefits of Using VB Environment

- Programming with VB is accomplished visually. While writing the program, programmer can see how his program will look during run time. This is a great advantage over other programming languages.
- Programmer is free to change and experiment with his design until he is satisfied with features like color, size and images based on requirements.
- VB supports several types of modules that contain declarations; event procedures and various supporting information for their respective forms and controls.
- VB also supports sub procedures, function procedures, and property procedures.
- The immediate window is very useful when debugging a project. By entering a variable or expression within this window, one can see the corresponding value immediately.

6.3 VB.NET AS PROGRAMMING ENVIRONMENT

Visual Basic .NET provides the easiest, most productive language and tool for rapidly building Windows and Web applications. It basically comes with enhanced visual designers, increased application performance, and a powerful integrated development environment (IDE). In addition to these features, it also consists with a powerful new forms designer, an in-place menu editor, and automatic control anchoring and docking. VB.NET also delivers a new productivity features for building more robust applications easily and quickly. With an improved and a significantly reduced startup time, VB.NET offers fast, automatic formatting of code. By using VB.NET one can create Web applications using the shared Web Forms Designer and the familiar "drag and drop" feature. One can double-click and write code to respond to events. VB.NET comes with an enhanced HTML Editor for working with complex Web pages.

6.3.1 General Features of VB.NET

1. **Form**: Form is a window, in which one can add different elements in order to create a complete application. Every application is based on development of form. Some features of form are: border, title bar caption, control menu, minimize and maximize button.
2. **MDI form**: Using MDI option, all of the windows are contained within a single resizable parent window. One can easily switch between the two modes simultaneously.
3. **Button**: It is used to execute all the events. One can start the coded event by clicking the command button.
4. **Text box**: Generally each and every VB project consist of a text box control, which are commonly used for accepting user input for entering data.
5. **Label**: A label is a control similar to a text box in which both display text. The main difference is that, it displays read-only text as far as the user is concerned even though one can alter the caption at a run-time.
6. **Radio Button**: Option button control is also called radio buttons used to allow the user to select only one from a group of options. Usually all option buttons are grouped together within a frame control.
7. **Check box**: A check box control is similar to an option button. But, the fundamental difference between check box and option button is that the check box allows multiple selections in a single form or frame.
8. **Timer**: The timer is one of the few controls which is always hidden at run time. It checks the system clock and acts accordingly.

9. **Frame control**: This has a effect of grouping the number of controls, such as radio buttons and check boxes. When the frame is moved, the other controls move simultaneously.
10. **The Menu Strip:** It displays application commands and option groups by functionality.

In VB.NET, the tool box contains the tools that can be used to place various controls on forms. It displays all the standard Visual Basic Controls plus any custom controls and objects that are added to project with the Custom Controls dialog box and there are a number of basic tools in the Toolbox that can be also used to create user interfaces.

6.3.2 Connection of VB.6/.NET with MATLAB 7.4

Basically, a database consists of one or more large complex files that store data in a structured format and VB is ideal for managing data, since it has built-in database engine. Data stored by VB can also be used by other applications like Microsoft Access or Word.

VB's database library is called ADO.net. It uses a disconnected data model. This means that the database connection is only used when retrieving or updating data. Operations like navigating through the data, or even adding and changing records, can all be done without going back to the source database.

(A) **Working with Access:** VB and Microsoft Access make excellent partners. User can use Access to created databases, set up validation rules, and do interactive editing, while using visual basic to develop a packaged application. A Microsoft Access database file, which has an extension of .mdb, contains tables, queries, forms, reports, pages, macros, and modules, which are referred to as database objects. Forms, reports, pages, macros, and modules are generally concerned with letting users work with and display data.

(B) Working with Client Server: VB supports several database technologies, including ADO.NET, OLEDB and ODBC. ODBC stands for open database connectivity. To connect to a particular database, one needs a suitable ODBC driver. Most major database managers have suitable drivers, like Oracle to open-source alternatives like SQL. User can connect to server database systems like SQL Server, Oracle or DB2. This is the most complex type of database programming, but is necessary for good performance on large networks.

6.3.3 Benefits of VB.NET Framework

1. VB.NET provides managed code execution that runs under the Common Language Runtime, resulting in robust, stable and secure applications. All features of the .NET framework are readily available in VB.NET.
2. VB.NET is totally object oriented which is a major additional feature.
3. The .NET framework comes with ADO.NET, which follows the disconnected paradigm, i.e. once the required records are fetched the connection no longer exists. It also retrieves the records that are expected to be accessed in the immediate future. This enhances Scalability of the application to a great extent.
4. VB.NET uses XML to transfer data between the various layers in the DNA Architecture i.e. data are passed as simple text strings.
5. Error handling has changed in VB.NET. A new Try-Catch-Finally block has been introduced to handle errors and exceptions as a unit, allowing appropriate action to be taken at the place the error occurred thus discouraging the use of ON ERROR GOTO statement.
6. Another important feature added to VB.NET is free threading against the VB single-threaded apartment feature. In many situations developers need spawning of a new thread to run as a background process and increase the usability of the application. VB.NET allows developers to spawn threads wherever they feel like, hence giving freedom and better control on the application.

7. Security has become more robust in VB.NET. In addition to the role-based security in VB 6, VB.NET comes with a new security model, Code Access security. For example one can set the security to a component such that the component cannot access the database.
8. The CLR takes care of garbage collection i.e. the CLR releases resources as soon as an object is no more in use. This relieves the developer from thinking of ways to manage memory. CLR does this for them.

6.4 MATLAB AS NUMERICAL AND GRAPHICAL PROCESSOR

MATLAB is a high performance computing environment which allows easy matrix manipulation, plotting of functions, data implementation of algorithm, creation of user interface and interfacing with programs in other language. Using Matlab one can work out simulation and modeling for verifying experimental results. It also facilitates graphical displays having one of the best GUI facility.

MATLAB is an interactive system whose basic data element is an array that does not require dimensioning. This allows one to solve many technical computing problems, especially those with matrix and vector formulations, in a fraction of the time it would take to write a program in a scalar non interactive language such as VB 6, VB.NET, C, C++ or Fortran.

6.4.1 Connection of VB 6/.NET with MATLAB 7.4

The development of compatibility condition in IFM is carried out using Matlab software. It requires Matlab interfacing in VB.Net environment for further numerical work. Thus, linking or integrating techniques helps in development of separate working library for accessing Matlab functionalities. Linking of VB6 with Matlab is feasible by following two approaches.

a. **COM Approach**: In this approach Matlab expresses itself as integral part of a COM Automation Server. In running coding line of VB6, Matlab is considered as object. By clicking reference of the project button of main menu, different linking library can be seen from which Matlab Automation Server Type Library is selected using check box. Thus, the connection between VB6 and Matlab is established generated which can be seen using **Figs. 6.2 and 6.3.**

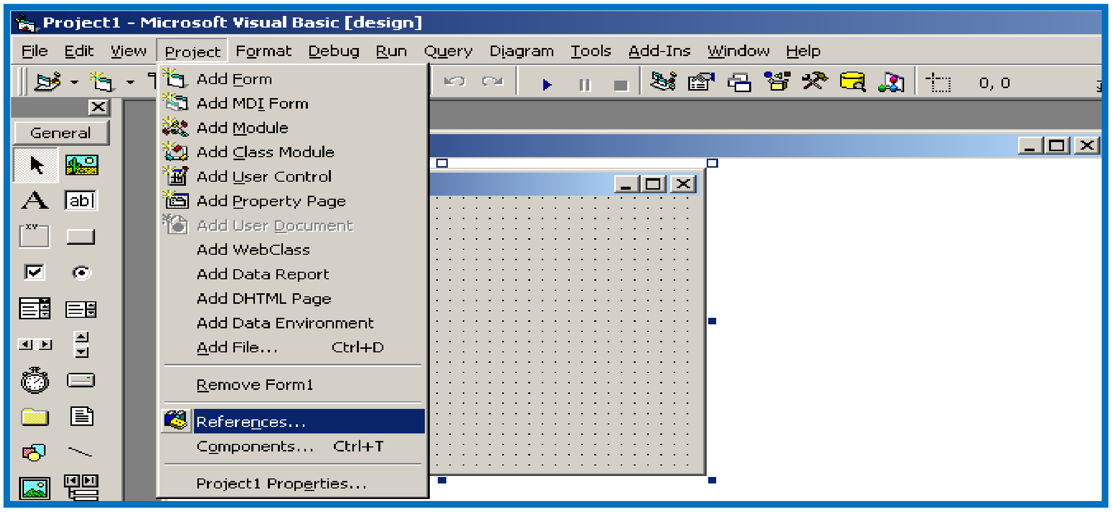


Fig. 6.2 Main Menu Showing Reference of VB 6

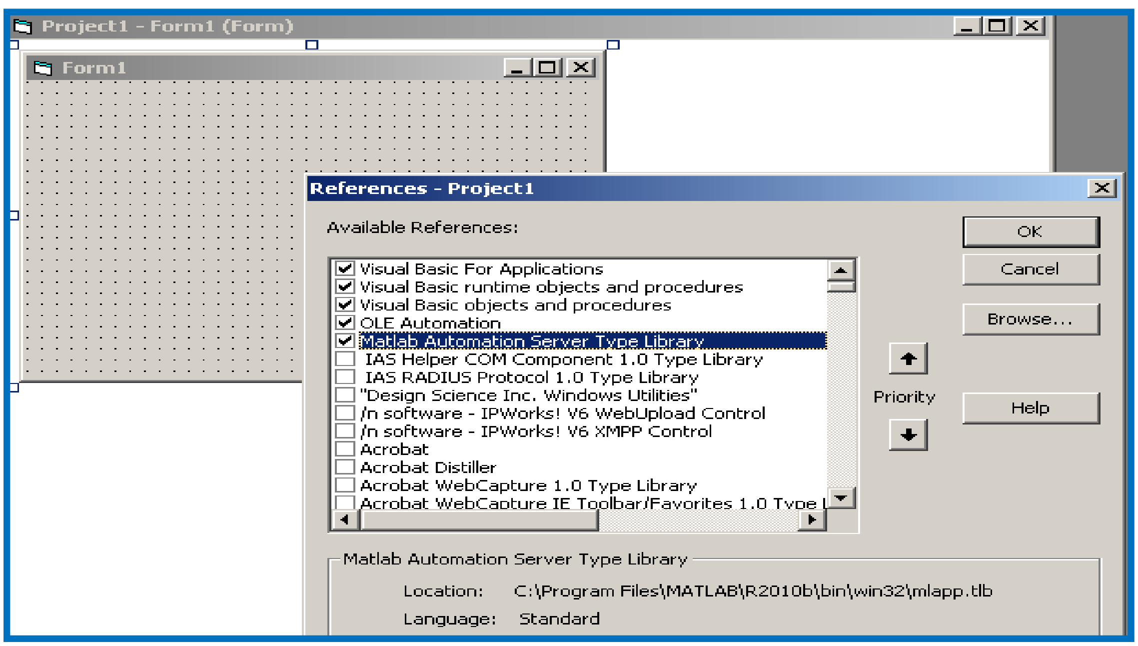


Fig. 6.3 Clicking of Matlab Automation Server Type Library

By clicking view of main menu as per **Fig 6.4** of the given project work, for an object browser, MLapp is selected for Matlab Application purpose, which is referred as Matlab.Application.

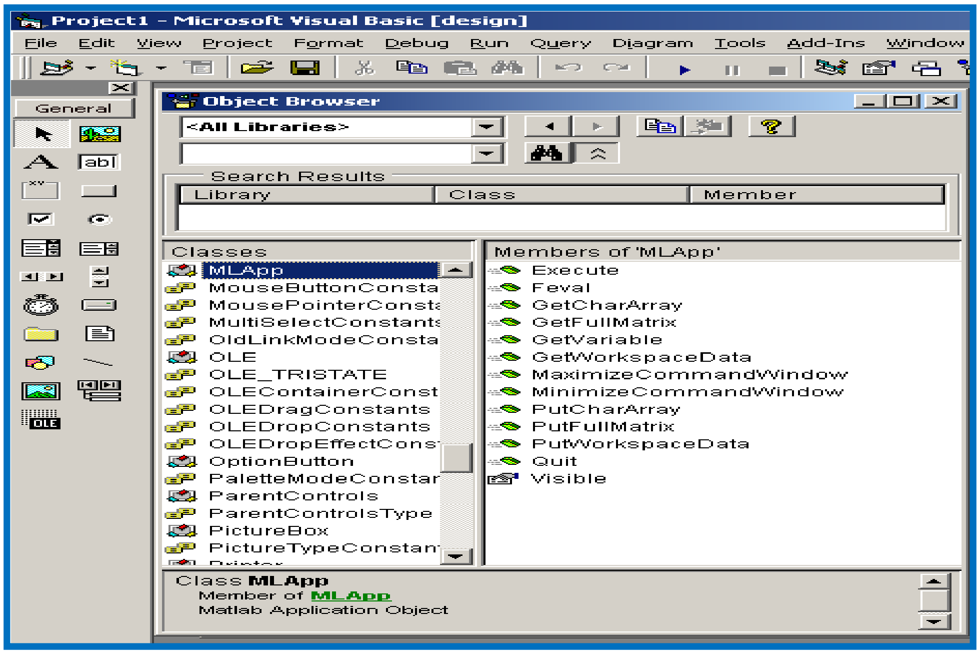


Fig 6.4 Selection of MLApp for Matlab Application

Once the connection is ready using clicking above features, the Matlab being considered as server is enabled by handle object on matlab screen i.e. by typing h = actxserver('Matlab.Application') as shown in **Fig. 6.5.** It gives separate matlab command window on which by typing 'enable service('Automation Server') gives number 1 as response of valid connection procedure, as depicted in **Fig. 6.6.**

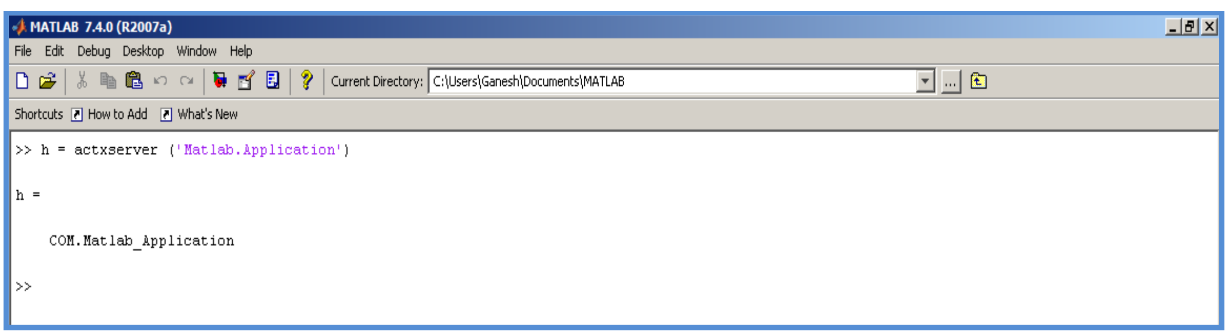


Fig 6.5 Handling Procedure Matlab in COM Application

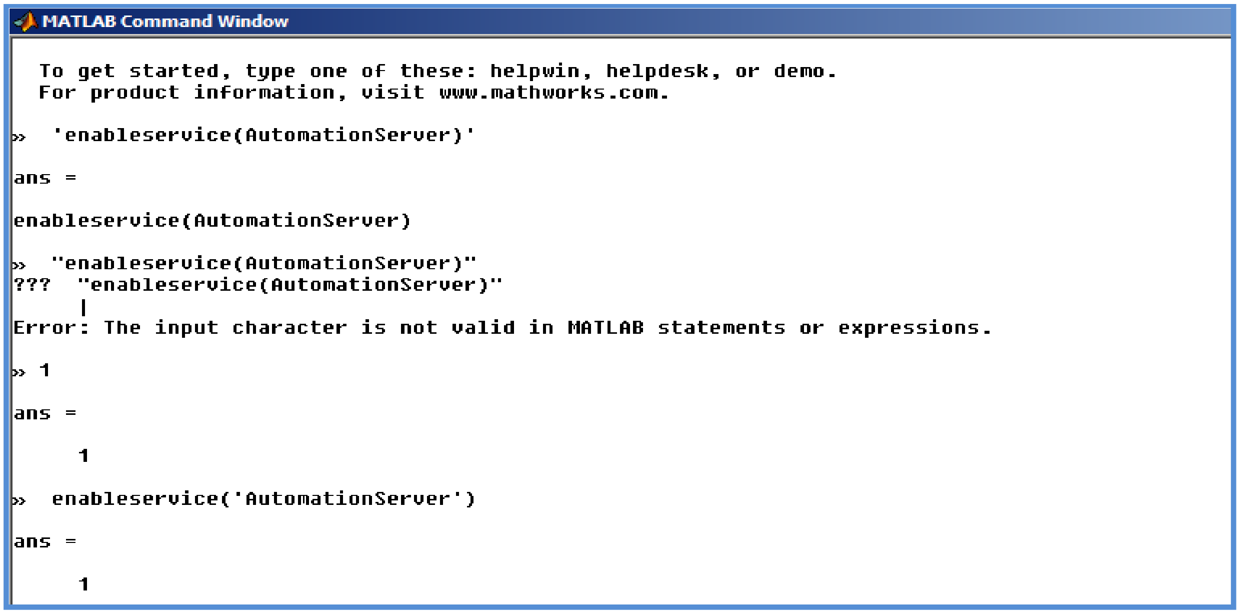


Fig 6.6 Enabling service of Automation Server

b. **DDE Approach**: The Dynamic Data Exchange is quite powerful service of the Windows based applications which enables the communication and exchange of data. It is the ancestor of the whole COM application. Here also the client is considered as .NET program and the server as the DDE Server, which communicates. The DEE communication is made with the service name, like Matlab or WinWord, about a topic of discussion, like system or Engine and based on the exchange of data elements called items. The main limitation of the DDE Exchange is the data types are of the Window Clipboard data types.

6.4.2 Autogeneration of Compatibility Conditions

The Compatibility Conditions (CCs) are the backbone of the IFM based analysis of indeterminate structures. LIUT based mathematical procedure for selecting coefficients of deformations vector {β}, which gives [C]{β} = 0 irrespective of size or problem is the main core concept of IFM based procedure. In LIUT technique mathematically out of all the coefficients required few of them are kept constants and by using trial and error procedure others are modified till the product of developed coefficients [C] and deformation vector {β} equals to zero. This procedure is repeated depending upon requirement of number of compatibility conditions which makes the complete global equilibrium matrix [B] a square one by concatenating force based compatibility conditions at the bottom. The complete procedure is readily developed through Matlab based programming module named as "mtechexamplemod(B)", where [B] is the equilibrium matrix to be supplied based on which the [C] matrix is to be generated. This procedure is illustrated here with help of an example.

1. Open Matlab Command window and change the directory CCPROG for desktop position, where the module "mtechexamplemod" is available as shown in **Fig. 6.7**.

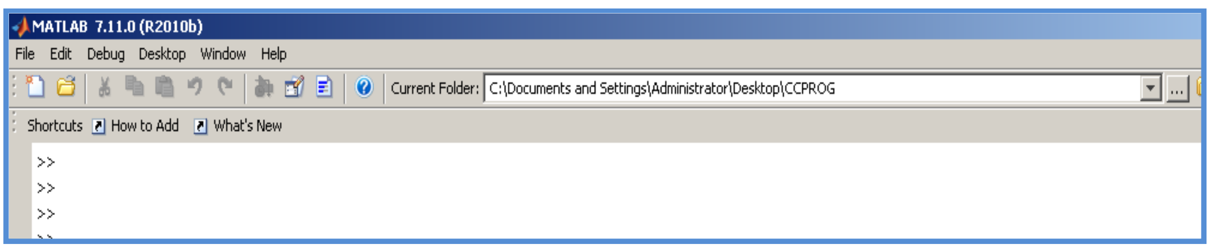


Fig 6.7 Open Matlab Command Window for Changing Directory

2. Supply [B] matrix of size 3 x 4 at command prompt in Matlab window as shown in **Fig. 6.8.**

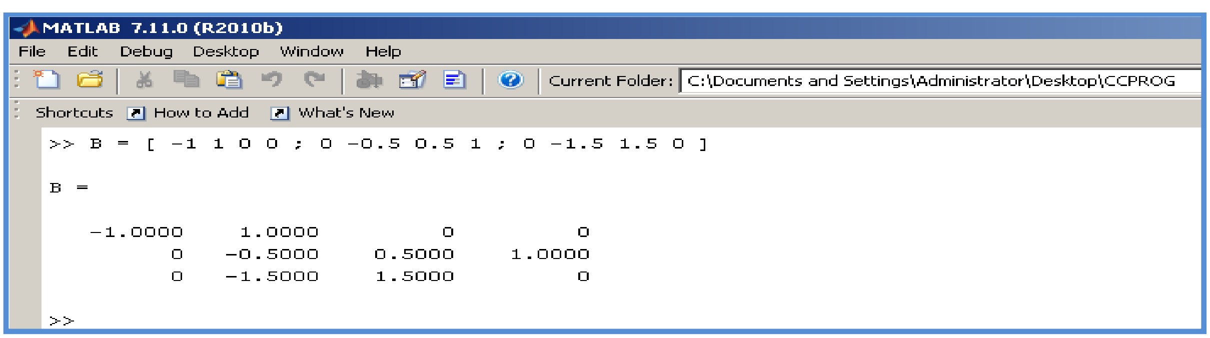


Fig. 6.8 Entering [B] Matrix of Size 3 x 4

3. Type z = mtechexamplemod(B) at command prompt which asks for number of conditions required. Out of four number of unknowns β_1,

β_3 and β_4 are selected and denoted as "codeindB" as independent and β_2 as "codedepB" as dependent unknown as shown in **Fig. 6.9.**

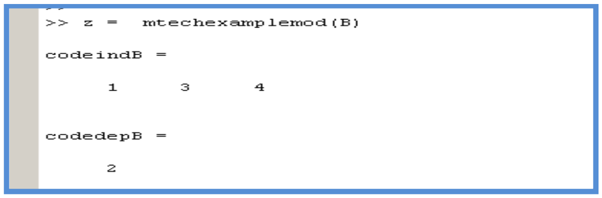


Fig. 6.9 CodeindB and CodedepB

4. **Figure 6.10** shows number of compatibility conditions required, thus, it gives "alphadep" as coefficient of β_2 and "alphaind" as coefficients of β_1, β_3, β_4.

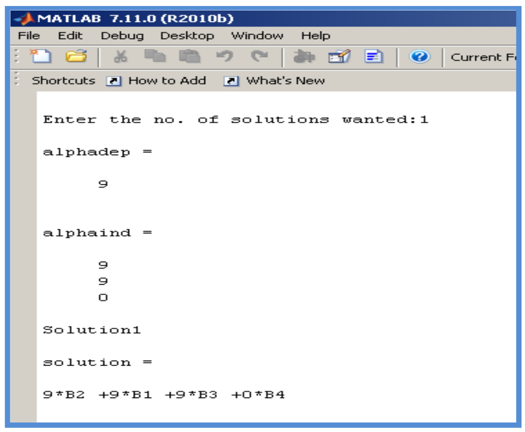


Fig 6.10 Coefficients of alphadepB and alphaind

5. **Figure 6.11** shows [C] matrix of size 1 x 4 which is a coefficient matrix of deformation vector {β} and is named as z.cMatrix. The mathematical verification of null property is done by multiplication of [C] with $[B]^T$ and is worked out as z.cTransposeB in Matlab.

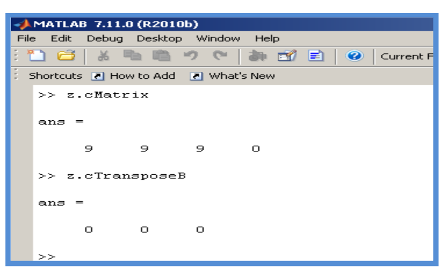


Fig 6.11 [C] matrix and its Verification

6.4.3 Development of Shape function using Matlab

Consider a Beam element having length L with two displacement degrees of freedom at each end. The displacement function w(x) is expressed in terms of generalized coordinates as follows

$$w_{(x)} = \alpha_1 + \alpha_2 x + \alpha_3 x^2 + \alpha_4 x^3 \qquad \text{... (6.1)}$$

which is converted to matrix form as

$$w_{(x)} = [1 \; x \; x^2 \; x^3] \begin{bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \end{bmatrix} = [A]\{\alpha\} \qquad \text{... (6.2)}$$

Complete $[A_1]$ matrix is developed by substituting values of x = 0 and x = L in $w_{(x)}$ and $w'_{(x)}$. The further mathematical operations are developed directly in Matlab Command Editor as shown in **Figs. 6.12~6.15**.

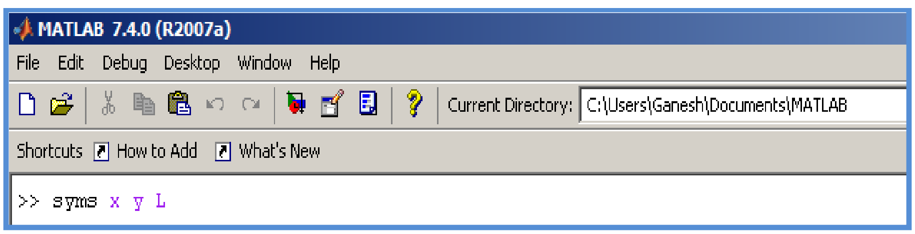


Fig. 6.12 Defining Variables x, y and L

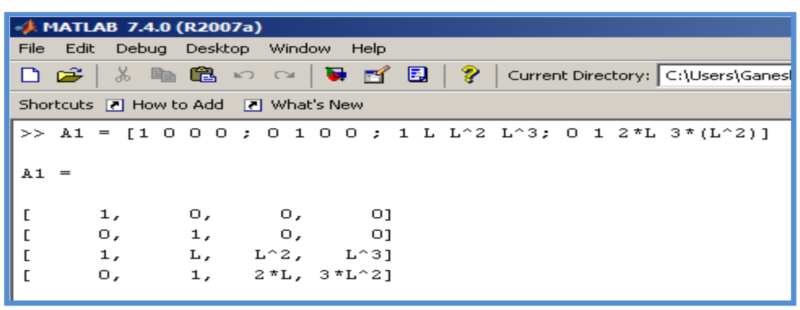


Fig. 6.13 Coefficient Matrix [A1] of size 4 x 4

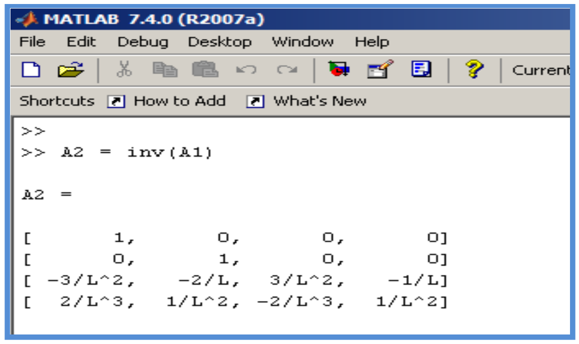


Fig. 6.14 Calculation of $[A_2] = [A_1]^{-1}$

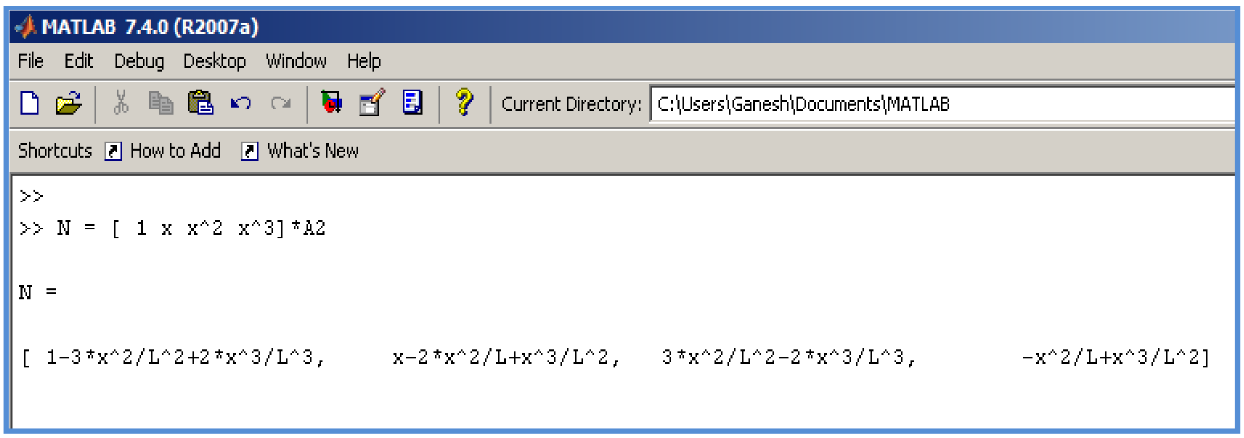


Fig. 6.15 Development of Shape Function [N]

6.4.4 Development of 2D Moments Contours

A simply supported plate 4000 mm x 4000 mm is considered here for illustration purpose. The left bottom quadrant is selected for drawing moment contours, which is discretized into total 25 numbers of elements with 36 nodal points as depicted in **Fig 6.16.** The moments are calculated at each node and expressed here as value calculated in the box of the given mesh gridlines as shown in **Fig. 6.17.** i.e. along line AB being simply supported edge all moments are zero, along line EF the values are (0.0, 101.446, 175.55, 219.382, 236.958, 231.931) and so on.

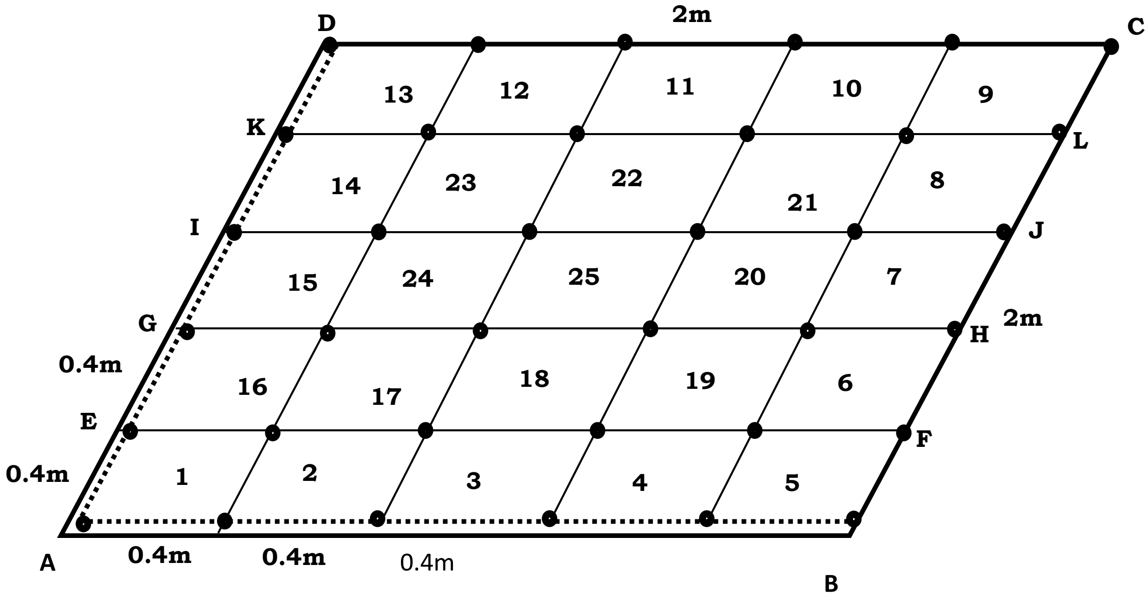


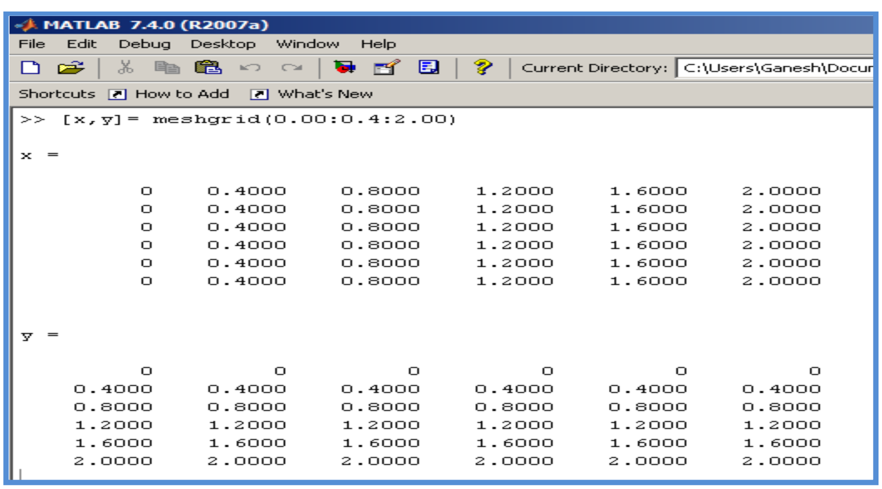
Fig 6.16 Numbering of Nodes and Elements of 5x5 Discretization

A	0	0	0	0	0	0	B
E	0	101.446	175.5593	219.382	236.958	231.931	F
G	0	175.559	320.8282	412.512	451.312	445.074	H
I	0	219.382	412.516	552.758	620.634	722.00	J
K	0	236.958	451.312	620.634	739.736	765.284	L
D	0	231.931	445.07	722.00	765.284	783.438	C

Fig. 6.17 Corresponding Position of Nodal Moments

Following are the steps for plotting of moments contours:

1. Using mesh grid function on Matlab editor, grid for total 36 nodes is developed by using function given in **Fig. 6.18**.




```
>> [x,y]= meshgrid(0.00:0.4:2.00)

x =

         0    0.4000    0.8000    1.2000    1.6000    2.0000
         0    0.4000    0.8000    1.2000    1.6000    2.0000
         0    0.4000    0.8000    1.2000    1.6000    2.0000
         0    0.4000    0.8000    1.2000    1.6000    2.0000
         0    0.4000    0.8000    1.2000    1.6000    2.0000
         0    0.4000    0.8000    1.2000    1.6000    2.0000


y =

         0         0         0         0         0         0
    0.4000    0.4000    0.4000    0.4000    0.4000    0.4000
    0.8000    0.8000    0.8000    0.8000    0.8000    0.8000
    1.2000    1.2000    1.2000    1.2000    1.2000    1.2000
    1.6000    1.6000    1.6000    1.6000    1.6000    1.6000
    2.0000    2.0000    2.0000    2.0000    2.0000    2.0000
```

Fig. 6.18 Development of Mesh Grid

2. Nodal moments indicated in **Fig. 6.19** are directly taken to form [Z] matrix of size 6x6 from MS Excel sheet, which are moment values in terms of z ordinates at grid points at an interval of 0.4m as depicted in **Fig 6.20**

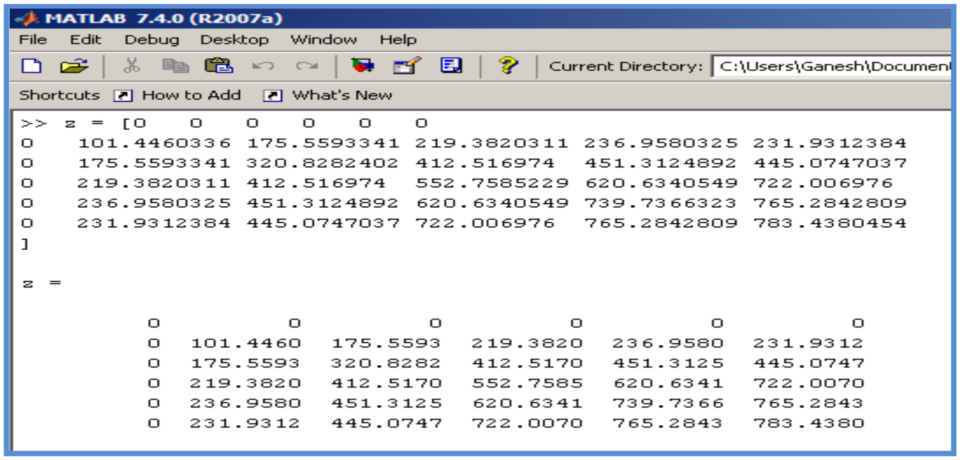


Fig. 6.20 [Z] Matrix

3. To generate contours, type [c, h] = contour(x, y, z, 10) where c stands for contour, h for handle function of matlab, while (x, y, z, 10) denotes all three dimensional cartesian coordinates with total 10 number of contours ranging from min and max values of [Z] matrix as depicted in **Fig. 6.21.**

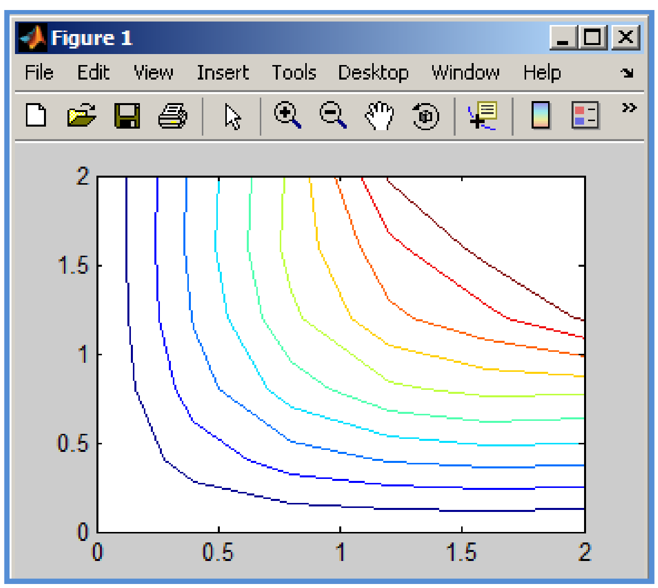


Fig. 6.21 Contour Plot for Moment

4. The labeling procedure is carried out by command “clabel (c, h)” as depicted in **Fig. 6.22.** The labeling along x-x and y-y axis is carried out by executing few functions from Tools of the Matlab editor.

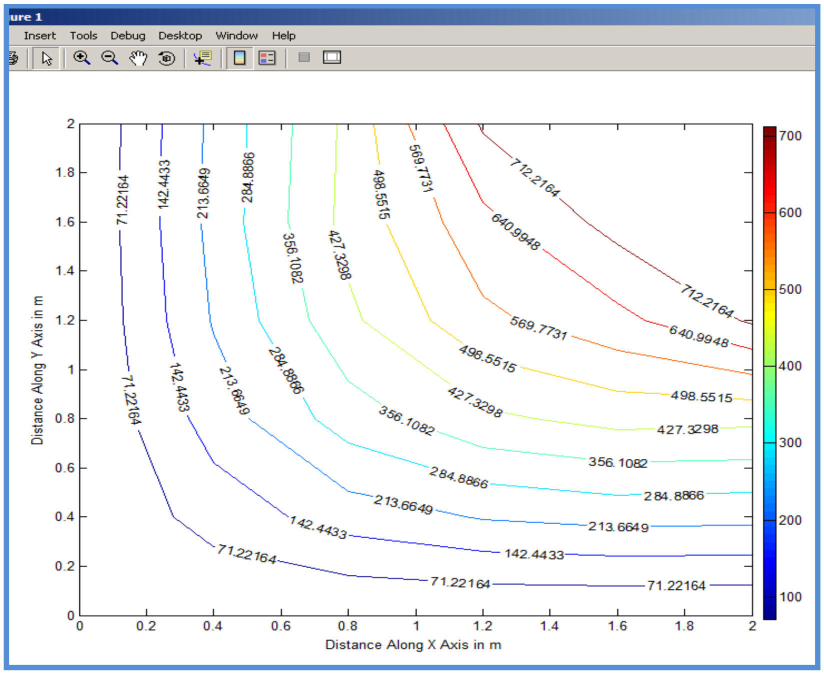


Fig. 6.22 Contour Plot for Moment

6.4.5 Plotting of 3D Deformation Pattern

The procedure for plotting 3D deflection pattern is illustrated here for a quarter plate depicted in **Fig 6.17.** Following 2 steps are required for plotting the elastic curve.

1. [Z] matrix for nodal deflection values at each node is generated and copied on matlab editor as depicted in **Fig. 6.23.**

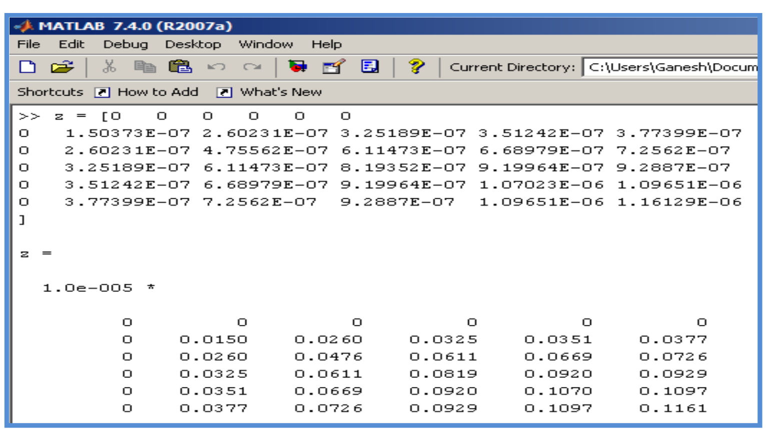


Fig. 6.23 [Z] Matrix For Deformed Shape

2. Type surf(x, y, -z) on matlab editor to plot deflection pattern of quarter plate as shown in **Fig. 6.24** for nodal deflection values specified in [Z] matrix.

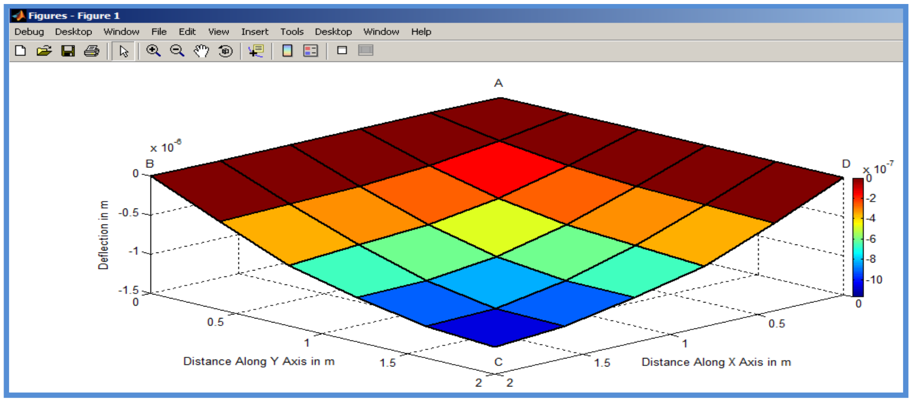


Fig. 6.24 Deflected Shape for Quarter Plate

CHAPTER 7

STATIC ANALYSIS OF FRAMED STRUCTURES

7.1 COMPUTER IMPLEMENTATION OF IFM

Computer implementation of force based IFM and displacement based DIFM methods are carried out using VB.NET and Matlab software. All the steps required for finding the solutions are programmed using systematic GUI based forms and procedures. Various forms are developed for supplying input data related to loading and geometrical information. Elemental equilibrium matrix [B] and Global flexibility matrix [G] are developed and saved in text file for use in further calculations. Special .m file is created for generating the compatibility conditions, based on equilibrium conditions, using MatLab software's having inbuilt LIUT (Linear Independent Unknown Technique) function. The global compatibility conditions are developed by carrying out the product of compatibility matrix [C] and Global Flexibility Matrix [G]. Concatenation of basic equilibrium matrix and the global compatibility matrix gives the global equilibrium matrix which is managed here using Matlab based command window directly. The load vector {P} is developed and called using text file. Solution of IFM based generated equations provides internal unknown matrix {F} which is carried out using direct inv module of Matlab. The nodal displacements are calculated by taking necessary matrices from VB.NET and Matlab command window directly. The final answer can be viewed on one of the forms created using of VB.NET.

The complete solution procedure for IFM is illustrated here with the help of a continuous beam example through graphical screen shots.

A three span prismatic continuous beam having three spans as 3m, 4m and 5m respectively as shown in **Fig. 7.1** is loaded with uniformly distributed loading of 10kN/m throughout. Calculate internal moments and nodal displacements by using value of EI as 1666.67 kN-m^2.

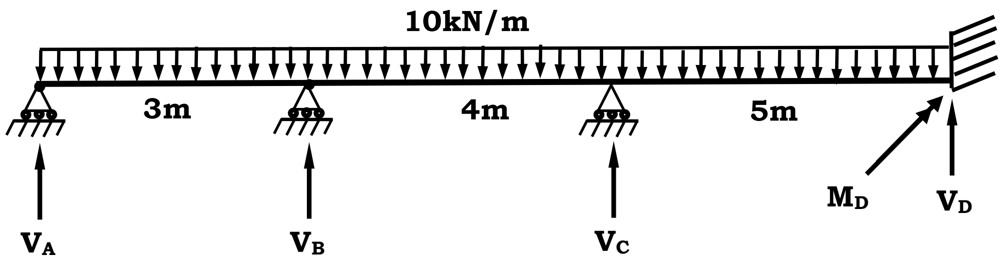


Fig.7.1Three Span Continuous Beam Example

Step1: Considering MDI Parent1 as main menu in that structure category is selected as Beam as shown in **Fig. 7.2**

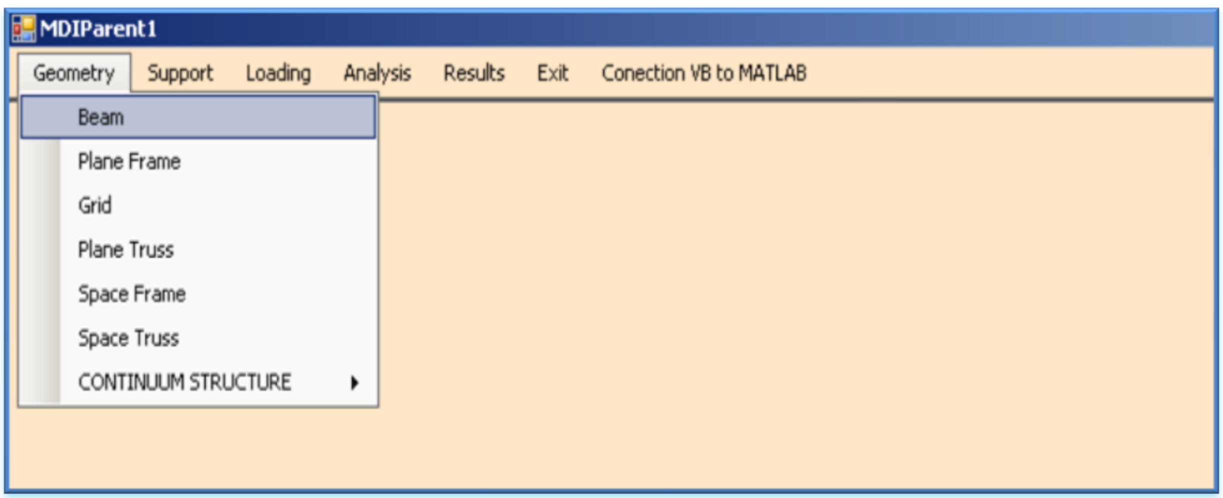


Fig. 7.2 Main Menu for Selecting Type of Structure

Step 2: In geometry menu as shown in **Fig. 7.3** number of spans is entered as 3 and in screen shot depicted in **Fig. 7.4** length of the members is entered as 3m, 4m and 5m.

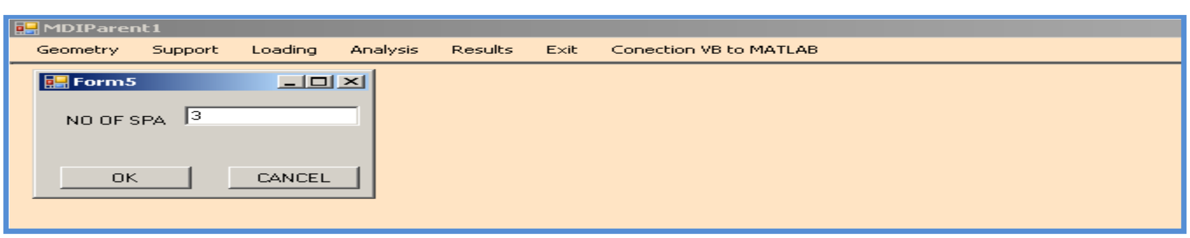


Fig.7.3 Numbers of Span

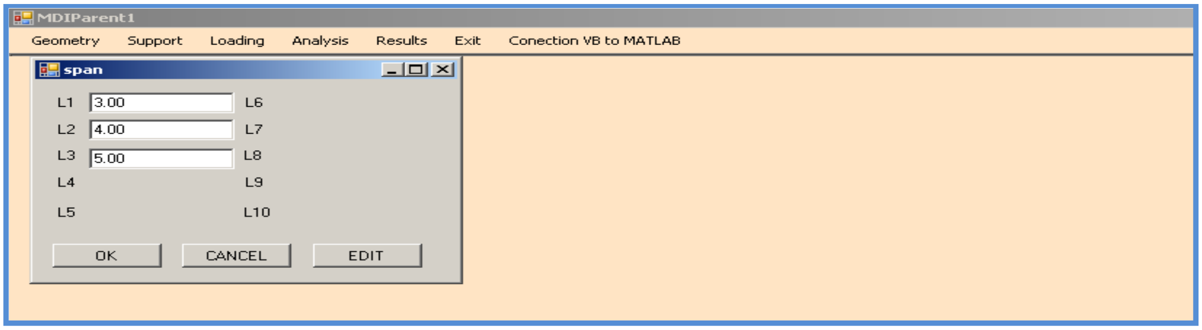


Fig.7.4 Length of Individual Span in m

Step 3: By clicking Ok button geometry of beam is drawn with four nodes and individual members as 1-2, 2-3 and 3-4 as shown in **Fig. 7.5.**

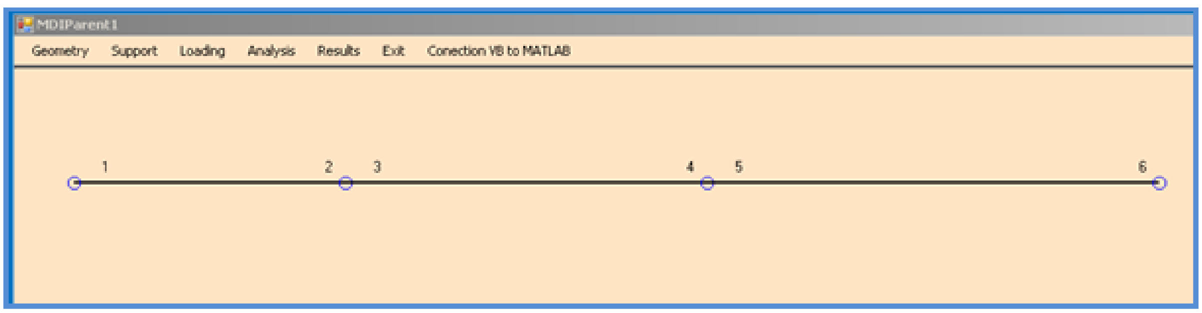


Fig.7.5 Geometry of Beam

Step 4: By clicking support condition in second sub menu a number of options for selecting the support condition for first three supports from left to right three hinged supports are selected and for right most end fixed support is selected **(Fig.7.6).**

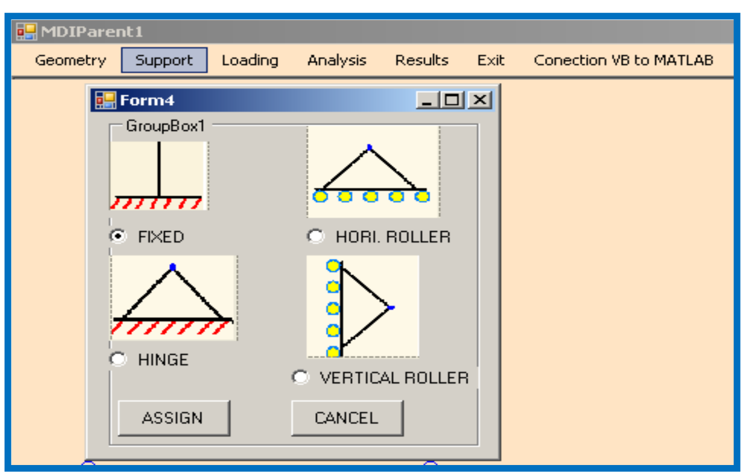


Fig. 7.6 Support Conditions

Step 5: Selecting individual support conditions by clicking button the complete beam geometry can be drawn as shown in **Fig. 7.7.**

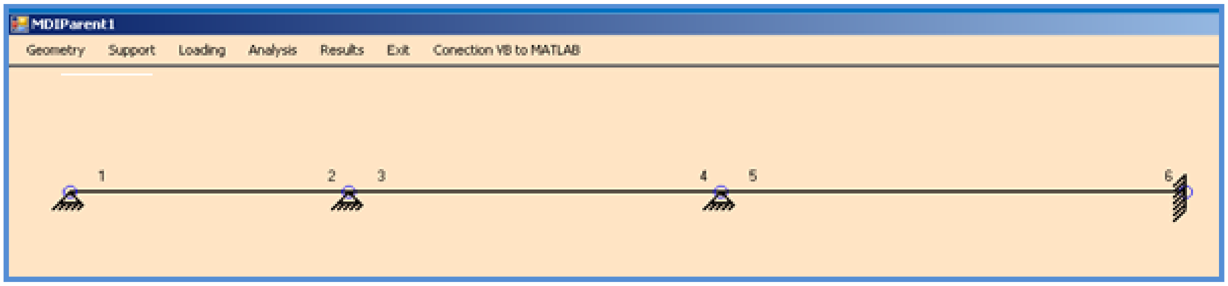


Fig 7.7 Support Conditions of Beam

Step 6: By clicking loading option in main menu different loading type is selected with its magnitude and other necessary details as depicted here in **Figs. 7.8** and **7.9**.

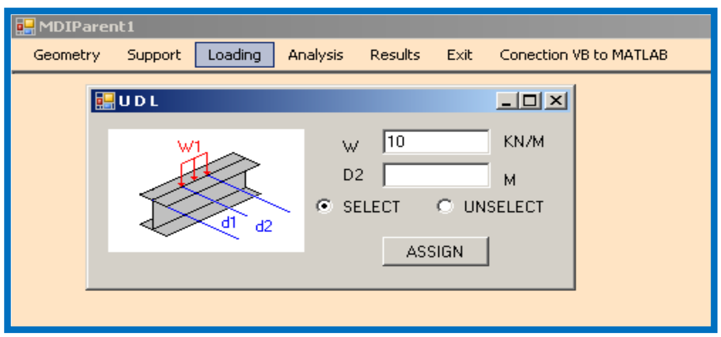


Fig.7.8 Load Selection

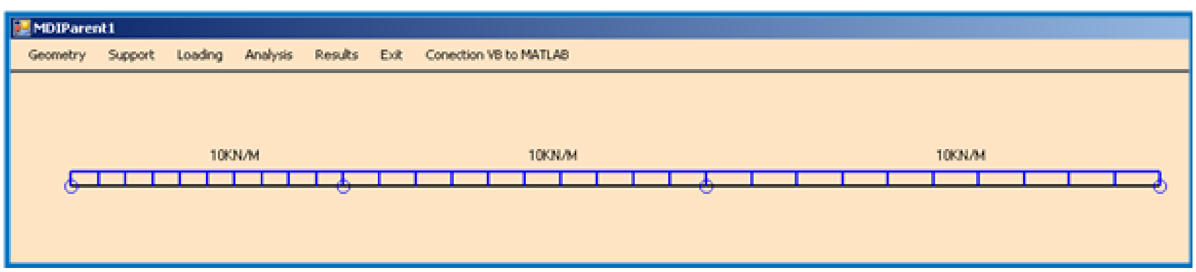


Fig. 7.9 Continuous Beam with UDL

Step 7: After supplying the necessary input data connection of VB.NET with Matlab is established by enabling externally (Chapter 6) and activating COM server library in which at front end Matlab is considered as object, while the solution obtained using object browser is considered as string. The coding on connection to VB.Net of Matlab is depicted here in **Fig. 7.10.**

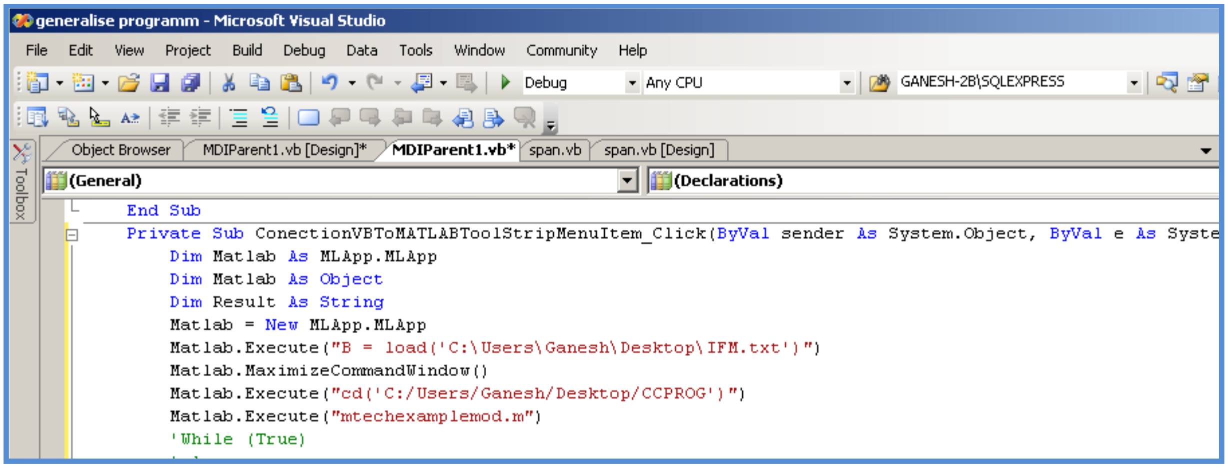


Fig. 7.10 Matlab Code in VB.Net in Main Menu

By clicking the Main menu of VB.NET, one can click object browser to select MLApp which shows various operations as depicted in **Fig. 7.11.**

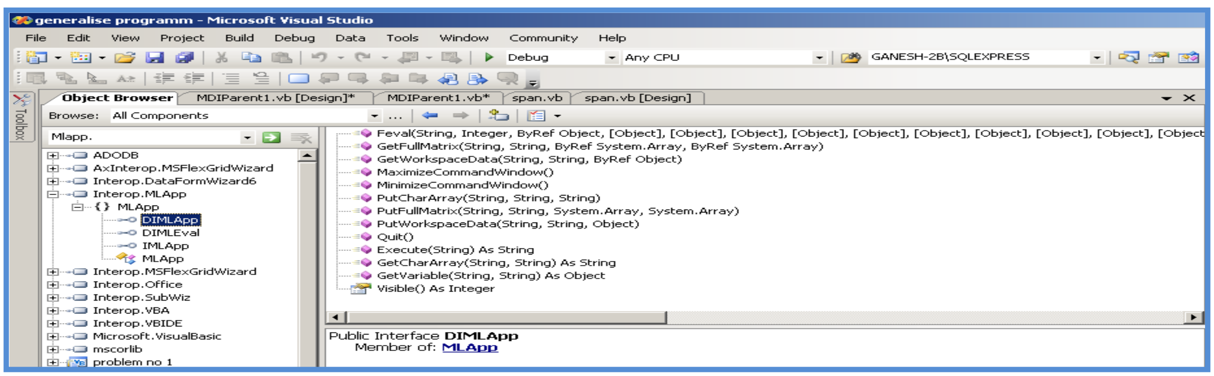


Fig. 7.11 Various Operations under MLApp Object.

Step 8: By following the operation given in Step 7, Matlab command window can be obtained using VB.Net form which a number of Matlab mathematical operations can be executed easily. It is depicted in **Fig.7.12.**

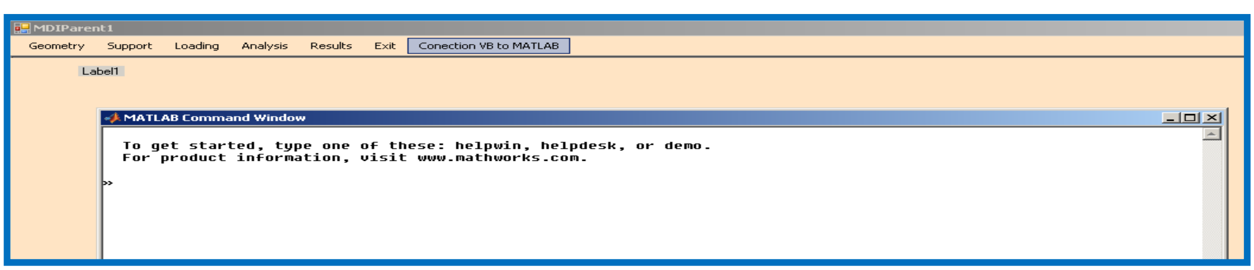


Fig. 7.12 Matlab Command Window

Step 9: The equilibrium matrix based on joint displacements can be seen with two rows corresponding to nodal displacements and columns corresponding to internal moments as depicted in **Fig.7.13.**

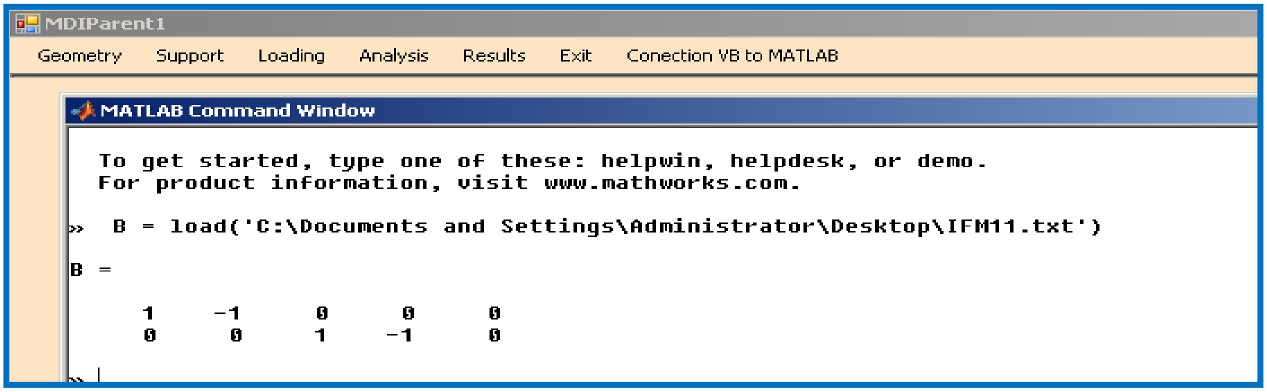


Fig. 7.13 Equilibrium Matrix in Matlab Command Window

Step10: By changing path to the folder named as "CCPROG", mtechexamplemod.m file is connected to command window of Matlab, which gives the necessary compatibility conditions required for making the global equilibrium matrix [S] from rectangular to square. The compatibility matrix consists of random distribution of dependent unknowns and independent unknowns which are known as 'codedepB' and 'codeindB' respectively. Thus out of total five moment unknowns the coefficients of M_1 and M_3 are categorized as dependent while the rest as independent as depicted in **Fig 7.14.**

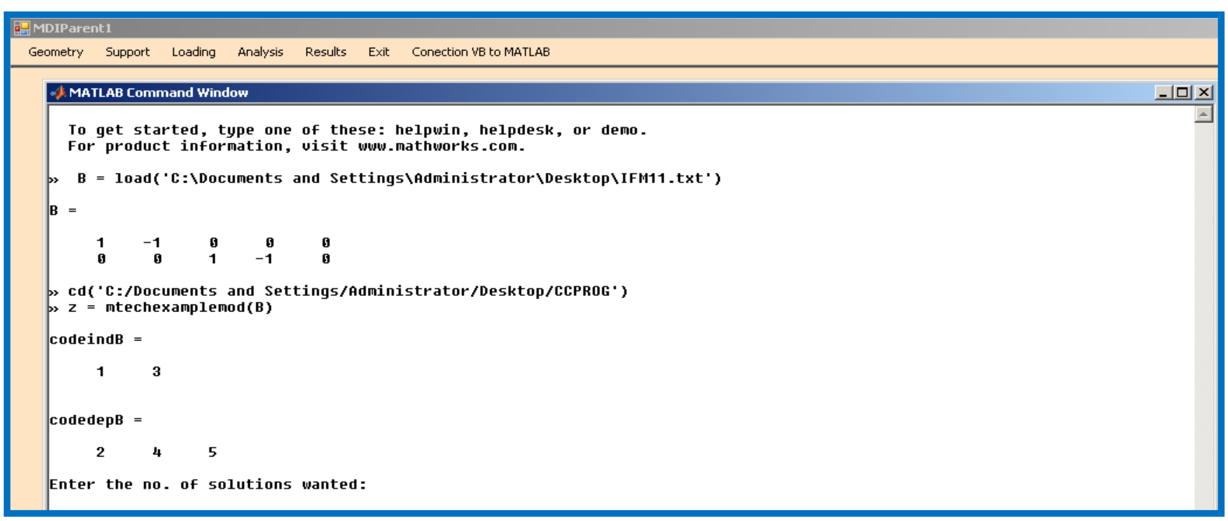


Fig.7.14 CodedepB and CodeindepB.

Step11: By selecting 3 conditions in **Fig. 7.14**, one can get complete equation as solution1, solution2 and solution3 with individual distribution of codedepB and codeindepB as depicted in **Fig. 7.15,** while the matrix named as z.cMatrix can be seen and product of [B]matrix and Transpose (z.cMatrix) can be checked for null value to verify the IFM based formulation as depicted in **Fig.7.16.**

```
MATLAB Command Window
Solution1
solution =
8*B2 +9*B4 +2*B5 +8*B1 +9*B3

alphadep =

     9
     6
     1

alphaind =

     9
     6
Solution2
solution =
9*B2 +6*B4 +1*B5 +9*B1 +6*B3

alphadep =

     3
     5
     9

alphaind =

     3
     5
Solution3
solution =
3*B2 +5*B4 +9*B5 +3*B1 +5*B3
```

Fig. 7.15 Solution1, Solution2 and Solution3

```
      cTransposeB: [3x2 double]

>> z.cMatrix

ans =

     8     8     9     9     2
     9     9     6     6     1
     3     3     5     5     9

>> B

B =

     1    -1     0     0     0
     0     0     1    -1     0

>> Nullmatrix = B*transpose(z.cMatrix)

Nullmatrix =

     0     0     0
     0     0     0
```

Fig.7.16 Z.cMatrix and Null Matrix

Step12: Normalized global compatibility conditions (GlobalCC) can be worked out by finding product of z.cMatrix and global flexibility matrix Gmatrix i.e. β = [G]{M}. The global equilibrium matrix (Smatrix) is obtained by just concatenating three rows of GlobalCC into two rows of basic equilibrium matrix [B], thus it becomes a square matrix as can be seen in **Fig. 7.17**

```
>> GlobalCC = z.cMatrix*Gmatrix*10

GlobalCC =

    0.0480    0.1000    0.1040    0.1000    0.0650
    0.0540    0.0960    0.0840    0.0650    0.0400
    0.0180    0.0440    0.0520    0.0950    0.1150

>> Smatrix = [B;GlobalCC]

Smatrix =

    1.0000   -1.0000         0         0         0
         0         0    1.0000   -1.0000         0
    0.0480    0.1000    0.1040    0.1000    0.0650
    0.0540    0.0960    0.0840    0.0650    0.0400
    0.0180    0.0440    0.0520    0.0950    0.1150
```

Fig.7.17 GlobalCC and Smatrix

Step13: The inverse of Smatrix is calculated directly by typing inv(Smatrix) at the command prompt. The respective numerical values and any variables like a, b c, L x … must be predefined as procedure i.e. syms x y a b…. .The loading vector {P} corresponding to nodal rotational displacement can be written of size 5 x 1 as depicted in **Fig. 7.18**. After multiplying Sinv with loading matrix {P}, the initial moments are calculated and by adding equivalent joint loads final moments are calculated as shown in **Fig. 7.19.** Result for nodal displacements is depicted in **Fig. 7.20.**

```
>> Sinv = inv(Smatrix)

Sinv =

    0.5373   -0.1493  -21.8725   27.0225    2.9636
   -0.4627   -0.1493  -21.8725   27.0225    2.9636
    0.1194    0.5224   28.7030  -25.2440   -7.4429
    0.1194   -0.4776   28.7030  -25.2440   -7.4429
   -0.0597    0.2388  -24.8978   17.6997   16.6117

>> Pmatrix = [-5.83
-7.5
0
0
0
]

Pmatrix =

   -5.8300
   -7.5000
         0
         0
         0
```

Fig.7.18 Sinv and {P} Vector

```
>> Mmatrix = Sinv*Pmatrix

Mmatrix =

   -2.0132
    3.8168
   -4.6140
    2.8860
   -1.4430

>> Mcorrection = [-7.5
-13.33333333
-13.33333333
-20.83333333
-20.83333333
]

Mcorrection =

   -7.5000
  -13.3333
  -13.3333
  -20.8333
  -20.8333

>> Mfinal = Mmatrix + Mcorrection

Mfinal =

   -9.5132
   -9.5165
  -17.9473
  -17.9473
  -22.2763
```

Fig.7.19 Final Moments in kN-m

```
>> Disp = transpose(Sinv)*Gmatrix*Mmatrix

Disp =

   -0.0012
   -0.0022
   -0.0000
    0.0000
    0.0000
```

Fig.7.20 Nodal Displacements

7.2 COMPUTER IMPLEMENTATION OF DIFM

As DIFM is mathematically modified form of IFM it gives identical solution. The programming of the total solution procedure is done using VB 6, in which Matlab is not required for the development of compatibility conditions. Different elemental level matrices such as Equilibrium matrix [Be], Flexibility matrix [Ge] and Dual matrix [De] are directly developed using basic input as geometrical parameters of framed structures. Depending upon possible free joint displacements, different global matrices are generated using programming procedure based on standard stiffness assembly procedure. By calling inverting routine, nodal displacements {δ} are calculated and finally the moments are calculated.

A two span prismatic continuous beam is analysed here as an illustrative example of DIFM to find internal moments and nodal displacements by assuming value of EI as 1666.67 kN-m^2 (**Fig. 7.21).**

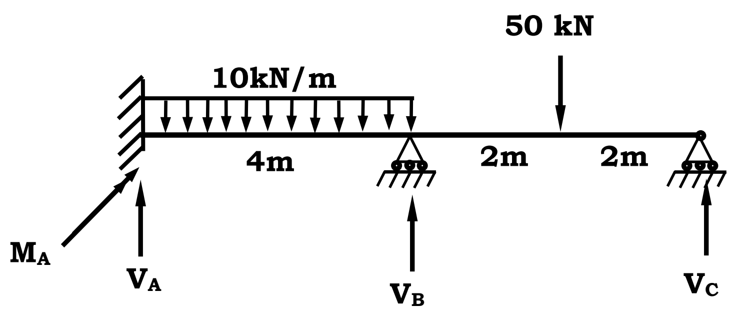


Fig. 7.21 Key Diagram for Continuous Beam Example

Step1: Through MDI Parent1 initial form structure category is selected as beam as shown in **Fig. 7.22.**

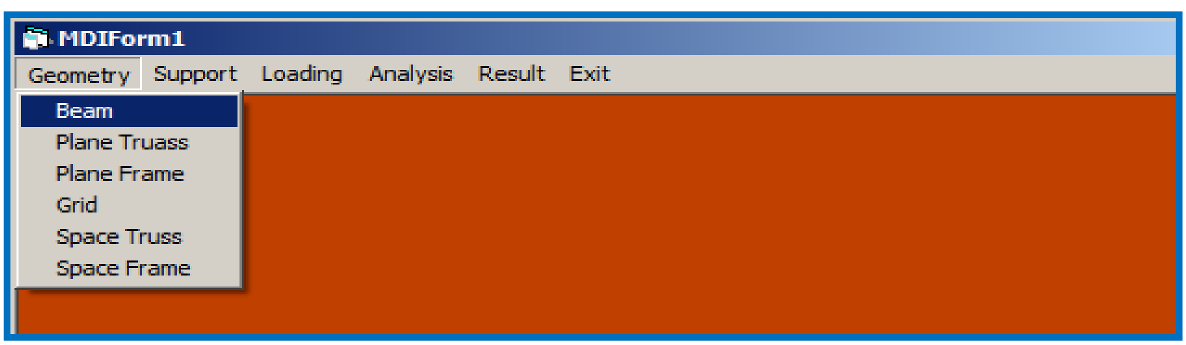


Fig. 7.22 Selection of Type of Structure

Step 2: The number of span with their individual length and flexural rigidities are entered through form depicted in **Figs. 7.23** and **7.24.**

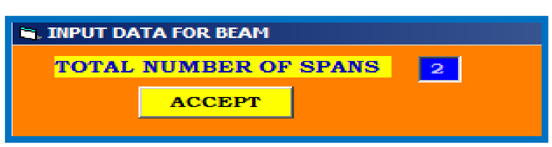


Fig. 7.23 Number of Spans

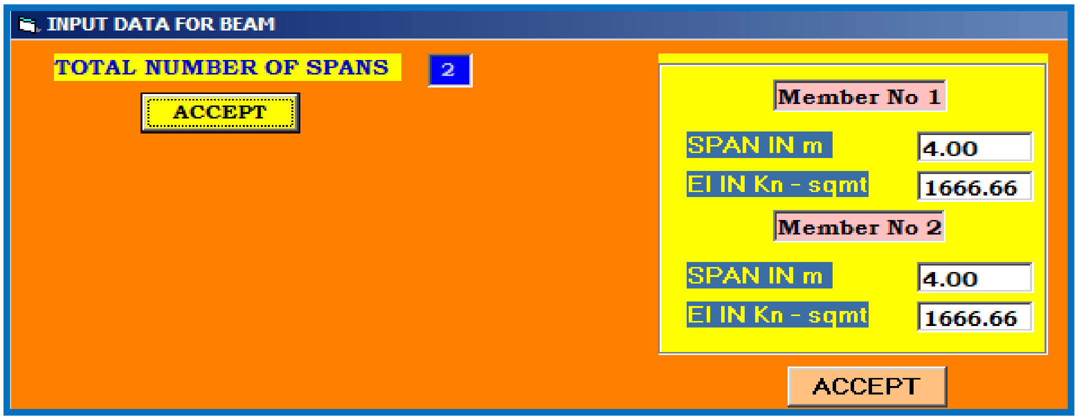


Fig.7.24 Member Properties

Step 3: Once details of **Fig.7.24** are accepted, form for selecting the type of support can be opened as shown in **Fig. 7.25.**

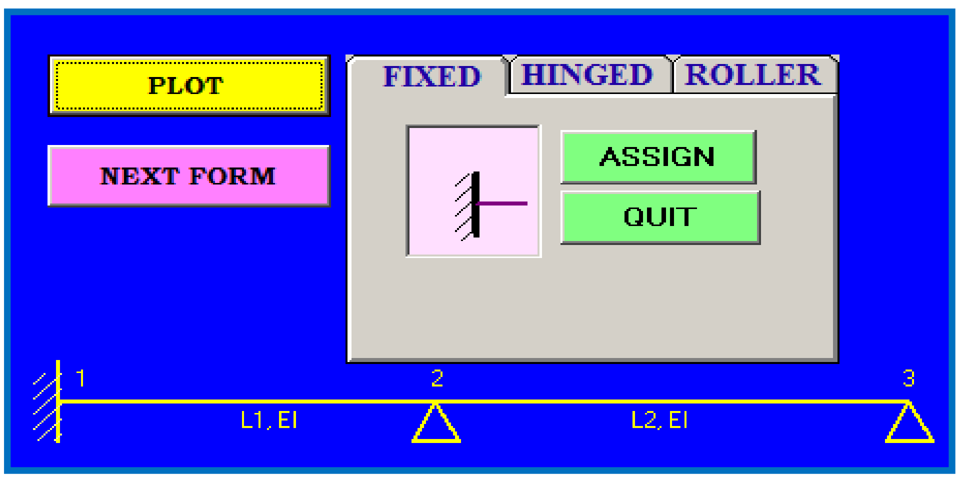


Fig. 7.25 Option for Support Conditions

Step 4: Once next form of **Fig. 7.26** is clicked the type of loading can be selected. Once the selection is made it shows the beam.

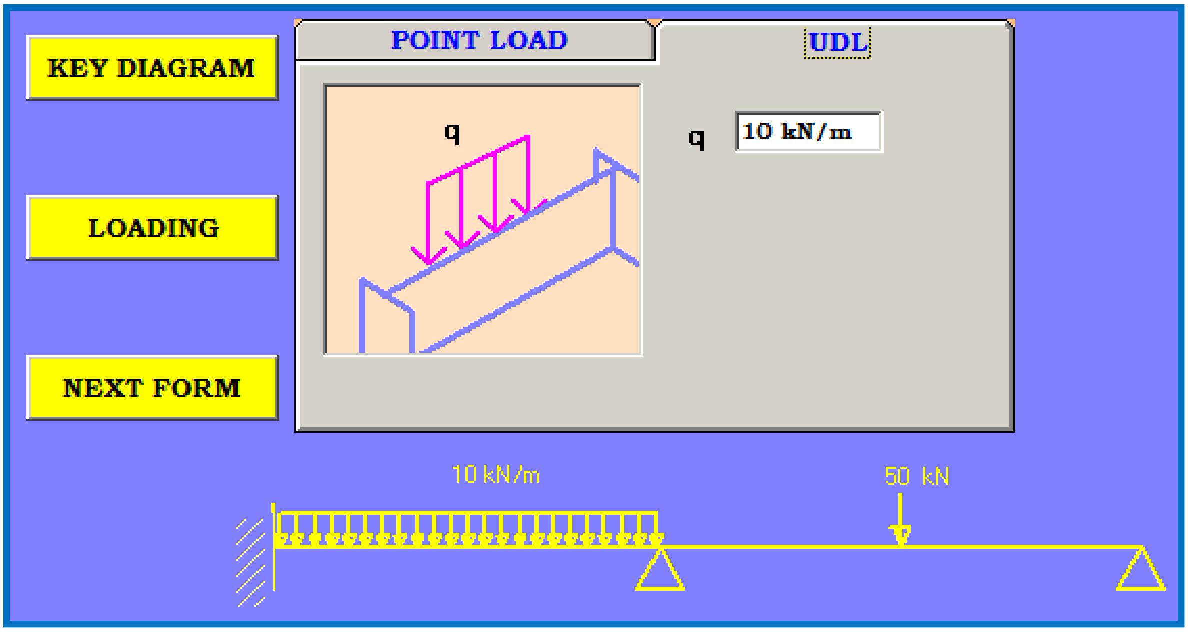


Fig. 7.26 Key Diagram with Option for Support Conditions

Step 5: Various elemental matrices are calculated with coding written on double click of the command buttons, which represents all the necessary matrices. All the matrices are depicted in MSFlexgrid as shown in **Fig. 7.27**.

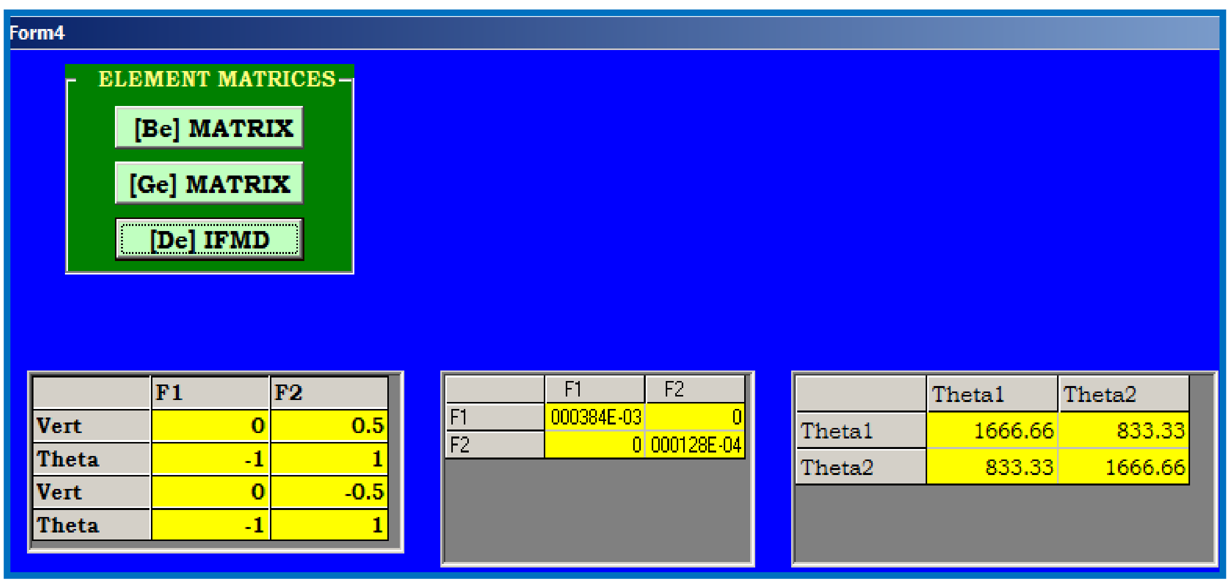


Fig.7.27 Different Elemental Matrices

Step 6: The next form as shown in **Fig. 7.28** can be seen by double click on form4 (**Fig. 7.27**). The Visual basic code is scripted for the event at double click of the same form.

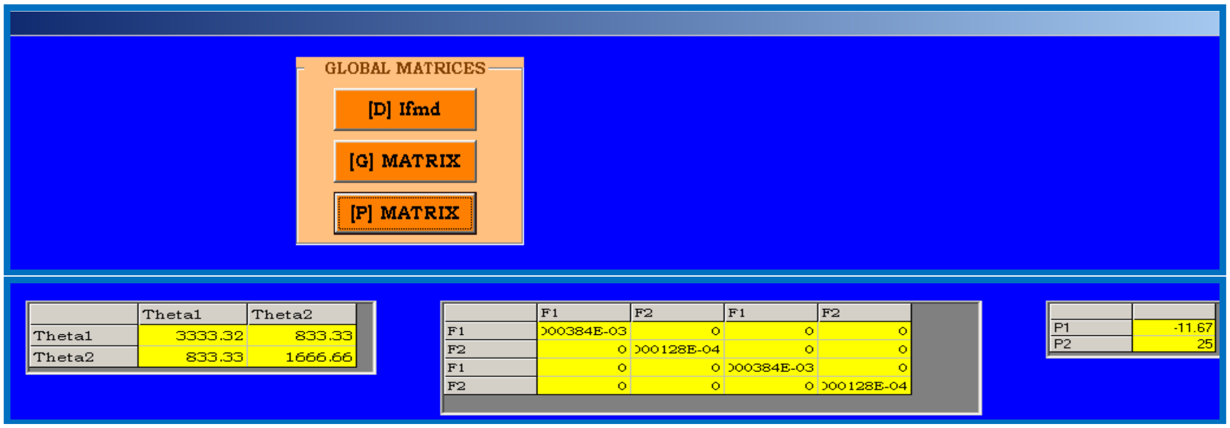


Fig. 7.28 Various Global Matrices

Step 7: The primary nodal rotational displacements can be calculated by clicking command button Displacement whereas internal moments can be calculated by clicking command of Moments as **Fig. 7.29.**

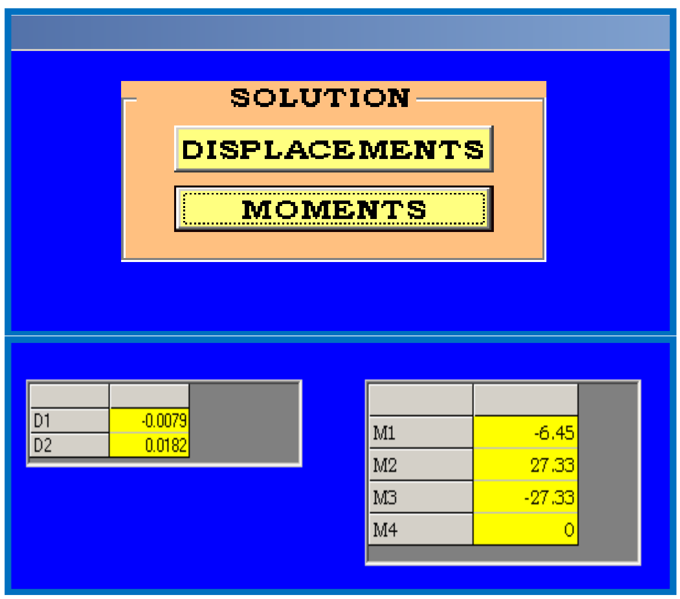


Fig. 7.29 Primary and Secondary Unknowns

7.3 EXAMPLE OF STATIC ANALYSIS OF BEAM

A three span continuous beam as shown in **Fig. 7.30** is analysed here by assuming equals to 1666.67 kN-m^2 for all the spans.

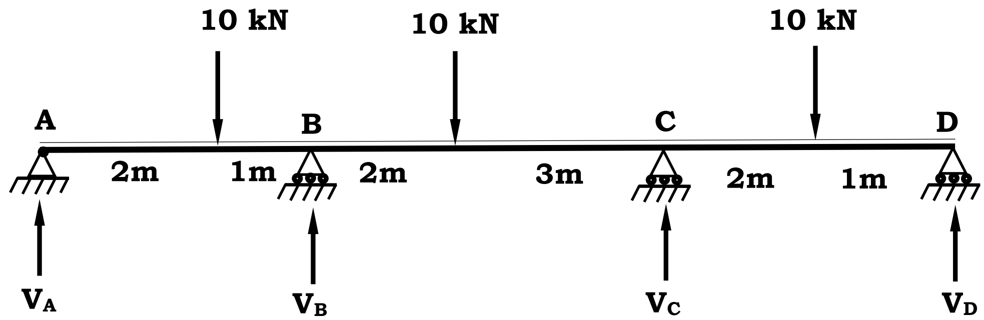


Fig.7.30 A Continuous Beam Example

Step 0 – Solution strategy: The beam has two degrees of statically indeterminate. It is discretized into three elements. Each beam has two moment unknowns and hence whole beam has 4 moment unknowns (n) as shown in **Fig. 7.31** and 2 intermediate rotational displacements as unknowns (m). The extreme ends are simply supported which reduces directly number of unknowns and size of matrices by omitting the end moments for being zero.

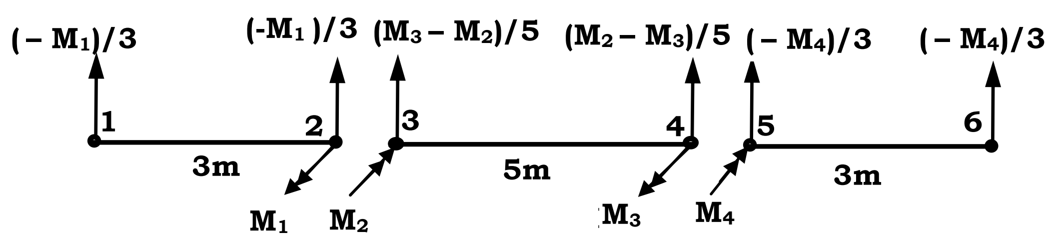


Fig. 7.31 Three Element Discretization

Step 1 – Formulate the equilibrium equations: The EE can be written along two rotational displacement directions as follows:

Along θ_B, $M_1 - M_2 = -2.76$ ... (7.1)

Along θ_C, $M_3 - M_4 = 2.58$ ... (7.2)

In matrix notation ([B] {F} = {P}), Thus the Equilibrium Equations (EEs) can be written as

$$\begin{bmatrix} 1 & -1 & 0 & 0 \\ 0 & 0 & 1 & -1 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -2.76 \\ +2.58 \\ 0.00 \\ 0.00 \end{Bmatrix} \qquad \ldots (7.3)$$

Steps 2 – Derive the deformation displacement relations: The Displacement Deformation Relations are obtained as ($\{\beta\} = [B]^T \{\delta\}$).

$$\beta_1 = \theta_C,\ \beta_2 = -\theta_C,\ \beta_3 = \theta_B \text{ and } \beta_4 = -\theta_B \qquad \ldots (7.4)$$

Step 3 – Generate the compatibility conditions: The CC for the problem can be obtained using 'mtechexamplemod', which is .m file for calculating compatibility conditions based on [B] matrix.

$$\begin{bmatrix} 8 & 8 & 9 & 9 \\ 2 & 2 & 9 & 9 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = [C]\{\beta\} \qquad \ldots (7.5)$$

Correctness of CC can be verified from its null property ($[B]\ [C]^T = [0]$).

Step 4 – Formulate force deformation relations: Substituting each β in terms of end moments for beam member and writing in terms of internal moments,

$$\beta_i = \frac{L}{6EI}(2M_i + M_j),\ \beta_j = \frac{L}{6EI}(2M_j + M_i) \qquad \ldots (7.6)$$

Formulating above equations for all the three members, one can write

$$\beta_1 = \frac{3}{6EI}(2M_1) \qquad \beta_2 = \frac{5}{6EI}(2M_2 + M_2)$$

$$\beta_3 = \frac{5}{6EI}(2M_3 + M_4) \quad \text{and } \beta_4 = \frac{3}{6EI}(2M_4) \qquad \ldots (7.7)$$

Arranging above Eq. (7.7) in matrix form, one can write

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = \frac{1}{EI}\begin{bmatrix} \frac{6}{6} & 0 & 0 & 0 \\ 0 & \frac{10}{6} & \frac{5}{6} & 0 \\ 0 & \frac{5}{6} & \frac{10}{6} & 0 \\ 0 & 0 & 0 & \frac{6}{6} \end{bmatrix}\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = [G]\{M\} \qquad \ldots (7.8)$$

Substituting {β} from Eq. (7.8) into Eq. (7.5) and concatenating the same matrix from bottom side of [B], one can write

$$\begin{bmatrix} 1 & -1 & 0 & 0 \\ 0 & 0 & 1 & -1 \\ 0.048 & 0.125 & 0.13 & 0.054 \\ 0012 & 0.065 & 0.10 & 0.054 \end{bmatrix}\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -2.76 \\ +2.58 \\ 0.00 \\ 0.00 \end{Bmatrix} \qquad \ldots (7.9)$$

or [S]{M} = {P}

Solving above equations, one can get values of internal moments as

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -1.9481 \\ 0.8119 \\ 0.7138 \\ -1.8662 \end{Bmatrix} \qquad \ldots (7.10)$$

Corresponding correcting vector of moments due to loading is obtained as

$$\begin{Bmatrix} M_{11} \\ M_{21} \\ M_{31} \\ M_{41} \end{Bmatrix} = \begin{Bmatrix} -4.44 \\ -7.2 \\ -4.8 \\ -2.22 \end{Bmatrix} \qquad \ldots (7.11)$$

Thus adding above moment vectors, the final moments are

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -6.381 \\ -6.381 \\ -4.084 \\ -4.086 \end{Bmatrix} \text{kN-m} \qquad \ldots (7.12)$$

The nodal displacements {δ} can be calculated form [J][G]{M}, where [J] = $[S^{-1}]^T$, [G] = Global flexibility matrix and {M} represents values calculated from Eq. (7.10). Thus substituting all related matrix into the above relation, one can have rotation at B and C ends as -0.0012 and 0.0011 radians respectively.

7.4 EXAMPLE OF STATIC ANALYSIS OF PLANE TRUSS

A plane truss with total six members is depicted in **Fig. 7.32.** The length of vertical and horizontal members is considered as 2m with cross sectional dimensions of each member as 0.01m x 0.01m. The truss is made of mild steel having Young's modulus of elasticity E as 2.01×10^8 kN/m^2. Calculate the internal forces and nodal displacements.

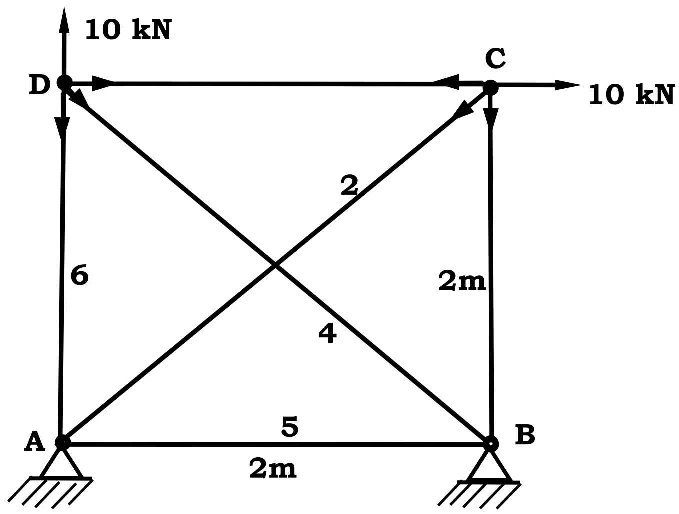


Fig. 7.32 Plane Truss Example

Step 0 – Solution strategy: The truss is statically indeterminate to two degree. Each member has one internal axial force (F_1, F_2....F_6) as unknown (n) and total four free nodal displacements (m).

Step 1 – Formulate the equilibrium equations: The EEs can be written in terms of internal forces at joint 3 and 4 along horizontal and vertical displacement direction as follows:

At joint '3' in [δch] direction,

$$\overleftarrow{-F_1} - \overleftarrow{F_2}\cos45^0 + \overrightarrow{10} = 0$$

At joint '3' in [δcv] direction,

$$\overset{\downarrow}{-F_3} - \overset{\downarrow}{F_2}\cos45^0 = 0$$

Similarly, one can write the equations at joint '4'. The equilibrium equations (EEs) can be written as

$$\begin{bmatrix} 1 & 0.707 & 0 & 0 & 0 & 0 \\ 0 & 0.707 & 1 & 0 & 0 & 0 \\ -1 & 0 & 0 & -0.707 & 0 & 0 \\ 0 & 0 & 0 & 0.707 & 0 & 1 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} = \begin{Bmatrix} 10.00 \\ 0.00 \\ 0.00 \\ 10.00 \end{Bmatrix}$$

Step 2 – Derive the deformation displacement relations: The DDRs are obtained from ($\{\beta\} = [B]^T\{\delta\}$) as

$$\beta_1 = \delta_{CH} - \delta_{DH}$$
$$\beta_2 = 0.707\delta_{CH} + 0.707\delta_{CV}$$
$$\beta_3 = \delta_{CV}$$
$$\beta_4 = 0.707\delta_{DV} + 0.707\delta_{DH}$$
$$\beta_5 = 0$$
$$\beta_6 = \delta_{DV}$$

Step 3 – Generate the compatibility conditions: The CC for the problem can be obtained using 'mtechexamplemod', which is .m file for calculating compatibility conditions based on [B] matrix.

$$\begin{bmatrix} 9.00 & -12.7298 & 9.0 & -12.7298 & 8.00 & 9.00 \\ 9.00 & -12.7298 & 9.00 & -12.7298 & 2.00 & 9.00 \end{bmatrix} \{\beta\} = [C]\{\beta\}$$

Correctness of CCs can be verified from its null property ($[B][C]^T$) which is worked out to be

$$1.00 \times 10^{-14} \begin{bmatrix} -0.1776 & -0.1776 & -0.3513 & -0.1776 \\ -0.1776 & -0.1776 & -0.3513 & -0.1776 \end{bmatrix}$$

Step 4 – Formulate Force Deformation Relations: The force deformation relation for individual member can be written in terms of axial rigidities, which is as follows:

$$\beta_1 = F_1L_1/A_1E_1,\ \beta_2 = F_2L_2/A_2E_2 \ldots\ldots\ \beta_n = F_nL_n/A_nE_n \qquad \ldots(7.13)$$

Substituting the lengths and axial rigidities in Eq. (7.13) and writing in a matrix form one gets

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \end{Bmatrix} = \frac{1}{2.01 \text{ x } 10^{6}} \begin{bmatrix} 2.00 & 0.00 & 0.00 & 0.00 & 0.00 & 0.00 \\ & 2.828 & 0.00 & 0.00 & 0.00 & 0.00 \\ & & 2.00 & 0.00 & 0.00 & 0.00 \\ & \text{Symm} & & 2.828 & 0.00 & 0.00 \\ & & & & 2.00 & 0.00 \\ & & & & & 2.00 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} = [G]\{F\} \quad \ldots (7.14)$$

Substituting {β} from Eq.(7.14) into Step 3, after normalization, the complete IFM matrix is obtained as

$$\begin{bmatrix} 1 & 0.707 & 0 & 0 & 0 & 0 \\ 0 & 0.707 & 1 & 0 & 0 & 0 \\ -1 & 0 & 0 & -0.707 & 0 & 0 \\ 0 & 0 & 0 & 0.707 & 0 & 1 \\ 0.0896 & -0.1791 & 0.0896 & -0.1791 & 0.0796 & 0.0896 \\ 0.0896 & -0.1791 & 0.0896 & -0.1791 & 0.0199 & 0.0896 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} = \begin{Bmatrix} 10.00 \\ 0.00 \\ 0.00 \\ 10.00 \\ 0.00 \\ 0.00 \end{Bmatrix}$$

Solving above equations, one can get the values of internal forces as

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} = \begin{Bmatrix} +3.2674 \\ +9.5227 \\ -6.7326 \\ -4.6216 \\ 0.00 \\ +13.2674 \end{Bmatrix} \text{kN} \quad \ldots (7.15)$$

The nodal displacements {δ} are obtained as follows:

$$\{\delta\} = \begin{Bmatrix} \delta_{CH} \\ \delta_{CV} \\ \delta_{DH} \\ \delta_{DV} \end{Bmatrix} = \begin{Bmatrix} +0.2565 \\ -0.067 \\ +0.2240 \\ +0.1321 \end{Bmatrix} \text{ x } 10^{-6} \text{ m}$$

7.5 EXAMPLES OF STATIC ANALYSIS OF PLANE FRAMES

7.5.1 Hinged Footed Plane Frame Example

A one bay two storey plane frame having total six members is shown in **Fig. 7.33.** The length of vertical members is considered as 4m whereas length of horizontal members is considered as 6m with area of cross section of each members as 0.01m x 0.01m. Considering E as 2.01×10^8 kN/m^2 calculate the internal forces and nodal displacements.

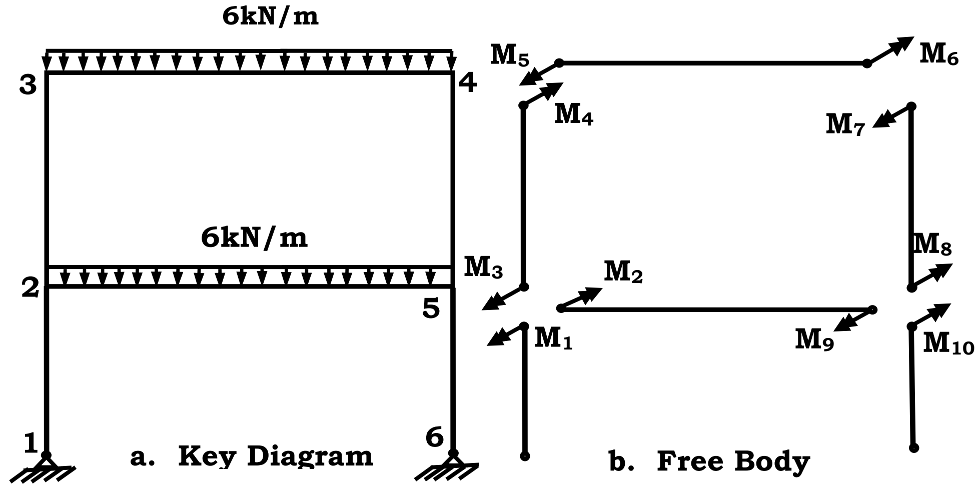


Fig.7.33 Hinged Footed Plane Frame Example

Step 0 – Solution strategy: The plane frame is having total ten internal moments (n = 10). Further, the portal frame being symmetrical it will have only rotational degrees of freedom (m = 4).

Step 1 – Formulate the equilibrium equations: The EEs can be written in terms of internal forces at joint 2, 3, 4 and 5 along moment direction only which are as follows:

At joint '2' in [Θ_2] direction

$$M_1 - M_2 + M_3 = -18 \qquad \ldots (7.16)$$

writing similar condition at each joint in rotational direction in matrix notation, the equilibrium equations (EEs) can be written as

$$\begin{bmatrix} 1 & -1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 1 & -1 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_7 \\ M_8 \\ M_9 \\ M_{10} \end{Bmatrix} = \begin{Bmatrix} -18.00 \\ -18.00 \\ 18.00 \\ 18.00 \end{Bmatrix} \qquad \ldots (7.17)$$

Step 2 – Derive the deformation displacement relations: The DDRs are expressed as ({β} = [B] ^{T} {δ})

$\beta_1 = \theta_1$, $\beta_2 = -\theta_1$, $\beta_3 = \theta_1$, $\beta_4 = -\theta_2$, $\beta_5 = \theta_2$, $\beta_6 = \theta_3$,

$\beta_7 = \theta_3$, $\beta_8 = -\theta_4$, $\beta_9 = \theta_4$ and $\beta_{10} = -\theta_4$ ... (7.18)

Step 3 – Generate the compatibility conditions: The CCs for the plane problem are obtained by using 'mtechexamplemod', which is .m file in which a complete [B] matrix is supplied through input file for calculating the compatibility conditions based on LIUT concept.

$$\begin{bmatrix} -1 & 8 & 8 & 2 & 2 & 9 & 9 & 6 & 9 & 1 \\ -2 & 3 & 5 & 9 & 9 & 9 & 9 & 2 & 11 & 9 \\ 4 & 9 & 5 & 8 & 8 & 2 & 2 & 4 & 13 & 9 \\ -1 & 8 & 9 & 6 & 6 & 1 & 1 & 8 & 17 & 9 \\ 0 & 7 & 7 & 7 & 7 & 4 & 4 & 6 & 8 & 2 \\ 6 & 7 & 1 & 3 & 3 & 1 & 1 & 1 & 9 & 8 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \\ \beta_7 \\ \beta_8 \\ \beta_9 \\ \beta_{10} \end{Bmatrix} = [C]\{\beta\} \quad \text{... (7.19)}$$

Correctness of CC can be verified from its null property ([B] [C]^T).

Step 4 – Formulate force deformation relations: The force deformation relation for individual member can be written in terms of bending rigidity EI. In the matrix form it can be written as

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \\ \beta_7 \\ \beta_8 \\ \beta_9 \\ \beta_{10} \end{Bmatrix} = \frac{1}{EI} \begin{bmatrix} 1.3 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 2.0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & 0 & 1.3 & 0.6 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & 0.6 & 1.3 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & 0 & 2 & 1 & 0 & 0 & 0 & 0 \\ & & & & 1 & 2 & 0 & 0 & 0 & 0 \\ & & & & & 0 & 1.3 & 0.6 & 0 & 0 \\ & & & & & & 0.6 & 1.3 & 0 & 0 \\ & & & & & & & 0 & 2.0 & 0 \\ & & & & & & & & 0 & 1.3 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_7 \\ M_8 \\ M_9 \\ M_{10} \end{Bmatrix} = [G]\{F\} \quad \text{...(7.20)}$$

Substituting {β} and after concatenating the matrix into [B] matrix from bottom side, the basic IFM relation is obtained as

$$\begin{bmatrix} 1 & -1 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 1 & -1 \\ 0.0192 & 0.079 & 0.036 & 0.0504 & 0.0684 & 0.0396 & 0.0144 & 0.0216 & 0.0828 & 0.0192 \\ -0.0048 & 0.090 & 0.0432 & 0.0288 & 0.0324 & 0.0432 & 0.0366 & 0.036 & 0.1044 & 0.0288 \\ 0 & 0.0792 & 0.0408 & 0.0312 & 0.0468 & 0.0612 & 0.048 & 0.0456 & 0.0828 & 0.0096 \\ -0.0144 & 0.0468 & 0.0456 & 0.0552 & 0.0792 & 0.0612 & 0.0336 & 0.0384 & 0.072 & 0.0144 \\ 0.0192 & 0.1152 & 0.0264 & 0.0312 & 0.0612 & 0.0684 & 0.0552 & 0.060 & 0.1548 & 0.0432 \\ 0.0144 & 0.0756 & 0.0144 & 0.0144 & 0.0252 & 0.0288 & 0.0336 & 0.0456 & 0.0972 & 0.0144 \end{bmatrix}$$

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_7 \\ M_8 \\ M_9 \\ M_{10} \end{Bmatrix} = \begin{Bmatrix} -18 \\ -18 \\ 18 \\ 18 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \qquad \ldots (7.21)$$

or [S]{M} = {P}

Solving above set of equations one can get values of internal nodal moments. After adding corresponding correction moments in M_2, M_5, M_6 and M_9, the final moments are found as,

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_6 \\ M_7 \\ M_8 \\ M_9 \\ M_{10} \end{Bmatrix} = \begin{Bmatrix} -4.4505 \\ 16.022 \\ -11.5714 \\ -14.2418 \\ -14.2418 \\ -14.1428 \\ -14.2418 \\ -11.5714 \\ 16.022 \\ -4.4505 \end{Bmatrix} \text{kN} - \text{m} \qquad \ldots (7.22)$$

The solution obtained in terms of all the joint moments using STAAD.pro *v8i* are found in good agreement with maximum difference of 1.75%.

The nodal displacements {δ} can be worked out by using [J][G]{M} and are found as

$$\{\delta\} = \begin{Bmatrix} \theta_2 \\ \theta_3 \\ \theta_4 \\ \theta_5 \end{Bmatrix} = \begin{Bmatrix} -0.0036 \\ -0.0068 \\ +0.0068 \\ +0.0036 \end{Bmatrix} \text{ radians} \qquad \dots (7.23)$$

7.5.2 Two Bay Two Storey Plane Frame Example

A plane Frame having two bay and two storey having total ten members is shown in **Fig. 7.34.** Considering cross sectional dimension of each member as 0.01m x 0.01m, modulus of elasticity E as 2.01 x 10^8 kN/m^2. Calculate the internal forces and nodal displacement.

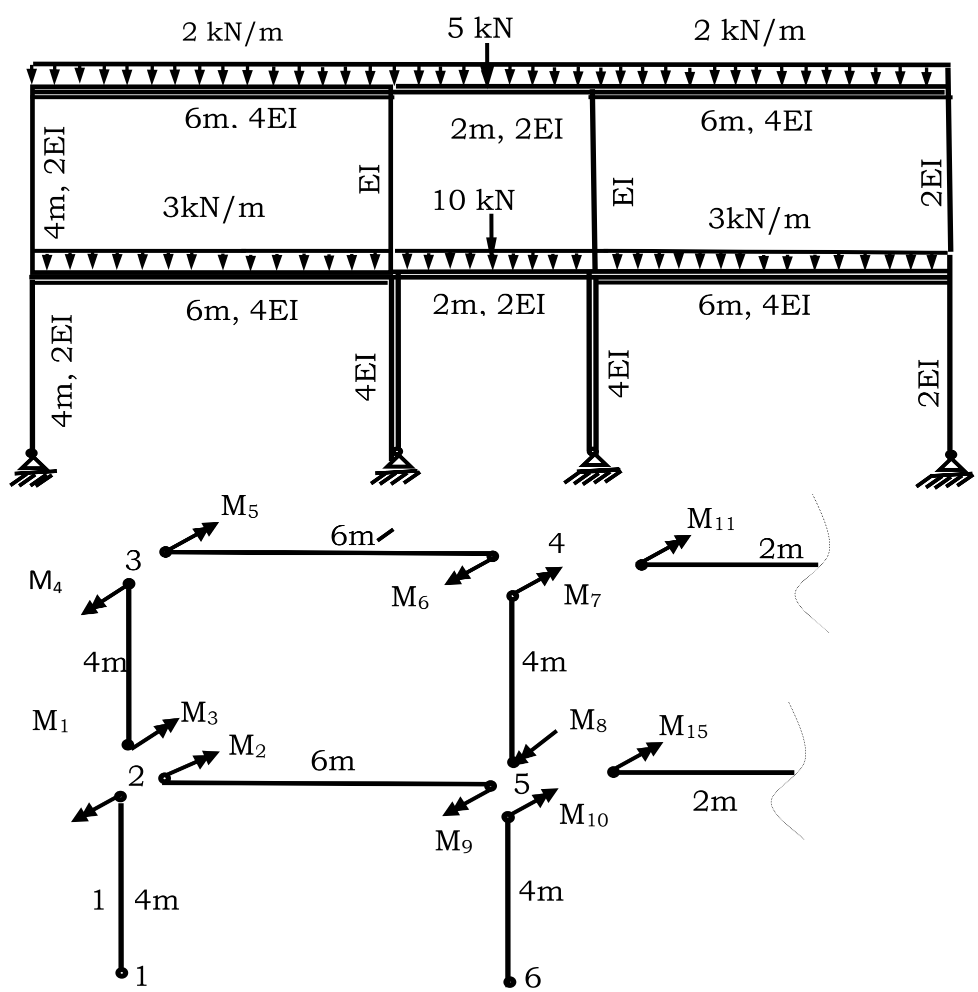


Fig. 7.34 Free Body Diagram for Half Structure

The plane frame has total 24 internal moments (M_1 to M_{24}) and total eight free rotational nodal displacements (θ_1 to θ_8). Thus total 16 compatibility conditions are developed using Matlab based .m file 'Mtechexamplemod'. The solution obtained by IFM based formulation for the displacements is compared with stiffness based STAAD.pro solution shown in **Table 7.1** whereas the solution for internal moments is compared with modified flexibility technique (MFT) and STAAD.pro solution in **Table 7.2.**

Table 7.1 Solution of Nodal Displacements Fixed Footed Portal Frame

Joint Rotation (radians)	IFM	STAAD.pro *v8i*
θ_2	-0.000770	-0.000792
θ_5	-0.000544	-0.000564
θ_8	0.000544	0.000564
θ_9	0.000770	0.000792
θ_3	-0.00071	0.00075
θ_4	-0.00049	0.00053
θ_7	0.00049	0.00053
θ_{10}	0.00071	0.00075

Table 7.2 Comparison of End Moments

Moments (kN-m)	Integrated Force Method (IFM)	Modified Flexibility Technique (MDT)	STAAD.pro *v8i*
M_1	-2.731	-2.63	-2.67
M_2	-6.672	-6.43	-6.55
M_3	3.940	-3.81	-3.98
M_4	-3.834	-3.74	-3.85
M_5	-3.834	-3.74	-3.85
M_6	-5.362	-5.61	-5.65
M_7	-2.082	-2.18	-1.988
M_8	2.113	2.00	2.32
M_9	-8.765	-8.94	-8.97
M_{10}	2.356	2.00	2.132

M_{11}	-3.641	-3.43	-3.465
M_{12}	-3.641	-3.43	-3.465
M_{13}	-2.032	-2.18	-2.21
M_{14}	-2.113	-2.00	-2.32
M_{15}	-4.887	-4.83	-4.89
M_{16}	-4.887	-4.83	-4.89
M_{17}	-5.362	-5.61	-5.65
M_{18}	-3.834	-3.74	-3.85
M_{19}	-3.834	-3.74	-3.85
M_{20}	3.9404	3.81	3.98
M_{21}	-8.665	-8.94	-8.97
M_{22}	-6.6721	-6.43	-6.55
M_{23}	2.856	2.00	2.13
M_{24}	-2.7316	-2.63	-2.67

7.5.3 Examples of Considering Axial Deformation

(i) IFM based Solution

A bent having two members is shown in Fig. 7.35. Calculate the internal forces and nodal displacements by considering axial deformation. Consider modulus of elasticity E as 2.01×10^8 kN/m^2 and cross sectional dimensions as 0.1m x 0.1m for each member.

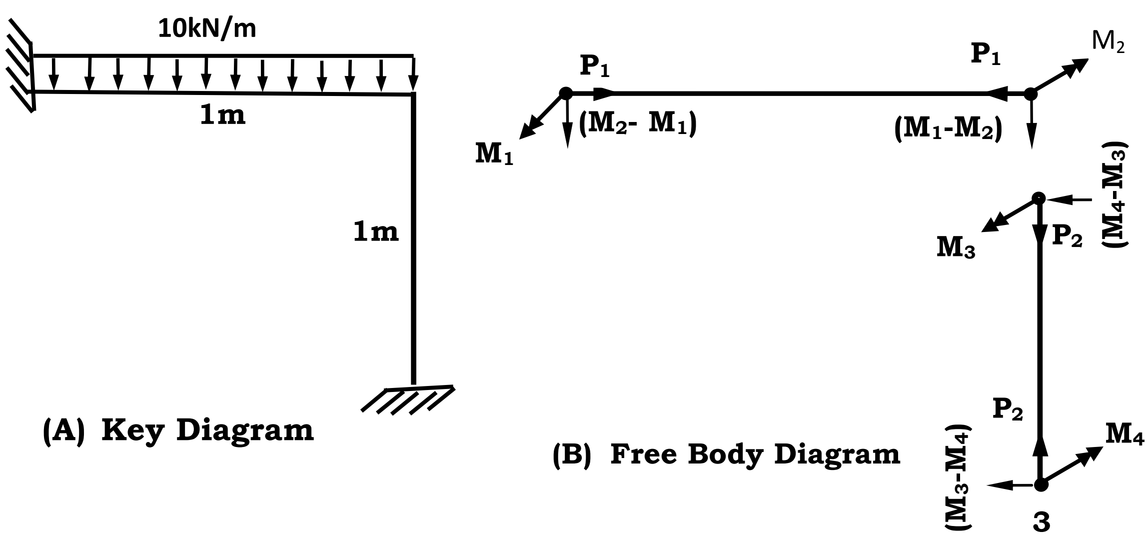


Fig.7.35 A Two Member Plane Frame (IFM)

Step 0 – Solution strategy: The plane frame has two axial forces P_1 and P_2 and four internal nodal moments as M_1, M_2, M_3 and M_4. Thus, there are total six unknowns. Due to axial deformation in the members total three displacements are possible at intermediate joint. As there is no sway, three compatibility conditions are required.

Step 1 – Formulate the equilibrium equations: The EEs can be written at joint '2' as

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 1 & -1 \\ 0 & 1 & -1 & 1 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 \end{bmatrix} \begin{Bmatrix} P_1 \\ P_2 \\ M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 0.00 \\ 0.00 \\ -0.833 \end{Bmatrix} \quad \ldots (7.24)$$

The Displacement Deformation Relations can be written as

$\beta_1 = \delta_H$, $\beta_2 = \delta_V$, $\beta_3 = -\delta_V$,

$\beta_4 = -\theta + \delta_V$, $\beta_5 = \delta_H + \theta$, and $\beta_6 = -\delta_H$

Step 3 – Generate the compatibility conditions: Calculation of the compatibility conditions is based on LIUT concept and can be written as

$$\begin{bmatrix} 7 & -1 & 8 & 9 & 9 & 2 \\ -5 & 3 & 9 & 6 & 6 & 1 \\ 4 & -2 & 3 & 5 & 5 & 9 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \end{Bmatrix} = [C]\{\beta\} \quad \ldots (7.25)$$

Step 4 – Formulate Force Deformation Relations: The force deformation relation for individual member can be written in terms of axial and bending rigidities as

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \end{Bmatrix} = 10^{-3} \begin{bmatrix} 0.0005 & 0 & 0 & 0 & 0 & 0 \\ & 0.0005 & 0 & 0 & 0 & 0 \\ & & 0.2 & 0.1 & 0 & 0 \\ & & 0.1 & 0.2 & 0 & 0 \\ & & & 0 & 0.2 & 0.1 \\ & & & & 0.1 & 0.2 \end{bmatrix} \begin{Bmatrix} P_1 \\ P_2 \\ M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = [G]\{F\} \quad \ldots (7.26)$$

Substituting {β} from Eq. (7.26) into Eq. (7.25), and after normalizing the matrix and concatenating into [B] matrix, the basic IFM relation can be written as

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 1 & -1 \\ 0 & 1 & -1 & 1 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 \\ -0.00035 & 0.0 & 0.25 & 0.26 & 0.2 & 0.13 \\ -0.0002 & 0.0001 & 0.24 & 0.21 & 0.13 & 0.08 \\ 0.0002 & -0.0001 & 0.11 & 0.13 & 0.19 & 0.23 \end{bmatrix} \begin{Bmatrix} P_1 \\ P_2 \\ M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 0.0 \\ 0.0 \\ -0.83 \\ 0.0 \\ 0.0 \\ 0.0 \end{Bmatrix} \quad \ldots (7.26)$$

Solving above equations, one can get values of internal axial forces and moments. After adding corresponding correction values to M_1 and M_2, the final values are found as follows:

$$\begin{Bmatrix} P_1 \\ P_2 \\ M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} -0.605 \\ -6.6543 \\ -1.0586 \\ 0.4043 \\ 0.4043 \\ -0.2007 \end{Bmatrix}$$

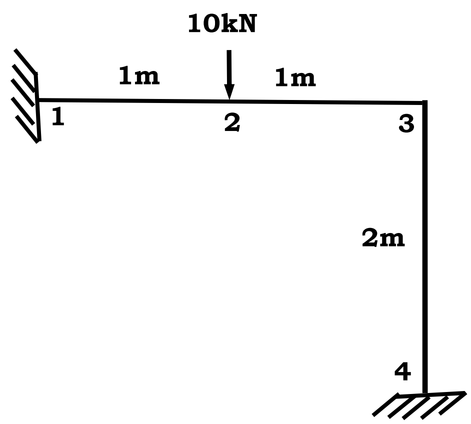
The result for displacements is found as follows:

$$\{\delta\} = \begin{Bmatrix} \delta_H \\ \delta_V \\ \Theta \end{Bmatrix} = \begin{Bmatrix} 0.003 \text{ m} \\ 0.026 \text{ m} \\ -0.611 \text{ rad} \end{Bmatrix} \times 10^{-4}$$

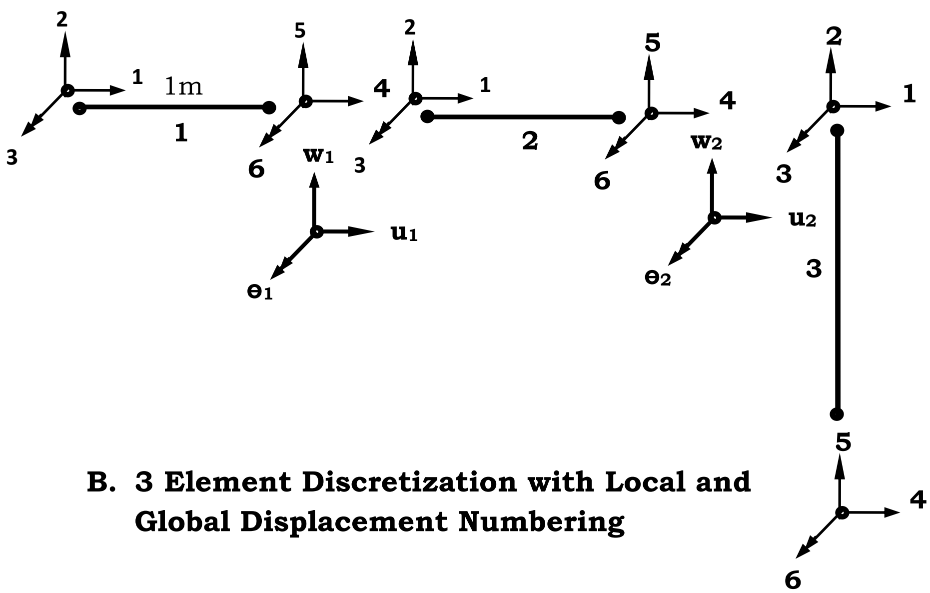
The above results obtained using IFM are found to exactly match with stiffness based solution.

(ii) DIFM Application to Plane Frame Example

A bent having two members is shown in **Fig.7.36.** The cross sectional dimensions of each one as 0.01m x 0.01m. All the internal forces and nodal displacements by considering axial deformation are worked out using the Dual Integrated Force Method (DIFM). Use E = 2.01x10^8 kN/m^2.



(A) Key Diagram Diagram



B. 3 Element Discretization with Local and Global Displacement Numbering

Fig. 7.36 A Two Member Plane Frame (DIFM)

Step 0 – Solution strategy: Three element discretization with local and global displacement numbering is shown in **Fig.7.36(B)**. It indicates that the frame has three numbers of axial forces (P_1, P_2 and P_3). There are total six internal moments (M_1 to M_6). As problem is solved using displacement based

DIFM approach all the equations are basically converted into mathematical form where global displacements (u_1, w_1, θ_1, u_2, w_2, θ_2) are the prime unknowns. Using these the internal moments are calculated.

Step 1 – Develop Elemental Matrices: Following are the major elemental matrices: (i) Basic Equilibrium Matrix [B_e] which is of size (m x n) where m is the number of local displacement degrees of freedom while n is the local internal unknowns per element, (ii) Basic Flexibility Matrix [G_e] which is of size (n x n) where n is the number of local internal unknowns per element and (iii) Elemental Pseudo stiffness Matrix $[D]_{difm(e)}$. The basic detail about calculation of these matrices is already given in Chapter 4. For example, for element 1 all the matrices are worked out by substituting a = 0.5m, EI = 0.1675 kN-m^2 and AE = 20100 kN as follows.

$$[Be] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 2 \\ 0 & -1 & 1 \\ -1 & 0 & 0 \\ 0 & 0 & -2 \\ 0 & 1 & 1 \end{bmatrix} \quad [Ge] = \begin{bmatrix} 4.97 \text{ x } 10^{-5} & 0 & 0 \\ 0 & 5.96967 & 0 \\ 0 & 0 & 1.989890 \end{bmatrix} \quad \ldots (7.27)$$

The Pseudo Stiffness matrix $[D]_{difm(1)}$ is of size 6 x 6 which is as follows.

$$\left[D_{difm(1)}\right] = 10^4 \begin{bmatrix} 2.01 & 0 & 0 & -2.01 & 0 & 0 \\ 0 & 0.0002 & 0.0001 & 0 & -0.0002 & 0.0001 \\ & & 0.0001 & 0 & -0.0001 & 0.0000335 \\ & & & 2.01 & 0 & 0 \\ & \text{sym} & & & 0.0002 & -0.0001 \\ & & & & & 0.0001 \end{bmatrix} \ldots (7.28)$$

Similarly, matrices are worked out for the elements 2 and 3.

Step 2 – Develop Dual Matrix $[D]_{difm}$: The dual matrix $[D]_{difm}$ which is of size 6 x 6 is formulated from the elemental matrices following the process quite similar to regular stiffness approach.

Step 3 – Calculate unknowns: The DIFM equation based on stiffness analogy is given by

$$[D]_{difm}\{\delta\} = \{P\} \quad \ldots (7.29)$$

Where $[D]_{difm}$ is the global pseudo stiffness matrix of size 6 x 6 along global displacement degrees of freedom at joints 2 and 3, $\{\delta\}$ is the unknown displacement vector of size 6 x 1 and $\{P\}$ is the load vector of size 6 x 1.

After substituting all the necessary matrices into Eq. (7.29) and solving for displacements gives

$$\begin{Bmatrix} u_2 \\ w_2 \\ \theta_{\theta_2} \end{Bmatrix} = \begin{Bmatrix} -0.000098\,m \\ -0.00004\,m \\ +3.5276\,rad \end{Bmatrix} \quad \ldots (7.30)$$

The solution obtained is matching with stiffness based approach **[92]**. Once the displacement vector is available, the internal moments are worked out using either global matrix approach or individual element approach using the following equation:

$$\{F\} = [G]^{-1}[B]^T\{\delta\} \quad \ldots (7.31)$$

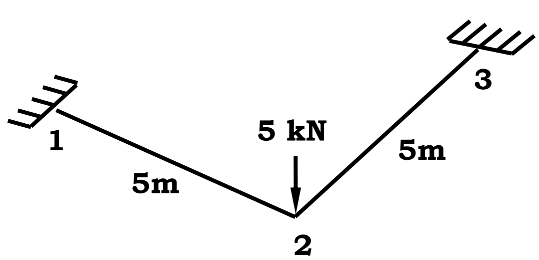
The internal actions at right joint of element 2 are as follows

$$\begin{Bmatrix} P_2 \\ V_2 \\ M_2 \end{Bmatrix} = \begin{Bmatrix} -0.96658 \\ +4.002 \\ -1.276 \end{Bmatrix}$$

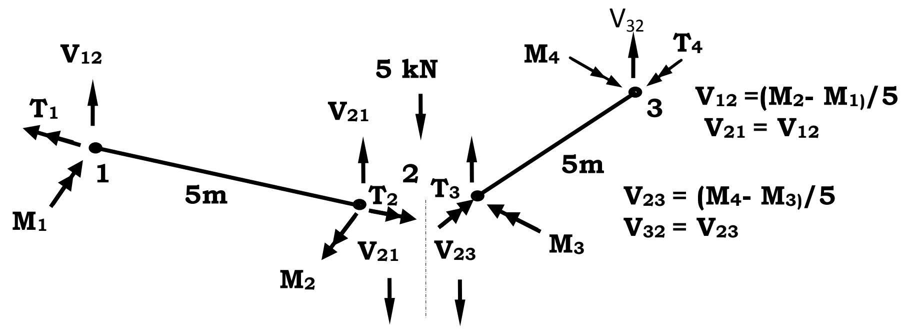
7.6 EXAMPLES OF STATIC ANALYSIS OF GRIDS

7.6.1 Grid Example 1

A grid structure consisting of two members 1-2 and 2-3, subjected to a point load 10kN at the junction, as shown in **Fig. 7.37** is considered here with flexural and torsional rigidities (EI and GJ) equal to1666.67 kN-m^2 for each member.



A. Key Diagram



B. Free Body Diagram

Fig. 7.34 Grid Example

Step 0 — Set up Equilibrium Equations By referring **Fig. 7.37** three equilibrium equations can be written in matrix form as [B]{F} = {P}.

$$\begin{bmatrix} 0 & 0 & 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 & 0 & 0 \\ 1/5 & 0 & -1/5 & 0 & -1/5 & 0 & 1/5 & 0 \end{bmatrix} \begin{Bmatrix} M_1 \\ T_1 \\ M_2 \\ T_2 \\ M_3 \\ T_3 \\ M_4 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10 \end{Bmatrix} \qquad \ldots (7.32)$$

Step 1 — Relate deformation to displacements: The displacement deformation relation $\{\beta\} = [B]^T\{\delta\}$ can be formulated by using equilibrium matrix [B]. There are three displacements $\{\delta\}$: two rotations and one translation. The eight deformations $\{\beta\}$ are induced by eight unknown moments (M_1, T_1, M_2, T_2, M_3, T_3, M_4 and T_4). The displacement deformation relation can be written as

$$\begin{pmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \\ \beta_7 \\ \beta_8 \end{pmatrix} = \begin{bmatrix} 0 & 0 & 1/5 \\ 0 & 0 & 0 \\ 1 & 0 & -1/5 \\ 0 & -1 & 0 \\ 0 & 1 & -1/5 \\ -1 & 0 & 0 \\ 0 & 0 & 1/5 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} \theta_M \\ \theta_T \\ \delta_V \end{Bmatrix} \qquad \ldots (7.33)$$

Step 2— Generate Compatibility Conditions Five relations between the eight deformations can be obtained by eliminating three displacements from the DDR as follows.

$$\begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 1 & 1 & 1 & 1 & 1 & 1 & 1 & 1 \\ 2 & 0 & 1 & 0 & 0 & 1 & -1 & 0 \\ 0 & 0 & 1 & 1 & 1 & 1 & 2 & 0 \end{bmatrix} \begin{pmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \\ \beta_7 \\ \beta_8 \end{pmatrix} = \begin{Bmatrix} O \\ O \\ O \\ O \\ O \\ O \\ O \\ O \end{Bmatrix} \qquad \dots (7.34)$$

Step 3 — Write force deformation relation: Force deformation relation can be written in terms of [G] matrix for bending moments Mi, Mj and for twisting moments Ti, Tj respectively.

$$\beta i = \frac{L}{6EI}(2Mi + Mj), \qquad \beta j = \frac{L}{6EI}(2Mj + Mi)$$

$$\beta i = \frac{TiL}{GJ}, \qquad \beta j = \frac{TjL}{GJ}$$

From the above relation the flexibility matrix [G] can be written as

$$[G] = \frac{L}{6EI}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}\begin{Bmatrix} M_i \\ M_j \end{Bmatrix} \qquad \dots (7.35)$$

Step 4 — Express Compatibility in terms of Forces The compatibility conditions are expressed in terms of forces by eliminating the deformations.

$$\begin{bmatrix} 1 & 2 & 1 & 2 & 1 & 2 & 1 & 2 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 5 & 0 & 4 & 0 & -1 & 6 & -2 & 0 \\ 1 & 0 & 2 & 6 & 4 & 6 & 5 & 0 \end{bmatrix} \begin{Bmatrix} M_1 \\ T_1 \\ M_2 \\ T_2 \\ M_3 \\ T_3 \\ M_4 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \qquad \dots (7.36)$$

Step 5 — Calculate Unknowns: The equilibrium equations and compatibility conditions can be coupled to obtain the unknowns.

$$\begin{bmatrix} 0 & 0 & 1 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1 & 0 & 0 & 0 \\ 1/5 & 0 & -1/5 & 0 & -1/5 & 0 & 1/5 & 0 \\ 1 & 2 & 1 & 2 & 1 & 2 & 1 & 2 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 \\ 5 & 0 & 4 & 0 & -1 & 6 & -2 & 0 \\ 1 & 0 & 2 & 6 & 4 & 6 & 5 & 0 \end{bmatrix} \begin{Bmatrix} M_1 \\ T_1 \\ M_2 \\ T_2 \\ M_3 \\ T_3 \\ M_4 \\ T_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -5 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \quad \ldots (7.37)$$

Solution of above set of equations gives results for internal bending and twisting moments which are reported in **Table 7.3.**

Table 7.3 Internal Moments

Moments	M_1	T_1	M_2	T_2	M_3	T_3	M_4	T_4
IFM (kN-m)	-9.375	3.125	-3.125	3.125	-3.125	3.125	-9.375	3.125

The nodal displacements at joint 2 are calculated using [J][G]{M} and are found as follows:

$$\begin{Bmatrix} \theta_M \\ \theta_T \\ \delta_V \end{Bmatrix} = \begin{Bmatrix} -0.0094 \\ +0.0094 \\ -0.0391 \end{Bmatrix} \quad \ldots (7.37)$$

7.6.2 Grid Example 2

A grid structure is subjected to UDL of 10 kN/m is shown in **Fig. 7.38.** Considering for both the members EI and GJ = 1666.67 kN-m^2, calculate the internal moments and nodal displacements.

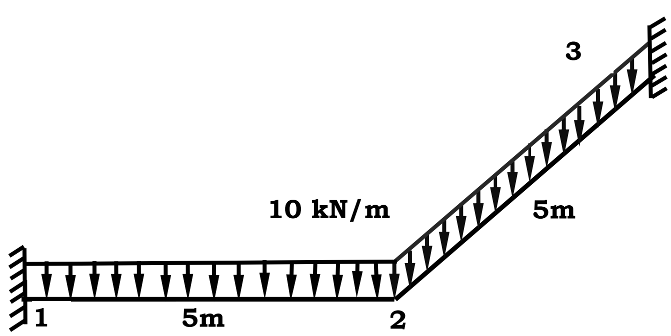


Fig. 7.38 Key diagram for Grid Example 2

Following the procedure outlined in the previous example, the solution obtained for internal moments presented in **Table 7.4** and is compared with the stiffness based solution.

Table 7.4 Joint Moments

Moments (kN-m)	IFM	Stiffness Method
M_1	-104.165	-104.165
T_1	-20.833	-20.833
M_2	20.833	20.833
T_2	-20.833	-20.833
M_3	20.833	20.833
T_3	20.833	20.833
M_4	-104.165	-104.165
T_4	-20.833	-20.833

Solution for the nodal displacements is as follows:

$$\begin{Bmatrix} \theta_M \\ \theta_T \\ \delta_V \end{Bmatrix} = \begin{Bmatrix} -0.125 \\ +0.125 \\ -0.4687 \end{Bmatrix} \quad \text{... (7.38)}$$

7.7 EXAMPLE OF STATIC ANALYSS OF SPACE TRUSS

A pin jointed space structure consisting of eight members subjected to point loads of 10 kN at the two top most points of the truss is shown in **Fig.7.39** Considering axial rigidity (AE) as 2.01 x 10^6 kN the axial force in each member is worked out.

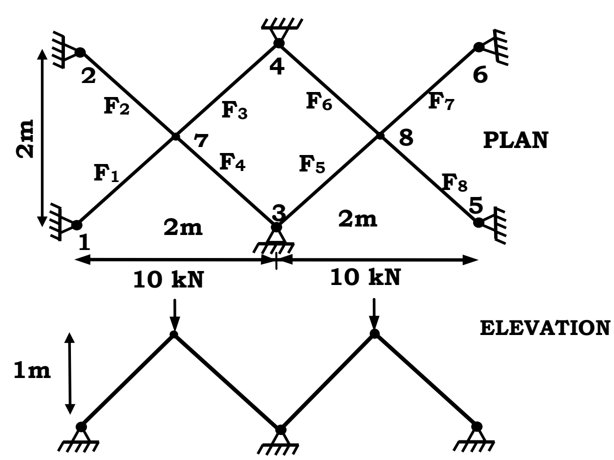


Fig. 7.39 Space Truss Example

Step 1— Set up equilibrium Equations: Six three equilibrium equations 6 free displacements can be written in matrix form as

$$\begin{bmatrix} 0.57 & 0.57 & -0.57 & -0.57 & 0 & 0 & 0 & 0 \\ 0.57 & -0.57 & 0.57 & -0.57 & 0 & 0 & 0 & 0 \\ 0.57 & 0.57 & 0.57 & 0.57 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.57 & 0.57 & -0.57 & -0.57 \\ 0 & 0 & 0 & 0 & 0.57 & -0.57 & 0.57 & -0.57 \\ 0 & 0 & 0 & 0 & 0.57 & 0.57 & 0.57 & 0.57 \end{bmatrix} \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10 \\ 0 \\ 0 \\ -10 \\ 0 \end{Bmatrix} \qquad \ldots (7.39)$$

Step 2— Relate deformation to displacement: The displacement deformation relation ($\{\beta\} = [B]^T\{x\}$) can be written by using equilibrium matrix [B]. There are three displacements at each free joint. The eight deformations $\{\beta\}$ induced in 8 members can be related to six joint displacements and the DDR can be written as

$$\begin{pmatrix} \beta 1 \\ \beta 2 \\ \beta 3 \\ \beta 4 \\ \beta 5 \\ \beta 6 \\ \beta 7 \\ \beta 8 \end{pmatrix} = \begin{bmatrix} 0.57 & 0.57 & 0.57 & 0 & 0 & 0 \\ 0.57 & -0.57 & 0.57 & 0 & 0 & 0 \\ -0.57 & 0.57 & 0.57 & 0 & 0 & 0 \\ -0.57 & -0.57 & 0.57 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0.57 & 0.57 & 0.57 \\ 0 & 0 & 0 & 0.57 & -0.57 & 0.57 \\ 0 & 0 & 0 & -0.57 & 0.57 & 0.57 \\ 0 & 0 & 0 & -0.57 & -0.57 & 0.57 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \cdot \\ \cdot \\ \delta_8 \end{Bmatrix} \qquad \ldots (7.40)$$

Step 3 — Generate compatibility conditions: Two relations between the six deformations can be obtained by eliminating six displacements from the DDR as follows:

$$\begin{bmatrix} 8 & -8 & -8 & 8 & 9 & -9 & -9 & 9 \\ 2 & -2 & -2 & 2 & 9 & -9 & -9 & 9 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \cdot \\ \cdot \\ \beta_8 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \qquad \ldots (7.41)$$

Step 4 — Write force deformation relation: Force deformation relation to define [G] can be written in terms of L, A and E as FL/AE.

Step 5 — Express compatibility of in terms forces: The compatibility conditions are expressed in terms of forces by eliminating the deformations as under.

$$10^{-5} x \begin{bmatrix} 0.6984 & -0.6984 & -0.6984 & 0.6984 & 0.7755 & -0.7755 & -0.7755 & 0.7755 \\ 0.1723 & -0.1723 & -0.1723 & 0.1723 & 0.7755 & -0.7755 & -0.7755 & 0.7755 \end{bmatrix} \begin{Bmatrix} F_1 \\ . \\ . \\ F_8 \end{Bmatrix} \quad \ldots (7.42)$$

Step 6 — Calculate unknowns: The equilibrium equations and compatibility conditions can be coupled to obtain the unknowns as follows

$$\begin{bmatrix} 0.57 & 0.57 & -0.57 & -0.57 & 0 & 0 & 0 & 0 \\ 0.57 & -0.57 & 0.57 & -0.57 & 0 & 0 & 0 & 0 \\ 0.57 & 0.57 & 0.57 & 0.57 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.57 & 0.57 & -0.57 & -0.57 \\ 0 & 0 & 0 & 0 & 0.57 & -0.57 & 0.57 & -0.57 \\ 0 & 0 & 0 & 0 & 0.57 & 0.57 & 0.57 & 0.57 \\ 1 & -1 & -1 & 1 & 0 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 1 & -1 & -1 & 1 \end{bmatrix} \begin{Bmatrix} \mathbf{F_1} \\ \mathbf{F_2} \\ \mathbf{F_3} \\ \mathbf{F_4} \\ \mathbf{F_5} \\ \mathbf{F_6} \\ \mathbf{F_7} \\ \mathbf{F_8} \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10 \\ 0 \\ 0 \\ -10 \\ 0 \\ 0 \end{Bmatrix} \quad \ldots (7.43)$$

The internal forces in different members of the space truss obtained by IFM are found to be matching with those obtained by flexibility and stiffness methods and are as follows

$$\{F\} = [-4.33\ 4.33\ -4.33\ 4.33\ -4.33\ 4.33\ -4.33\ 4.33]^T \quad \ldots (7.44)$$

The nodal displacement at joints 7 and 8 are found as follows:

$$\begin{Bmatrix} \delta_{7X} \\ \delta_{7Y} \\ \delta_{7Z} \\ \delta_{8X} \\ \delta_{8Y} \\ \delta_{8Z} \end{Bmatrix} = \begin{Bmatrix} 0.00 \\ 0.00 \\ -0.6463 \\ 0.00 \\ 0.00 \\ -0.6463 \end{Bmatrix} x\ 10^{-5} m \quad \ldots (7.45)$$

7.8 EXAMPLE OF STATIC ANALYSIS OF SPACE FRAME

7.8.1 Space Frame Example 1

A rigid jointed space frame structure consisting of three members orthogonal to each other is subjected to two point loads of 10 kN each at the junction of members as shown in **Fig. 7.40.** Considering axial rigidity (AE), bending rigidity (EI) and torsional rigidity (GJ) as 1.578 x 10^6 kN, 986.65 kN-m^2 and 758.953 kN-m^2 respectively, evaluate the internal forces developed in each member and nodal displacements.

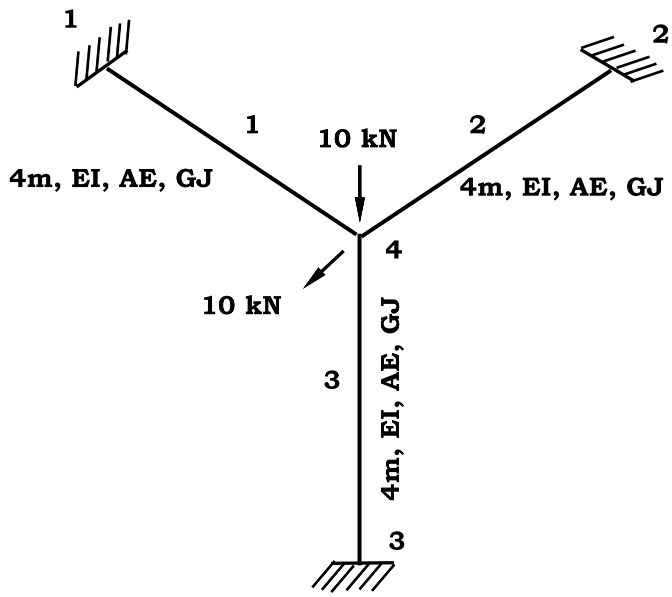


Fig. 7.40 Space Frame Example

Step 0 – Solution strategy: The space frame has 6 numbers of axial forces i.e. P_1 to P_6, 6 numbers of torsional moments i.e. T_1 to T_6 and total 12 flexural moments. At joint 4 as there are total six possible displacements, which is of lateral and rotational along each x-x, y-y and z-z directions. Thus the problem needs total 18 compatibility conditions to solve the complete problem using IFM based formulations.

Steps 1 – Formulate the equilibrium equations: The six EEs can be written in terms of internal forces at joint 4 along all the displacement direction, which can be written as

$$
\begin{bmatrix}
-1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{-1}{4} & \frac{1}{4} & \frac{1}{4} & \frac{-1}{4} & 0 & 0 \\
0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{1}{4} & \frac{-1}{4} & 0 & 0 & 0 & 0 & \frac{1}{4} & \frac{-1}{4} \\
0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & \frac{-1}{4} & \frac{1}{4} & \frac{1}{4} & \frac{-1}{4} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & -1 & 0 & 0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & -1 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
= \begin{Bmatrix} P_1 \\ . \\ . \\ P_6 \\ T_1 \\ . \\ . \\ T_6 \\ M_1 \\ . \\ . \\ M_{12} \end{Bmatrix}
\begin{Bmatrix} 0.00 \\ -10.00 \\ -10.00 \\ . \\ . \\ 0.00 \end{Bmatrix}
$$

... (7. 46)

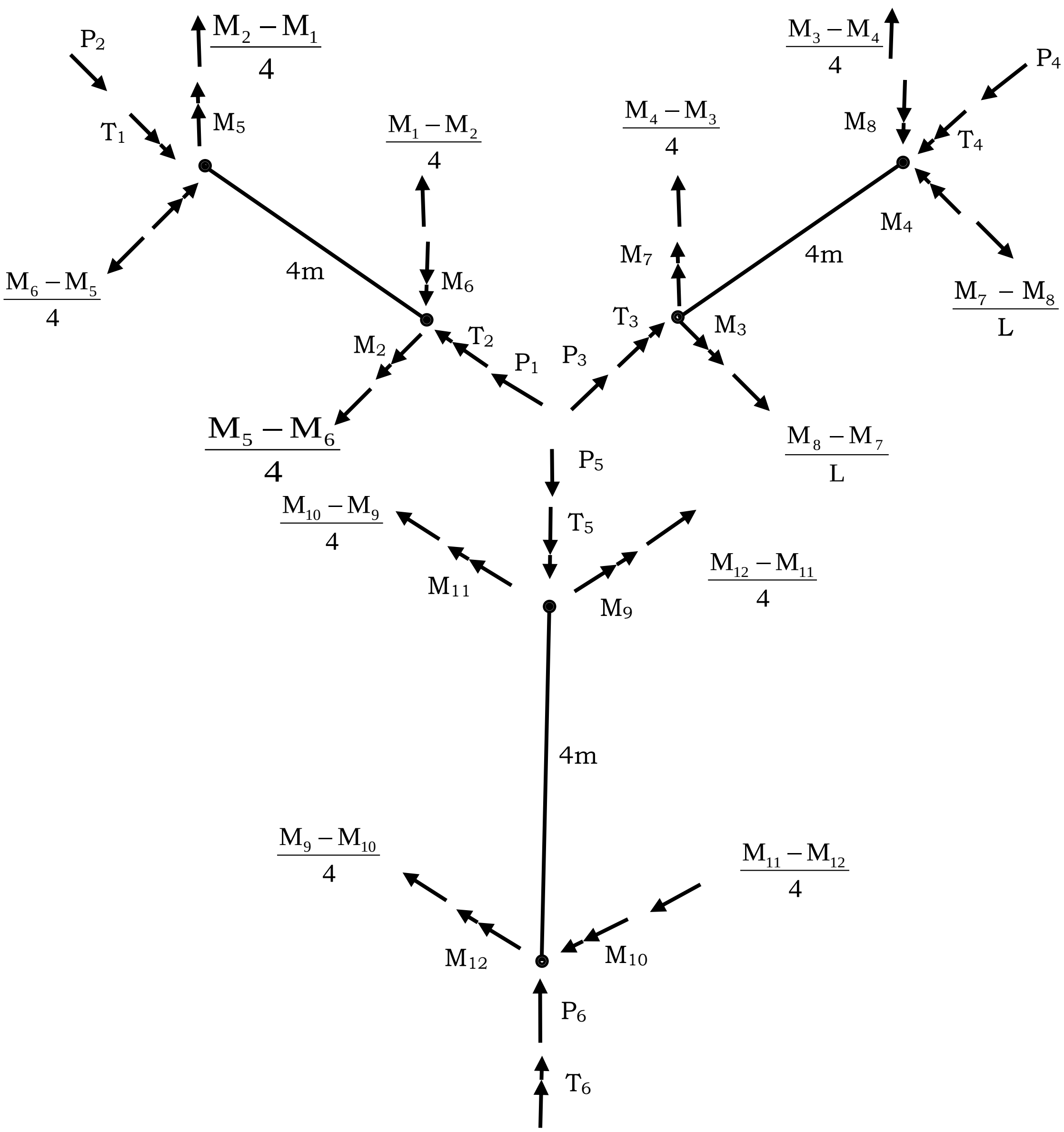


Fig. 7.41 Free Body Diagram

Using .m file total 18 compatibility conditions are developed. These are converted into the force deformation relation by multiplying by global flexibility which is of size (24 x 24). After normalizing the major components [C] matrix is concatenated to [B]matrix which yields a global equilibrium matrix [S]. Inverting [S] matrix using Matlab and solving for the unknowns one gets the solution for internal actions as reported in **Table 7.8.**

Table 7.8 Internal Forces

P_1 (kN)	P_2	P_3	P_4	P_5	P_6
0.00315	0.00315	9.9954	-9.9954	-9.9954	-9.9954
T_1(kN-m)	T_2	T_3	T_4	T_5	T_6
0.0016	0.0016	0.0008	0.0008	-0.0008	0.0008
M_1(kN-m)	M_2	M_3	M_4	M_5	M_6
0.0072	0.0051	0.0008	0.0051	0.0072	0.0051
M_7(kN-m)	M_8	M_9	M_{10}	M_{11}	M_{12}
0.0042	0.0021	0.0042	0.0021	0.0008	0.0051

Finally, the nodal displacement vector {δ} is worked out by using [J][G]{F}, which is as follows:

$$\begin{Bmatrix} \delta_{xx} \\ \delta_{yy} \\ \delta_{zz} \\ \Theta_{xx} \\ \Theta_{yy} \\ \Theta_{zz} \end{Bmatrix} = \begin{Bmatrix} -0.0001 \text{ x } 10^{-4}\text{m} \\ 0.2533 \text{ x } 10^{-4}\text{m} \\ 0.2533 \text{ x } 10^{-4}\text{m} \\ 0.0866 \text{ x } 10^{-4} \text{ rad} \\ -0.0433 \text{ x } 10^{-4}\text{rad} \\ 0.0433 \text{ x } 10^{-4} \text{ rad} \end{Bmatrix}$$

7.9 CONSIDERATION OF SECONDARY EFFECTS USING IFM

Deformation $\{\beta\}_i$ in the structure on account of the secondary effects such as temperature change, support settlement and prestrain effect can be easily included on the right side of the compatibility conditions of IFM through the effective initial deformation vector {δR}.

Such deformations, when due to thermal effects, represent temperature strains, which can be written as the product of coefficient of thermal expansion 'α' and the temperature change 'T' for the element length L as

$\{\beta\}_i = \alpha LT$ ... (7.48)

Initial deformation $\{\beta\}_i$ due to support settlement 'Δ' can be calculated from

$\{\beta\}_i = -\{B_r\}\,\Delta$ ... (7.49)

where $\{\beta_r\}$ represents transpose of row of equilibrium matrix related to reaction R at the point of support settlement.

The prestrain effect in the truss member can be tackled in the manner analogous to the temperature effect with the difference that the deformation term corresponding to temperature effect (αLT) is replaced now by extra length 'e' of the member.

An example of pin jointed structure is included here to demonstrate how temperature and prestrain effects are considered in IFM based methodology.

7.9.1 Illustrative Example of Temperature Effect

A three bar truss is subjected to temperature T_1, T_2 and T_3 in the three bars respectively in addition to nodal point loads as shown in Fig. 3. The truss is made of steel and has modulus of elasticity $E = 2 \times 10^5$ N/mm^2 and coefficient of thermal expansion $\alpha = 6.0 \times 10^{-6}/\,^\circ C$. The area of the three bars AD, BD and CD is respectively 500 mm^2, 400 mm^2, 400 mm^2. For finding the forces in the members of the truss the steps are as follows:

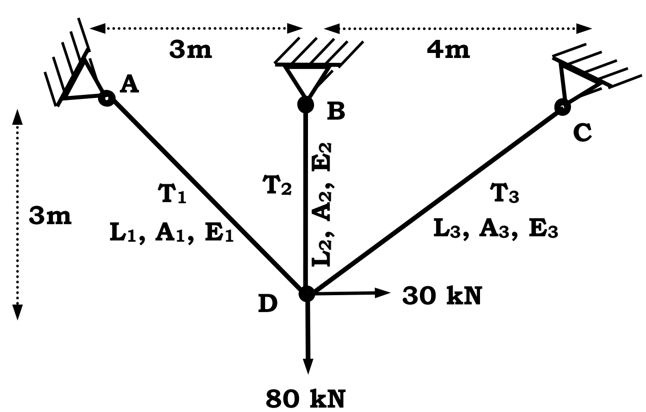


Fig. 7.42 Three Bar Truss Example

Step 1 – Equilibrium equations: Two EE for the problem can be obtained by force balance condition at the node D. In matrix notation ([B] {F} = {P}) can be written as

$$\begin{bmatrix} -0.707 & 0 & 0.8 \\ 0.707 & 1 & 0.6 \end{bmatrix} \begin{Bmatrix} F1 \\ F2 \\ F3 \end{Bmatrix} = \begin{bmatrix} 30 \\ 80 \end{bmatrix}$$

Step 2 – Find the deformation displacement relations: The DDRs are obtained as

$(\{\beta\} = [B]^T \{\delta\})$ where $\beta_1 = -0.707\delta_1 + 0.707\,\delta_2$,

$\beta_2 = \delta_2$ and $\beta_3 = 0.8\,\delta_1 + 0.8\delta_2$.

Step 3 – Generate the compatibility conditions: The CC for the problem can be written as [C] {β} = {0} where [C] = [0.8 -0.98 0.707]

Step 4 – Formulate force deformation relations: The FDR for the bars of the truss can be written as β = FL/AE . In matrix notations the FDR can be written as

$$\begin{bmatrix} 8.48 & 0 & 0 \\ 0 & 7.5 & 0 \\ 0 & 0 & 12.5 \end{bmatrix} \begin{Bmatrix} F_{AD} \\ F_{BD} \\ F_{CD} \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \end{Bmatrix}$$

Step 5 – Express the compatibility conditions in terms of forces:

Thermal analysis requires the inclusion of additional vector {δR} on the right side of CC i.e {δR} = – [C] $\{\beta\}_i$ where $\{\beta\}_i$ is the deformation vector corresponding to the thermal effect induced in the bars of the truss and it can be written as

$$\{\beta\}_i = \begin{Bmatrix} \beta 1_i \\ \beta 2_i \\ \beta 3_i \end{Bmatrix} = \alpha \begin{Bmatrix} T_1L_1 \\ T_2L_2 \\ T_3L_3 \end{Bmatrix}$$

and hence $\{\delta R\} = -[0.8\ -0.98\quad 0.707] \times \alpha \begin{Bmatrix} T_1L_1 \\ T_2L_2 \\ T_3L_3 \end{Bmatrix}$

with $T_1 = T_2 = T_3 = 80^oc$,

$$\{\delta R\} = -[0.8 \quad -0.98 \quad 0.707] \times 6.0 \times 10^{-6} \times 80 \begin{Bmatrix} 4.24 \\ 3.0 \\ 5.0 \end{Bmatrix} = -0.0019.$$

The compatibility conditions in terms of force variables can be written as

$$\frac{1}{E}[6.784 \quad -7.35 \quad 8.837] \begin{Bmatrix} F_{AD} \\ F_{BD} \\ F_{CD} \end{Bmatrix} = \{-0.0019\}$$

Steps 6 – Couple the Equilibrium equations and compatibility conditions: Coupling of EE and CC provides the following set of equations to solve for unknowns.

$$\begin{bmatrix} 0.707 & 0 & -0.8 \\ 0.707 & 1 & 0.6 \\ 6.784 & -7.35 & 8.837 \end{bmatrix} \begin{Bmatrix} F_{AD} \\ F_{BD} \\ F_{CD} \end{Bmatrix} = \begin{Bmatrix} 30 \\ 80 \\ -382.66 \end{Bmatrix}$$

The solution of set of equation provides bar forces as F_{AD} = 29.64 kN, F_{BD} = 65.827 kN and F_{CD} = -11.305 kN which is found to match exactly with the solution obtained using Stiffness method of analysis.

7.9.2 Illustrative Example of Prestrain Effect

In above example, if the member AD is shorter than its actual length by 3 mm (e = 3 mm), then all other steps being the same FDR, can be written now as

$$\{\delta R\} = -[0.8 \quad -0.98 \quad 0.707] \times \begin{Bmatrix} 3.0 \\ 0 \\ 0 \end{Bmatrix} = -0.0024$$

The CC in terms of FDR can be written as

$$\frac{1}{E}[6.784 \quad 7.35 \quad 8.837] \begin{Bmatrix} F_{AD} \\ F_{BD} \\ F_{CD} \end{Bmatrix} = \{-0.0024\}$$

Coupling FDR with EE leads to

$$\begin{bmatrix} -0.707 & 0 & 0.8 \\ 0.707 & 1 & 0.6 \\ 6.784 & -7.35 & 8.837 \end{bmatrix} \begin{Bmatrix} F_{AD} \\ F_{BD} \\ F_{CD} \end{Bmatrix} = \begin{Bmatrix} 30 \\ 80 \\ -480 \end{Bmatrix}$$

Solution of above equations gives forces in different members as F_{AD} = 25.531 kN, F_{BD} = 70.911 kN and F_{CD} = -14.931 kN. Forces in different members of the truss are oufn tomach exactly with those obtained with the stiffness method of analysis.

7.10 DISCUSSION OF RESULTS

- Solution of continuous beam problems in terms of joint and support moments as primary unknowns and nodal displacements as secondary unknowns is found in close agreement with those obtained using other matrix methods.
- For the beam members subjected to point load, it is preferable to discretize into two elements, to simplify generation of the load vector. Also, it is possible for simply supported extreme end to set reacting moment directly equals to zero and thus in a matrix corresponding row and column can be removed to reduce the size of matrix
- The Dual Integrated Force Method (DIFM) is mathematically modified version of IFM, in which nodal displacements are calculated first and then after substituting the displacements in basic equation internal forces are worked out. The solution for any problem using IFM and DIFM is always identical. The mathematical characteristics of dual matrix $[D]_{difm,}$ which is used in DIFM, are same as that of conventional displacement based stiffness matrix.
- Mathematical validity of [C]{β} = 0 is easy to check using the code developed as " mtechexamplemod(B)".
- As the results for pin-jointed structures are worked out directly in global displacement directions, the need of transformation of axis from local to global is not required. The solution obtained for internal forces and

nodal displacements is found to match with the solution obtained by using Stiffness method.

- A plane frame structure is attempted by considering and neglecting axial deformation. In both the cases, solution obtained using IFM are found to match with those obtained using stiffness method of analysis.
- In case of IFM the shearing force can be represented as the ratio of the net moment and length of the member to reduce the numerical work.
- The use of concept of symmetry which has been successfully demonstrated with the help of a plane frame example is applicable to all types of framed structures.
- IFM based solutions for the grid, pin-jointed and rigid jointed space structures are found to match with the solutions available in the literature.
- Results obtained for 3 member truss under temperature and prestrain effects are found to match exactly with the results obtained using stiffness method.
- Matlab based module named as "mtechexamplemod(B)", which calculates the coefficients corresponding to each deformation of each force variable using LIUT technique, gears up the total numerical work independent of sparsity of large size equilibrium matrix. The provision of auto selection of dependent and independent coefficients is also found advantageous for maintaining the overall accuracy of the solution.

CHAPTER 8

DYNAMIC ANALYSIS OF FRAMED STRUCTURES

8.1 SCREEN SHOTS OF COMPUTER IMPLEMENTATION

A preprocessor is developed in VB6 to facilitate data input through a number of forms developed for the dynamic analysis of different types of framed structures. Using menu editor and multiple display interface one can link different forms by hiding and un-hiding operations depending upon logic of the program. After invoking the preprocessor the first step is selection of type of structure out of the group of framed structures i.e. Beam, Plane Truss, Plane Frame, Grid, Space Truss and Space Frame. Once the structure is selected by the user, input data is supplied using Labels, Textboxes which are linked to different forms. The geometrical and material properties are supplied using various text boxes. Support conditions are selected by assigning property through the next form. Visual feedback is provided for each user action. The next form highlights the key diagram which enables the complete geometry of the structure.

By clicking the command button "Development of Matrices", various necessary properties are transferred from Form 1 to Form 3 and Form 2 to Form 3. This command button develops basic equilibrium matrix [B] and necessary global flexibility matrix [G] depending upon the basic unknowns 'm' and 'n', which are free nodal displacements and internal unknowns nodal moments respectively. Coding is written for development of text file which enables us to supply both the matrices named as IFM_FRAME.txt and Gmatrix.txt. The lumped mass matrix [M] is directly written in the main processor. The connection between VB 6 and Matlab is done through COM approach by linking an automation server in the given program.

Once the major matrices are developed the mathematical and matrix based operations are carried out in the main processor part of the software. Matlab facilitates a temporary command window which is developed by using interface procedure. The basic matrices [B] and [G] are activated by calling a text file. The lumped Mass matrix [M] is directly typed using command window. The Matlab in-built editor enables various types matrix based operations at the command prompt i.e. matrix addition, multiplication, transpose, inverse and eigen operator.

The step-by-step procedure followed for the dynamic analysis is illustrated here with the help of an example of a propped cantilever beam.

A mild steel propped cantilever beam having two elements is separated by lumped mass M_o = 10 kN-Sec^2/m. Each element is of 1m length having flexural rigidity EI as 1666.67 kN-m^2 as shown in **Fig. 8.1.**

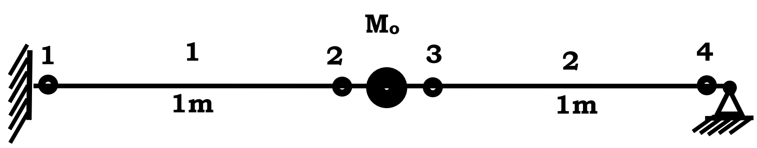


Fig. 8.1 Propped Cantilever Beam

Step1: From MDI form 1 main menu the type of structure is selected as beam as depicted in **Fig. 8.2.**

Fig.8.2 Selection of Type of Structure

Step 2: From beam option which displays different types of beams by using check box button propped cantilever case is selected. Length and flexural rigidity of each span are entered as shown in **Fig. 8.3.**

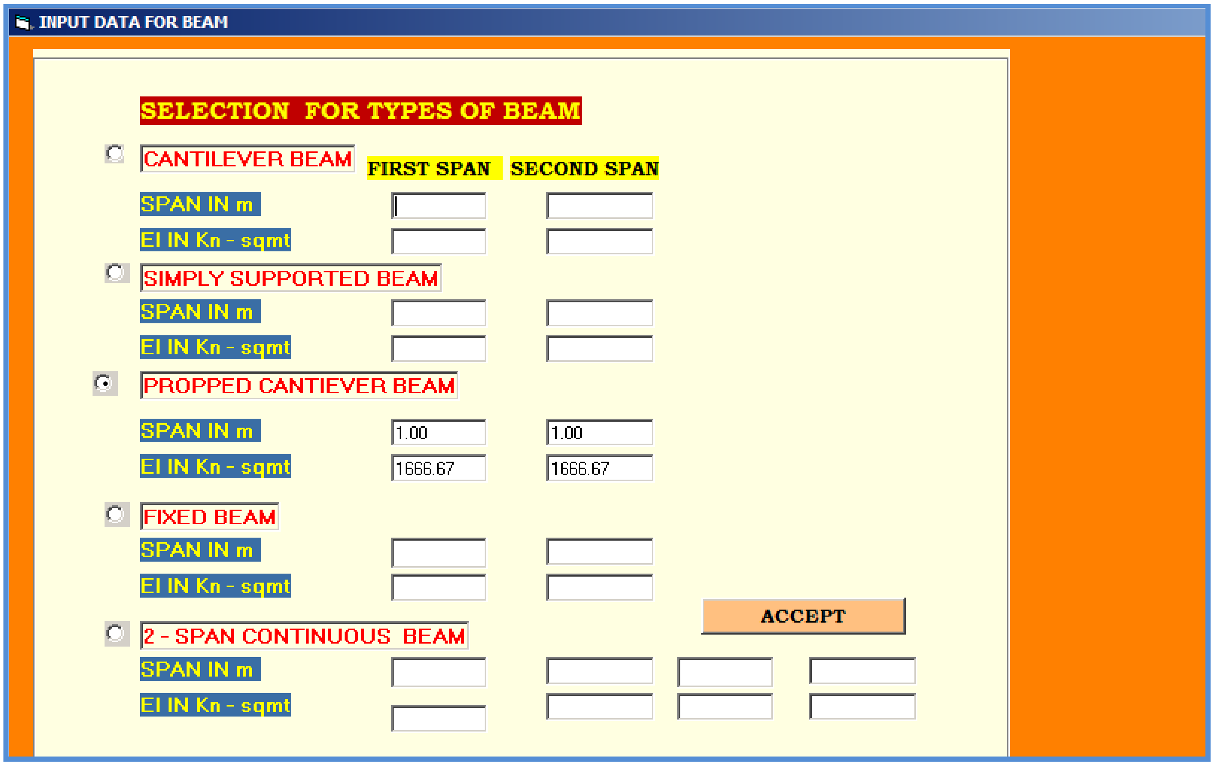


Fig. 8.3 Geometrical and Flexural Properties of Beam

Step 3: Once the entered data is accepted it depicts on screen the next form related to boundary conditions. The desired support condition is selected as depicted in **Fig. 8.4.** Selection of plot button draws the beam on the screen and clicking next button lead to the next form.

Fig. 8.4 Beam with Boundary Conditions

Step 4: Form 3 shows the button for drawing key diagram of the beam in addition to button for the calculation of the different matrices such as Basic

Equilibrium Matrix [B], Global Flexibility Matrix [G] and Mass Matrix [M] as shown in **Fig. 8.5**. In the same form clicking next button connects VB6 to Matlab using COM procedure as shown in **Fig. 8.6.**

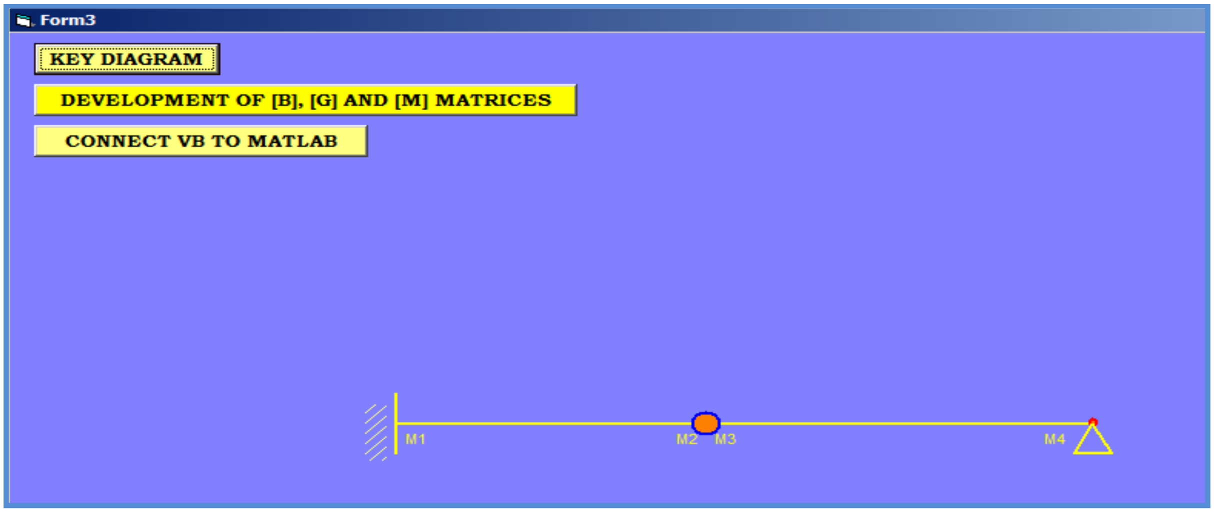


Fig.8.5 Key Diagram with Lumped Mass

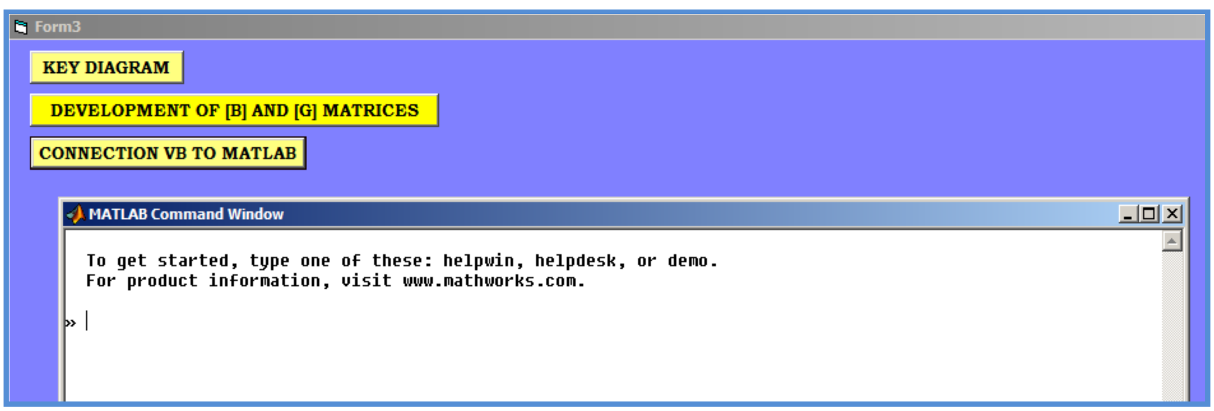


Fig.8.6 Connection of Visual Basic with Matlab

Step 5: Once Matlab command window is depicted the main processor starts and matrices are called using text file named as IFM_FRAME.txt, Gmatrix_Frame.txt and Mass.txt respectively as can be seen in **Fig. 8.7.**

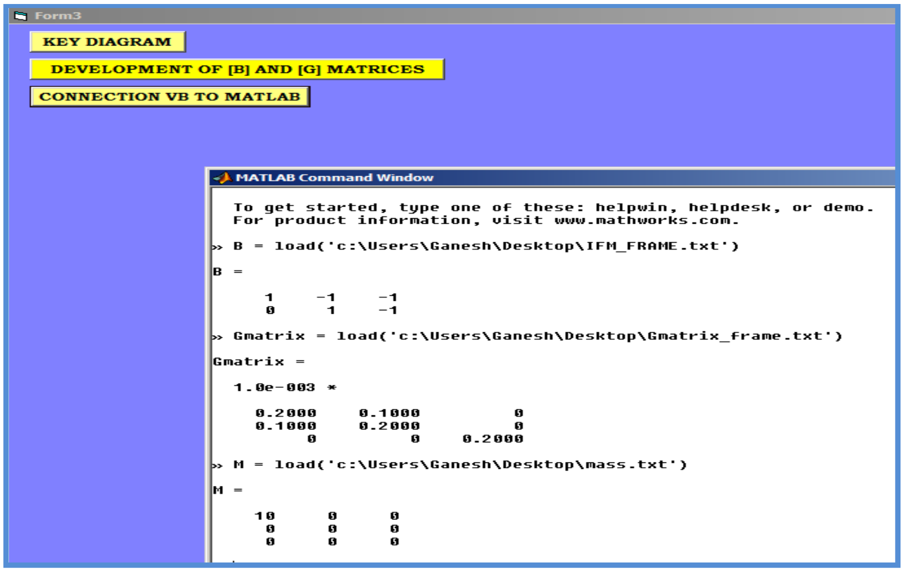


Fig.8.7 [Gmatrix] and [M] Matrix

Step 6: In the command window by changing path to folder named as “CCPROG” which is available at desktop, connection is established between a .m file which is Matlab based program for developing compatibility condition using LIUT technique. Thus by typing z = mtechexamplemod (B) in command window, one can enter number of compatibility conditions required. The dependent unknown codedepB and independent unknown codeindepB is auto selected for further calculation. The complete problem having three moment unknowns ‘n’ and having two free displacements at intermediate joint ‘m’ requires one compatibility condition which is named as z.cMatrix in matlab code. The screen shot of development of compatibility conditions is depicted in **Fig. 8.8.**

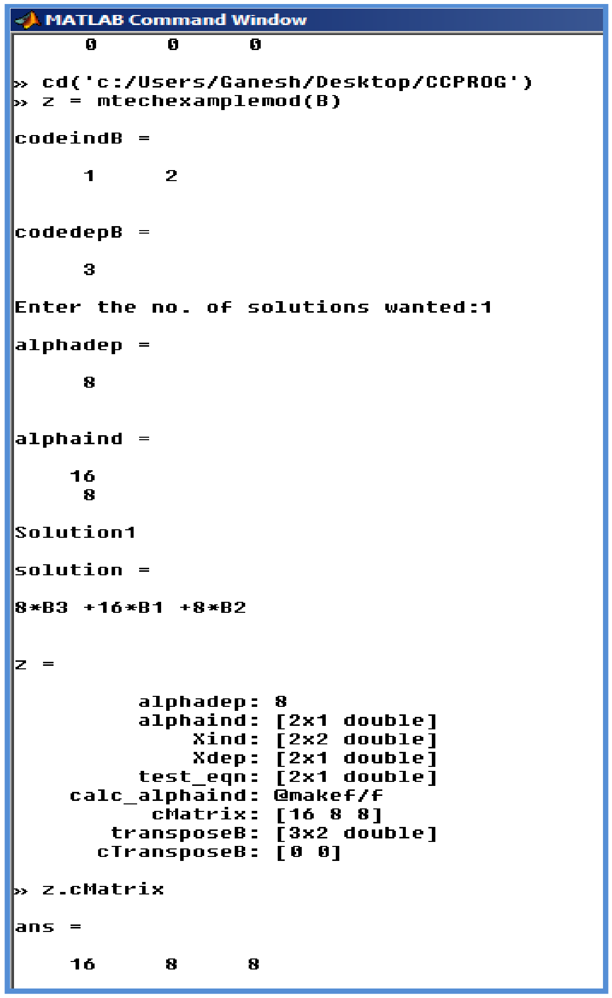
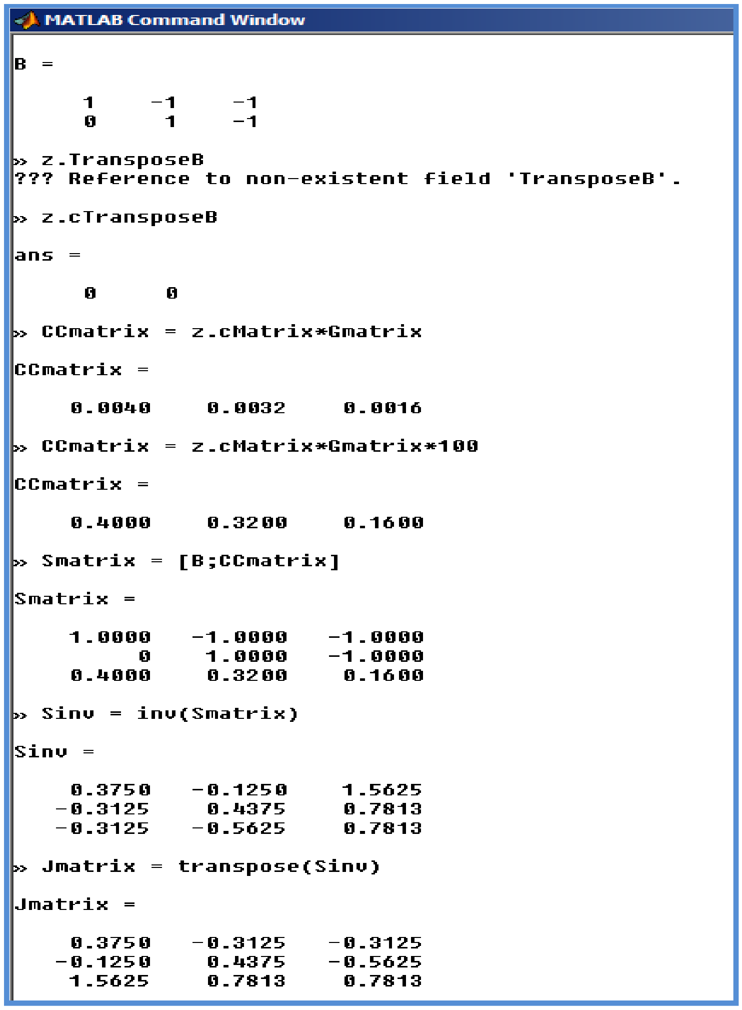


Fig. 8.8 Development of Compatibility Conditions

Step 7: Once 'z.cMatrix' is available it is checked for its null property by multiplying z.cMatrix with [B] matrix. Thus by typing 'z.cTransposeB' the null property check can be satisfied as depicted in the **Fig. 8.9.** Secondly, normalized global compatibility condition is also calculated by multiplying z.cMatrix with Gmatrix. The global equilibrium matrix [Smatrix] is developed by concatenation of [B] with the 'CCmatrix'. The inverse can be calculated by typing inv(Smatrix) in command window, which is named as 'Sinv'. The Jmatrix which is m rows of $[Smatrix^{-1}]^T$ is also worked out using the same. The complete procedure related to each operation is depicted in **Fig. 8.9.**



```
B =

     1    -1    -1
     0     1    -1

>> z.TransposeB
??? Reference to non-existent field 'TransposeB'.

>> z.cTransposeB

ans =

     0     0

>> CCmatrix = z.cMatrix*Gmatrix

CCmatrix =

    0.0040    0.0032    0.0016

>> CCmatrix = z.cMatrix*Gmatrix*100

CCmatrix =

    0.4000    0.3200    0.1600

>> Smatrix = [B;CCmatrix]

Smatrix =

    1.0000   -1.0000   -1.0000
         0    1.0000   -1.0000
    0.4000    0.3200    0.1600

>> Sinv = inv(Smatrix)

Sinv =

    0.3750   -0.1250    1.5625
   -0.3125    0.4375    0.7813
   -0.3125   -0.5625    0.7813

>> Jmatrix = transpose(Sinv)

Jmatrix =

    0.3750   -0.3125   -0.3125
   -0.1250    0.4375   -0.5625
    1.5625    0.7813    0.7813
```

Fig. 8.9 Different Matrix Operations

Step 8: Next, eigen value analysis is carried out using the inbuilt module. Product of [Mmatrix], [Jmatrix] and [Gmatrix], named as [MJGmatrix], is calculated directly. By typing directly on command window [M_modes, Omega] = eig(Smatrix, MJG) the necessary operation is carried out. Here Omega is a diagonal matrix of size (n x n), having possible values of frequencies along diagonal terms. Next, by substituting each value of 'ω' in matrix, the possible modes are calculated which is known as modal moment vector. It is named as M_modes corresponding to each value of frequency and calculated directly using Matlab software. **Figure 8.10** depicts the natural frequency value (ω = 47.8091 rad/sec) calculated in last line which is found to match with the exact solution 47.8091 rad/sec based on standard dynamics approach. The first column of M_mode matrix represents internal moments corresponding to value of $M_1 = 1$, for which M_2 and M_3 are worked out.

```
MATLAB Command Window

Jmatrix =

    0.3750   -0.3125   -0.3125
   -0.1250    0.4375   -0.5625
    1.5625    0.7813    0.7813

>> MJG = M*Jmatrix*Gmatrix

MJG =

  1.0e-003 *

    0.4375   -0.2500   -0.6250
         0         0         0
         0         0         0

>> Smatrix

Smatrix =

    1.0000   -1.0000   -1.0000
         0    1.0000   -1.0000
    0.4000    0.3200    0.1600

>> [M_modes,Omega] = eig(Smatrix,MJG)

M_modes =

    1.0000    1.0000    0.2535
   -0.8333    0.5000   -1.0000
   -0.8333    0.5000    0.5775


Omega =

  1.0e+003 *

    2.2857         0         0
         0       Inf         0
         0         0       Inf

>> w(1,1) = sqrt(Omega(1,1))

w =

   47.8091
```

Fig. 8.10 First Frequency and Moment Modal Values

Step 9: The first modal deflection values corresponding to first modal moments are calculated using standard IFM based formula i.e. $\{\delta\}$ = [Jmatrix]*[Gmatrix]*[M_modes]. Next, after normalizing the deflection value to unity the rotational value corresponding to that is calculated as depicted in **Fig. 8.11.**

```
MATLAB Command Window
>> First_modal  = Jmatrix*Gmatrix*[1.00 ; -0.8333; -0.833]

First_modal =

  1.0e-003 *

    0.1166
    0.0500
    0.0001

>> Normalized_First_modal = First_modal/First_modal(1,1)

Normalized_First_modal =

    1.0000
    0.4284
    0.0005
```

Fig. 8.11 First Modal Displacement Vector

8.2 EXAMPLES OF DYNAMICS ANALYSIS OF BEAMS

8.2.1 Simply Supported Beam Example

A mild steel beam which is simply supported at extreme ends has two segments of 1m each on either side of lumped mass Mo equal to 10 kN-Sec^2/m at centre as shown in **Fig. 8.12**. with EI = 1666.67 kN-m^2.

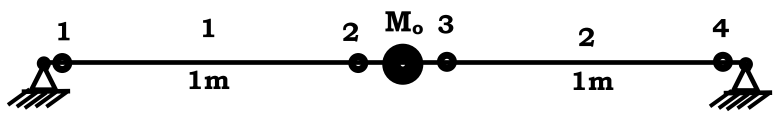


Fig. 8.12 Simply Supported Beam Example

Step 0 – Solution strategy: The beam has two segments on either side of the lumped mass (M_o). It has four internal moments (M_1, M_2, M_3 and M_4) out of which two extreme moments are zeros. The given problem has internal unknown moments as two (**Fig. 8.13**) and nodal displacements at intermediate joint as two. Thus the problem becomes statically determinate.

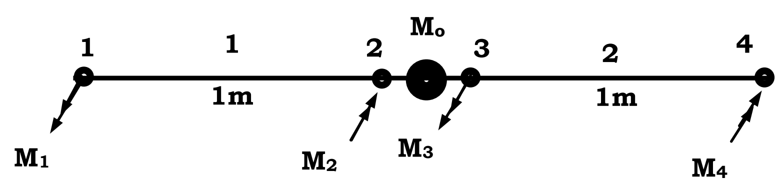


Fig. 8.13 Free Body Diagram

Steps 1 – Formulate the equilibrium equations: The EEs are written in terms of unknown moments at the intermediate joint i.e. at lumped mass along vertical displacement δ_{23} and rotational displacement Θ_{23} follows.

$$\begin{bmatrix} -1 & -1 \\ 1 & -1 \end{bmatrix} \begin{Bmatrix} M_2 \\ M_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

or [B]{M} = {P} … (8.1)

Step 2 – Derive the deformation displacement relations: The DDR is obtained in the form $\{\beta\} = [B]^T\{\delta\}$ and is written as

$$\begin{Bmatrix} \beta_2 \\ \beta_3 \end{Bmatrix} = \begin{bmatrix} -1 & 1 \\ -1 & -1 \end{bmatrix} \begin{Bmatrix} \delta_{23} \\ \theta\theta_{23} \end{Bmatrix}$$ …

(8.2)

Step 3 – Formulate force deformation relations: Force deformation relation for a beam problem is already derived in the previous illustrative example. The FDR in this case will be in terms of [G] matrix which is of size n x n. For end moments $\boldsymbol{M_i}$ and $\boldsymbol{M_j}$, the FDR can be written as

$$\beta_i = \frac{\ell}{6EI}\left(2M_i + M_j\right) \text{ and } \beta_j = \frac{\ell}{6EI}\left(M_i + 2M_j\right)$$ … (8.3)

Formulating above equations for two elements, one has

$$\beta_2 = \frac{1}{6EI}(2M_2) \quad \beta_3 = \frac{1}{6EI}(2M_3\ )$$ … (8.4)

In matrix form, the same can be expressed as

$$\begin{Bmatrix} \beta_2 \\ \beta_3 \end{Bmatrix} = \frac{2}{6EI}\begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}\begin{Bmatrix} M_2 \\ M_3 \end{Bmatrix} = [G]\{M\}$$ … (8.5)

Step 4 – Derive the lumped mass matrix: The Lumped mass matrix is directly written as given below.

$$[M_{mass}] = \begin{bmatrix} 10 & 0 \\ 0 & 0 \end{bmatrix}$$ … (8.6)

Step 5 – Calculate natural frequency: IFM based frequency equation is as follows:

$$[S]\{F\} = \lambda[J][G][M]\{F\}$$ … (8.7)

In case of beam bending problem the {F} vector represents internal moments {M}, while the same for pin jointed structures represents internal forces {F}. [S] is the global equilibrium matrix which is equal to [B] in this problem, [G] is the flexibility matrix, [M] is the mass matrix and $\lambda = \omega^2$ is known as eigen operator for frequency analysis.

Substituting all the above matrices in Eq. (8.7) and solving directly using Matlab eigen operator, the value of ω is found as 31.6227radians/sec, which is found to be matching with the standard dynamics formula of $\sqrt{\frac{48EI}{ML^3}}$ = 31.6228 radians/sec.

Step 6 – Calculate modal moments: Substituting the value of frequency obtained in Eq. (8.7) and calculating the modal moments, one gets

$$\begin{Bmatrix} M_2 \\ M_3 \end{Bmatrix} = \begin{Bmatrix} 1 \\ 1 \end{Bmatrix} \quad \text{... (8.8)}$$

Step 7 – Calculating nodal displacements: The nodal displacements can be calculated by substituting the values of moments into formula given below.

$$\begin{Bmatrix} \delta_{23} \\ \theta_{23} \end{Bmatrix} = [J][Gmatrix][M] = \begin{Bmatrix} -0.2 \times 10^{-3} \\ 0.000 \end{Bmatrix} \quad \text{... (8.9)}$$

It shows that the deflection value is maximum where slope equals to zero.

8.2.2 Fixed Beam Example:

A fixed mild steel beam as per **Fig. 8.14** has two segments of 1m each on either side of the lumped mass Mo equal to 10 kN-Sec²/m at centre. The beam has EI = 1666.67 kN-m².

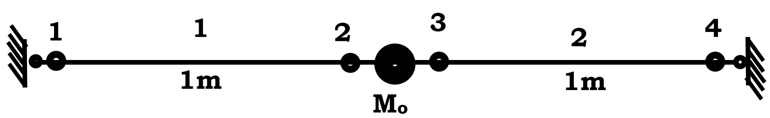


Fig. 8.14 Fixed Beam Example

Step 0 – Solution strategy: The given problem has four internal unknown moments (M_1, M_2, M_3 and M_4) and two nodal displacements at intermediate joint. Thus the problem is statically indeterminate (**Fig. 8.15).**

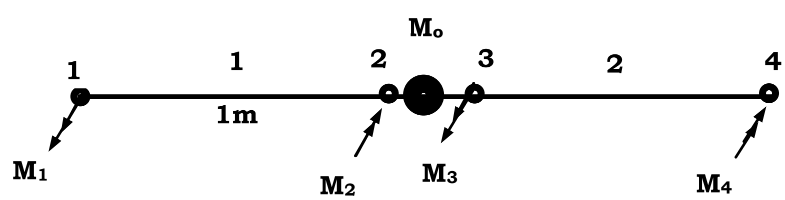


Fig. 8.15 Free Body Diagram

Step 1 – Formulate the equilibrium equations: The EEs are written in terms of unknown moments at the intermediate joint i.e. at lumped mass along vertical displacement δ_{23} and rotational displacement Θ_{23} as

$$\begin{bmatrix} 1 & -1 & -1 & 1 \\ 0 & 1 & -1 & 0 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

$$[B]\{M\} = \{P\} \qquad \ldots (8.10)$$

Step 2 – Derive the deformation displacement relations: The DDR is obtained as $\{\beta\} = [B]^T\{\delta\}$.

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = \begin{bmatrix} -1 & 0 \\ -1 & 1 \\ -1 & -1 \\ 1 & 1 \end{bmatrix} \begin{Bmatrix} \delta_{23} \\ \theta_{\theta_{23}} \end{Bmatrix} \qquad \ldots$$

(8.11)

Step 3 – Generate the compatibility conditions: The two compatibility conditions can be expressed using Matlab code as

$$\begin{bmatrix} 16 & 9 & 9 & 2 \\ 12 & 9 & 9 & 6 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix} \qquad \ldots (8.12)$$

Or $[C]\{\beta\} = \{0\}$

Step 3 – Formulate force deformation relations: For end moments $\boldsymbol{M_i}$ and $\boldsymbol{M_j}$, the FDR can be written as

$$\beta_i = \frac{\ell}{6EI}\left(2M_i + M_j\right) \quad \text{and} \quad \beta_j = \frac{\ell}{6EI}\left(M_i + 2M_j\right) \qquad \ldots (8.13)$$

Writing above equations for the two members leads to

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \end{Bmatrix} = \frac{2}{6EI} \begin{bmatrix} 2 & 1 & 0 & 0 \\ 1 & 2 & 0 & 0 \\ 0 & 0 & 2 & 1 \\ 0 & 0 & 1 & 2 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = [G]\{M\} \qquad \ldots (8.14)$$

Step 4 – Derive the lumped mass matrix: The Lumped mass matrix can be directly written as given below.

$$[M_{mass}] = \begin{bmatrix} 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \qquad \ldots (8.15)$$

Steps 5 – Calculate natural frequency: Calculate global normalized compatibility conditions by multiplying [C] matrix and Global Flexibility matrix [Gmatrix]. By concatenating into basic equilibrium matrix [B], a global equilibrium matrix [S] is developed. The basic IFM based equation now can be written by substituting all the related matrices into Eq. (8.7) as

$$\begin{bmatrix} 1 & -1 & =1 & 1 \\ 0 & 1 & -1 & 0 \\ 0.41 & 0.34 & 0.2 & 0.13 \\ 0.33 & 0.3 & 0.24 & 0.21 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} =$$

$$\frac{\omega^2}{1666.67} \begin{bmatrix} 10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} 0.25 & -0.25 & -0.25 & 0.25 \\ -0.25 & 0.5 & -0.5 & 0.25 \\ 5.55 & -0.6944 & 0.6944 & -6.994 \\ -4.6296 & 1.6204 & 1.6204 & 7.8704 \end{bmatrix} \begin{bmatrix} 2 & 1 & 0 & 0 \\ 1 & 2 & 0 & 0 \\ 0 & 0 & 2 & 1 \\ 0 & 0 & 1 & 2 \end{bmatrix} \qquad \ldots (8.16)$$

Solving above equations one can get the solution as $\omega = 63.2455\ \mathrm{rad/Sec}$, which is found to be matching with the exact solution of 63.2456 rad/seconds.

Step 6 – Calculate modal moments: Substituting the values of frequency calculated in Eqn.(8.16) the internal nodal moments can be calculated which is as

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \end{Bmatrix} = \begin{Bmatrix} 1.0 \\ -1.0 \\ -1.0 \\ 1.0 \end{Bmatrix} \qquad \ldots (8.17)$$

Steps 7 – Calculate Modal Moments (Internal Moments)

Substituting the values of nodal moments into Eq. (8.8) the nodal moments are calculated which is as below

$$\begin{Bmatrix} \delta_{23} \\ \theta_{23} \end{Bmatrix} = \begin{Bmatrix} 1.0 \times 10^{-4} \\ 0.000 \end{Bmatrix} \quad \ldots (8.18)$$

8.2.3 Continuous Beam Example

A two span continuous beam having two extreme ends as fixed is shown in **Fig. 8.16**. It is to be analysed considering four segments of 1m each on both sides of two lumped mass 'Mo'. Consider EI as 666.67 kN-m^2.

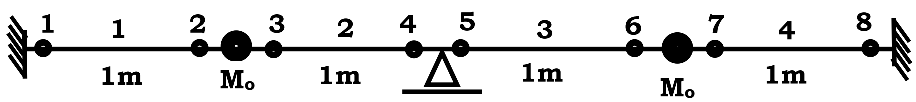


Fig. 8.16 Two Span Continuous Beam Example

Following three possibilities are considered here: (i) Direct Lumping of Constant Mass at given nodes (ii) Lumped Mass calculated based on basis of mass per unit length and number of members meeting at a joint [LMcase] and (iii) Consistent Mass which is again based on mass per unit length and number of members meeting at a joint [CMcase]. Solutions obtained based on different mass criteria are verified here by Stiffness based eigen value analysis.

(i) DNLM Case

Step 0 – Solution strategy: The continuous beam is analysed by considering four segments with lumping mass of M_o equals 10kN at the centre of each span. Each beam segment has internal moments (M_1, M_2). Thus, the complete problem has total eight internal unknown moments (n) and total five possible free joint displacements (m) as shown in **Fig. 8.17**. Thus the problem has 3 degrees of static indeterminacy.

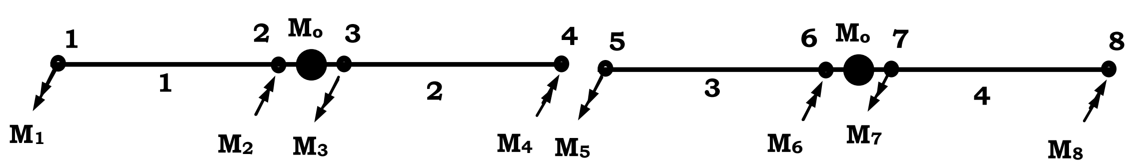


Fig. 8.17 Free Body Diagram

Step 1 – Formulate the equilibrium equations: The EEs are written in terms of unknown moments at the intermediate joints i.e. at lumped masses and at intermediate junctions corresponding to vertical displacements δ_{23}, δ_{67} and rotational displacements Θ_{23}, Θ_{45}, and Θ_{67}. In matrix form

$$\begin{bmatrix} 1 & -1 & -1 & 1 & 0 & 0 & 0 & 0 \\ 0 & 1 & -1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & -1 & -1 & 1 \\ 0 & 0 & 0 & 0 & 0 & 1 & -1 & 0 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_6 \\ M_7 \\ M_8 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

Which can be seen as [B]{M} = {P} or [S]{M} = {P].

Step 2 – Derive the deformation displacement relations: The DDR is obtained as $\{\beta\} = [B]^T\{\delta\}$.

$$\begin{Bmatrix} \beta_1 \\ \cdot \\ \cdot \\ \beta_8 \end{Bmatrix} = [B^T] \begin{Bmatrix} \delta_{23} \\ \theta_{23} \\ \theta_{45} \\ \delta_{67} \\ \theta_{67} \end{Bmatrix}$$

Step 3 – Generate the compatibility conditions: The three compatibility conditions for the beam problem can be worked out by using .m file of Matlab which is named as mtechexamplemod.

$$\begin{bmatrix} -8 & 2 & 2 & 12 & 12 & 9 & 9 & 6 \\ 1 & 1 & 1 & 1 & 1 & 3 & 3 & 5 \\ 2 & 9 & 9 & 9 & 16 & 9 & 9 & 2 \end{bmatrix} \{\beta\} = \{0\}$$

Which can be written in the form $[C]\{\beta\} = \{0\}$

Step 4 – Formulate force deformation relations: For end moments $\boldsymbol{M_i}$ and $\boldsymbol{M_j}$, the FDR can be written as

$$\beta_i = \frac{\ell}{6EI}\left(2M_i + M_j\right)$$

$$\beta_j = \frac{\ell}{6EI}\left(M_i + 2M_j\right)$$

Formulating above equations for all the three members and arranging in a matrix form one can write,

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \\ \beta_4 \\ \beta_5 \\ \beta_6 \\ \beta_7 \\ \beta_8 \end{Bmatrix} = \frac{1}{6EI}\begin{bmatrix} 2 & 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & 2 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 2 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 2 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 2 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & 2 \end{bmatrix}\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_6 \\ M_7 \\ M_8 \end{Bmatrix} = [G]\{M\}$$

Step 5 – Derive the mass matrix for DNLM Case: The lumped mass matrix can be is directly written as

$$[M_{DNLM}] = \begin{bmatrix} 10 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 10 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

Step 6 – Calculate natural frequency: The global normalized compatibility conditions are obtained by multiplying [C] matrix by Global Flexibility matrix [Gmatrix] and then concatenating into basic equilibrium matrix [B], a global equilibrium matrix [S] is developed.

The governing IFM based equations are developed by substituting all the required matrices in Eq. (8.7). After solving the equations using matlab based eigen module 'eig(a,b)'. The calculated frequencies based on DNLM case are as follows;

$$\begin{Bmatrix} \omega_1 \\ \omega_2 \end{Bmatrix} = \begin{Bmatrix} 47.8092 \\ 63.2456 \end{Bmatrix} rad/sec$$

In which first natural frequency is matching with value of propped cantilever case while second one is matching with the fixed case. As the mass is on both the side, a mechanism is developed at intermediate joint such that it works as a propped cantilever beam where moment equals to zero, with possible rotational value. On the other hand, whenever both the masses during motion are on one side of the beam the same joint having zero rotation with reacting moment values is considered as second mode of vibration.

Step 7 – Calculate modal moments: Substituting the above values of frequencies into basic equation of IFM, the internal nodal moments can be calculated. For the first natural frequency, the moments are

$$\begin{Bmatrix} M_1 \\ M_2 \\ M_3 \\ M_4 \\ M_5 \\ M_6 \\ M_7 \\ M_8 \end{Bmatrix} = \begin{Bmatrix} -1.00 \\ 0.833 \\ 0.833 \\ 0.00 \\ 0.00 \\ -0.833 \\ -0.833 \\ 1.00 \end{Bmatrix}$$

Step 8 – Calculate nodal displacements: Substituting the values of nodal moments calculated above into Eq. (8.9), the nodal displacements for first mode moments are found as

$$\begin{Bmatrix} \delta_{23} \\ \theta_{23} \\ \theta_{45} \\ \delta_{67} \\ \theta_{67} \end{Bmatrix} = \begin{Bmatrix} -0.1166 \\ -0.05 \\ 0.2 \\ 0.1166 \\ -0.05 \end{Bmatrix} x\ 10^{-3}$$

(i) Considering Lumping Mass Criteria (LM Case)

In this case, the contributory mass is considered at each node as per the number of members meeting at a junction corresponding to lateral deflection direction only. Considering mass per unit length of each member as 1kN/m, the lumped mass matrix for each member written as

1 2 3 4

$$[M_1] = [M_2] = [M_3] = [M_4] = \begin{bmatrix} 0.5 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0.5 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

After assembly, the global lumped mass matrix corresponding to vertical displacement based contributory criteria at joints (2-3) and (6-7) can be written as

$$[M_L] = \begin{array}{c} \begin{matrix} 1 & 2 & 3 & 4 & 5 \end{matrix} \\ \begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array}$$

(ii) Considering Consistent Mass (CM Case)

In this case, actual contributory mass is calculated at each node as per number of members meeting at junction, where contribution corresponding to rotation is also considered in addition to that corresponding to lateral displacement. Considering mass per unit length of each member as 1kN/m, Thus for all the beam members of 1m length the consistent mass is found as

$$[M_1] = [M_2] = [M_3] = [M_4] = 2.389 \times 10^{-3} \begin{array}{c} \begin{matrix} 1 & 2 & 3 & 4 \end{matrix} \\ \begin{bmatrix} 156 & & & \text{Symm} \\ 22 & 4 & & \\ 54 & 13 & 156 & \\ -13 & -3 & -22 & 4 \end{bmatrix} \end{array}$$

After assembly, the global consistent mass matrix is found as

$$[M_C] = \begin{array}{c} \begin{matrix} 1 & 2 & 3 & 4 & 5 \end{matrix} \\ \begin{bmatrix} 0.7428 & & & & \\ 0 & 0.1904 & & \text{Symm} & \\ -0.0309 & -0.00714 & 0.1904 & & \\ 0 & 0 & 0.0309 & 0.7424 & \\ 0 & 0 & -0.00714 & 0 & 0.1904 \end{bmatrix} \end{array}$$

By following the steps given above, the frequencies are worked out based on LM and CM cases. By substituting the frequencies, the internal forces and nodal displacements are calculated for the first frequency. The frequency values are checked using Standard stiffness based eigen value analysis. The comparison is presented in **Table 8.1**. The corresponding moments for the first natural frequency value are given in **Table 8.2**.

Table 8.1 Natural Frequencies for Continuous Beam

Natural Frequency (ω) Radians/Second			
Lumped Mass		Consistent Mass	
IFM	Stiffness Method	IFM	Stiffness Method **[93]**
151.188	151.187	158.881	158.879
200.001	200.00	232.11	232.10
		596.11	596.11
		836.66	836.66
		1558.33	1558.33

Table 8.2 Moments and Nodal Displacements

Based on Lumped Mass		Based on Consistent Mass	
Internal Moments	Nodal Displacements (δ) x 10^{-3}	Internal Moments	Nodal Displacements (δ) x 10^{-3}
M_1 = -1.0	δ_{23} = 1.00	M_1 = -1.0	$\delta_{23} = -0.1219$
M_2 = 0.83	Θ_{23} = 0.00	M_2 = 1.00	$\Theta_{23} = -0.0657$
M_3 = 0.83	Θ_{45} = 0.00	M_3 = -1.00	Θ_{45} = 0.2404
M_4 = 0.00	δ_{67} = 1.00	M_4 =-1.00	δ_{67} = 0.1219
M_5 = 0.00	Θ_{67} = 0.00	M_5 = 1.00	$\Theta_{67} = -0.0657$
M_6 = -0.83		M_6 = 1.00	
M_7 = -0.83		M_7 = -1.00	
M_8 = 1.00		M_8 = 1.0	

8.3 EXAMPLES OF DYNAMIC ANALYSIS OF PLANE TRUSS

Two examples of plane truss are solved here using IFM based formulation by considering (1) Direct Nodal Lumped Mass of 10kN at nodes under consideration (2), Lumped mass criteria as per members meeting at joints and (3) Consistent mass criteria. The solution obtained is compared with the stiffness based Eigen values analysis **[93].**

8.3.1 Three Member Truss Example

A three member truss shown in **Fig. 8.18** has three joints out of which two are loaded with direct nodal lumping of mass (Mo) equals to 10 kN-Sec^2/m. The members have axial rigidity as 2.01 x 10^6 kN. The truss is to be analysed under different mass criteria by considering unit weight as 1 kN/m.

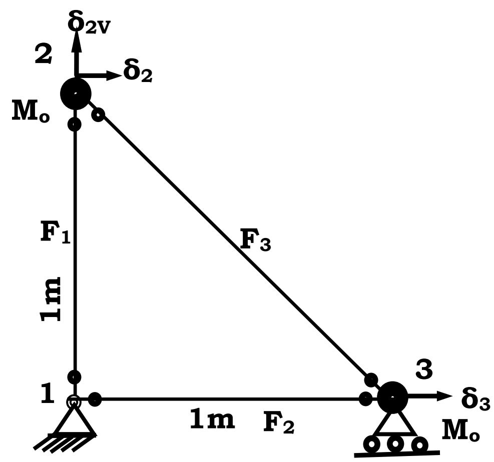


Fig. 8.18 Three Member Plane Truss Example

The complete methodology being same, the necessary changes in the derivation of mass matrices for different cases are discussed here.

(i) Direct Nodal Lumping of Mass (DNLM Case)

Three possible nodal displacements in the truss are δ_{3H}, δ_{2H} and δ_{2V} as shown in **Fig. 8.18**. Hence there are three possible frequencies corresponding to each nodal displacement with three internal force unknowns. Thus, the problem is having basic equilibrium matrix [B] which directly become a square global equilibrium matrix [S] as it does not require compatibility conditions. The nodal lumping mass matrix for M_o can be written as

$$[M_{DNLM}] = \begin{array}{c} \begin{matrix} 2_H & 2_V & 3_H \end{matrix} \\ \begin{bmatrix} 10 & 0 & 0 \\ 0 & 10 & 0 \\ 0 & 0 & 10 \end{bmatrix} \end{array}$$

After deriving all the necessary matrices and substituting in Eq. (8.16), the values of natural frequencies are calculated and are compared with the stiffness method based eigen values analysis in **Table 8.3.** After substituting each IFM based frequency value in same Eq. (8.16) relative internal force in each member is calculated by considering F_1 equals to 1, using Matlab software. By substituting the relative internal forces of each member the corresponding nodal displacements are calculated. The relative internal forces as well as nodal displacements obtained for first natural frequency are reported in **Table 8.4.**

Table 8.3 Natural Frequencies (DNLM Case)

Natural Frequency (ω) rad/sec	
IFM	Stiffness Based **[93]**
194.855	194.833
448.33	448.33
613.36	613.33

Table 8.4 Internal Forces and Nodal Displacements

Internal Forces (F)	Nodal Displacements (δ) x 10^{-6}
F_1 = -1.0	$\delta_{2H} = -0.8718$
F_2 = 0.232	$\delta_{3H} = -0.8718$
F_3 = 0.232	$\delta_{2V} = 1.00$

(ii) Considering Lumping Mass Criteria (LM Case)

In this case, actual contributory mass is calculated at each node as per the number of members meeting at the joint. The mass matrix for each member can be written as

$$[M_1] = [M_2] = [M_3] = \begin{array}{c} \begin{matrix} 1 & 2 & 3 & 4 \end{matrix} \\ \begin{bmatrix} 0.5 & 0 & 0 & 0 \\ 0 & 0.5 & 0 & 0 \\ 0 & 0 & 0.5 & 0 \\ 0 & 0 & 0 & 0.5 \end{bmatrix} \end{array}$$

The transformation from local to global direction is carried out by

$$[M_L] = [T]^T[M_1][T]$$

Where [T] is the transformation matrix for a plane truss member and is given by

$$[T] = \begin{bmatrix} C_x & C_y & 0 & 0 \\ -C_y & C_x & 0 & 0 \\ 0 & 0 & C_x & C_y \\ 0 & 0 & -C_y & C_x \end{bmatrix}$$

Next, the global lumped mass matrix is calculated, which is as follows

$$\begin{matrix} & 2_H & 2_V & 3_H \end{matrix}$$

$$[M_L] = \begin{bmatrix} 1.2069 & & \text{Symm} \\ 0 & 1.2069 & \\ 0 & 0 & 1.2069 \end{bmatrix}$$

After deriving all the necessary matrices and substituting in Eq. (8.16), the values of natural frequencies are calculated and are compared with those obtained using stiffness based eigen value analysis (**Table 8.5).** Relative internal force in each member is also calculated by considering F_1 equals to 1. Finally, the corresponding nodal displacements are calculated. Results obtained for the the relative internal forces as well as nodal displacements for first natural frequency are presented in **Table 8.6.**

Table 8.5 Natural Frequencies for Plane Truss

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method **[93]**
560.85	560.83
1290.5	1290.4
1765.4	1765.3

Table 8.6 Internal Forces and Nodal Displacements

Internal Force (F)	Nodal Displacements (δ) x 10^{-5}
0.8111	0.1879
0.8111	0.0434
1.000	0.0434

(iii) Considering Consistent Mass Criteria (CM Case)

In this case, actual contributory mass is calculated at each node as per the number of members meeting at junction. Thus for all the members of the plane truss the lumped mass matrix is as under.

$$[M_1] = \frac{1\,x\,1}{6}\begin{bmatrix} 2 & 0 & 1 & 0 \\ 0 & 2 & 0 & 1 \\ 1 & 0 & 2 & 0 \\ 0 & 1 & 0 & 2 \end{bmatrix} = [M_2] \quad \text{and,} \quad [M_3] = \frac{1\,x\,\sqrt{2}}{6}\begin{bmatrix} 2 & 0 & 1 & 0 \\ 0 & 2 & 0 & 1 \\ 1 & 0 & 2 & 0 \\ 0 & 1 & 0 & 2 \end{bmatrix}$$

(columns of $[M_1]$ labelled 1, 2, 3, 4)

The global consistent mass matrix is as follows

$$[M_L] = \begin{bmatrix} 0.8064 & & \text{Symm} \\ 0.00 & 0.8046 & \\ 0.2356 & 0.00 & 0.8046 \end{bmatrix}$$

(columns labelled 2_H, 2_V, 3_H)

After deriving all the necessary matrices and substituting in Eq. (8.16), values of natural frequencies are calculated and are verified by stiffness based eigen analysis as depicted in **Table 8.7** whereas results obtained for the relative internal forces as well as nodal displacements for the first natural frequency value are depicted in **Table 8.8.**

Table 8.7 Natural Frequencies for Plane Truss

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method
204.03	204.0
507.71	507.69
749.47	749.413

Table 8.8 Internal Forces and Nodal Displacements

Internal Force (F)	Nodal Displacements (δ) x 10^{-5}
-0.7868	0.1812
-1.00	0.0391
0.9273	0.0498

8.3.2 Eleven Member Truss Example

A 11- member truss having four joints loaded with direct nodal lumping of mass (Mo) equals to 10 kN-Sec^2/m. The axial rigidity is constant for all the members and is equal to 2.01 x 10^6 kN. The frequency analysis is to be carried out for lumped and consistent mass criteria by considering weight as 1 kN/m.

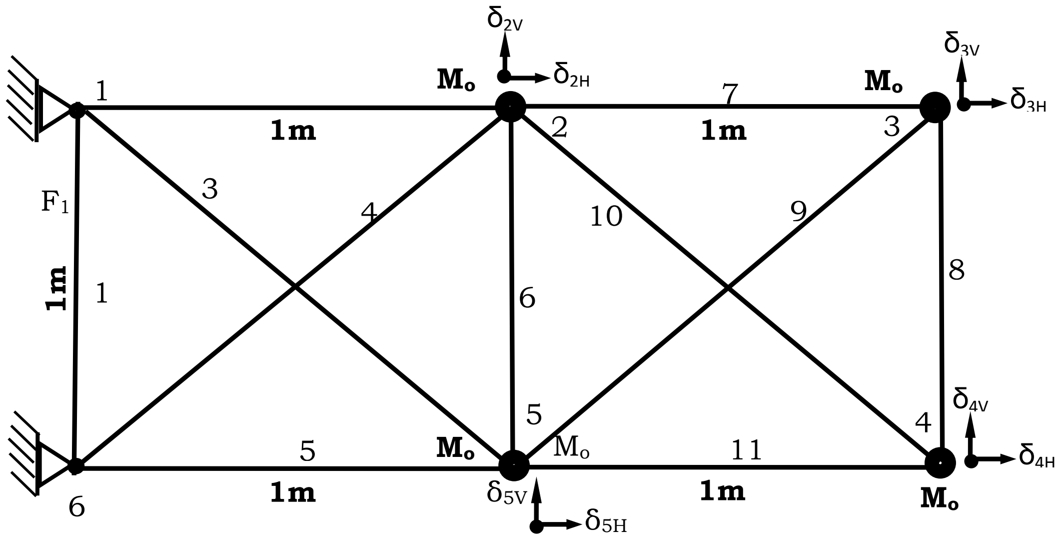


Fig. 8.19 Eleven Member Plane Truss Example

(i) Considering Direct Nodal Lumping Mass (DNLM Case)

There are total eight possible nodal displacements in the truss. Hence there are eight possible frequencies corresponding to each displacement while there are eleven internal force unknowns. The basic equilibrium matrix [B] is of size 8 x 11. Thus, the problem is three degree statically indeterminate as per IFM procedure.

The mass matrix M_o can be written by referring Eq. (8.6) as

$$
\begin{array}{c} \\ [M_{DNLM}] = \end{array}
\begin{array}{c}
\begin{array}{cccccccc} 2_H & 2_V & 3_H & 3_H & 4_H & 4_V & 5_H & 5_V \end{array} \\
\begin{bmatrix}
10 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & 10 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & 10 & 0 & 0 & 0 & 0 & 0 \\
 & & & 10 & 0 & 0 & 0 & 0 \\
 & & & & 10 & 0 & 0 & 0 \\
 & \text{Sym} & & & & 10 & 0 & 0 \\
 & & & & & & 10 & 0 \\
 & & & & & & & 10
\end{bmatrix}
\end{array}
$$

After deriving all the necessary matrices and substituting in Eq. (8.16), values of natural frequencies are calculated and are verified by stiffness based eigen value analysis (**Table 8.9).** After substituting each IFM based frequency value in same Eq. (8.16), the relative internal forces in each member are calculated by considering F_2 equals to 1. In this case as member one is constrained at support in both the direction, there is no internal force developed in it. By substituting the relative internal force of each member into Eq. (8.8) and using Matlab, corresponding nodal displacements are calculated. The relative internal forces as well as nodal displacements for first natural frequency are depicted in **Table 8.10.**

Table 8.9 Natural Frequencies

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method
104.665	101.04
312.9164	308.33
350.038	325.44
611.7482	589.33
643.8708	643.08
770.7261	748.19
789.6878	774.65
910.3956	874.34

Table 8.10 Internal Forces

Internal Forces (F)	Nodal Displacements (δ) x 10^{-6}
F_1 = 0.00	$\delta_{2H} = -0.8718$
F_2 = 1.00	$\delta_{2V} = -0.8718$
F_3 = 0.5049	$\delta_{3H} = 1.00$
F_4 = -0.5049	$\delta_{3V} = 1.00$
F_5 = -1.00	$\delta_{4H} = -0.8718$
F_6 = 0.00	$\delta_{4V} = -0.8718$
F_7 = 0.3305	$\delta_{5H} = 1.00$
F_8 = 0.00	$\delta_{5V} = 1.00$
F_9 = -0.3650	

F_{10} = 0.3650	
F_{11} = -0.3305	

(i) Considering Lumping Mass Criteria (LM Case)

The global is found as

$$
[M_L] = \begin{array}{c} \\ \end{array}
\begin{array}{cccccccc}
2_H & 2_V & 3_H & 3_H & 4_H & 4_V & 5_H & 5_V
\end{array}
$$

$$
[M_L] = \begin{bmatrix}
2.9142 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & 2.9142 & 0 & 0 & 0 & 0 & 0 & 0 \\
 & & 1.7071 & 0 & 0 & 0 & 0 & 0 \\
 & & & 1.7071 & 0 & 0 & 0 & 0 \\
 & & & & 1.7071 & 0 & 0 & 0 \\
 & \text{Symm} & & & & 1.7071 & 0 & 0 \\
 & & & & & & 2.9142 & 0 \\
 & & & & & & & 2.9142
\end{bmatrix}
$$

Following the procedure used in previous example, results obtained for natural frequencies, internal forces and nodal displacements are reported here in **Table 8.11** and **Table 8.12.**

Table 8.11 Natural Frequencies for Plane Truss

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method
240.44	231.68
688.911	650.068
703.41	678.89
1357.11	1238.9
1368.44	1339.4
1550.11	1553.83
1550.11	1553.83
1944.44	1864.55

Table 8.12 Internal Forces and Nodal Displacements

Internal Force (F)	Nodal Displacements (δ) x 10^{-5}
F_1 = 0.00	$\delta_{2H} = -0.04977$
F_2 = 1.00	$\delta_{2V} = 0.0944$
F_3 = 0.5513	$\delta_{3H} = -0.0644$
F_4 = -0.5513	$\delta_{3V} = 0.2344$
F_5 = -1.000	$\delta_{4H} = 0.0644$
F_6 = 0.00	$\delta_{4V} = 0.2345$
F_7 = 0.2950	$\delta_{5H} = 0.0497$

F_8 = 0.000	δ_{5V} = 0.0944
F_9 = -0.3273	
F_{10} = -0.3273	
F_{11} = -0.2950	

(ii) Considering Consistent Mass Criteria (CM Case)

Based on consistent mass criteria, the global consistent mass matrix is found as follows

$$[M_C] = \begin{array}{c} \begin{matrix} 2_H & 2_V & 3_H & 3_V & 4_H & 4_V & 5_H & 5_V \end{matrix} \\ \begin{bmatrix} 1.9423 & 0 & 0.1667 & 0 & 0.2356 & 0 & 0.1667 & 0 \\ & 1.9423 & 0 & 0.1667 & 0 & 0.2356 & 0 & 0.1667 \\ & & 0.333 & 0 & 0.1667 & 0 & 0 & 0 \\ & & & 0.333 & 0 & 0.1667 & 0 & 0 \\ & & & & 1.1378 & 0 & 0.1667 & 0 \\ & \text{Symm} & & & & 1.1378 & 0 & 0.1667 \\ & & & & & & 1.9423 & 0 \\ & & & & & & & 1.9423 \end{bmatrix} \end{array}$$

Natural frequency results are presented in **Table 8.13** along with the comparison with solution obtained based on stiffness method of analysis whereas the results obtained for internal forces and nodal displacements are reported here in **Table 8.14.**

Table 8.13 Natural Frequencies for Plane Truss

Natural Frequency (ω) rad/sec	
IFM	Eigen Analysis (Stiffness Based)
257.994	247.4
720.41	709.01
916.041	850.7
1723..11	1648.44
1368.44	1690.01
1799.11	1846.88
2073.50	2057.01
2602.22	2494.88

Table 8.14 Internal Forces and Nodal Displacements

Internal Forces(F)		Nodal Displacements (δ) x 10^{-5}
F_1 = 0.00	F_8 = 0.000	δ_{2H} = 0.04988
F_2 = 1.00	F_9 = 0.3310	δ_{2V} = −0.0994
F_3 = 0.6188	F_{10} = -0.3310	δ_{3H} = −0.0634

$F_4 =$ -0.6188	$F_{11} = 0.2726$	$\delta_{3V} = -0.2391$
$F_5 =$ -1.000		$\delta_{4H} = -0.0633$
$F_6 =$ 0.00		$\delta_{4V} = -0.2391$
$F_7 =$ -0.2726		$\delta_{5H} = -0.0499$, $\delta_{5v =.} - 0.098$

8.4 EXAMPLE OF DYNAMIC ANALYSIS OF PLANE FRAME

A fixed footed portal frame is considered here with direct nodal lumping of mass (M_o) equals to 10 kN-Sec^2/m at the centre of beam as depicted in **Fig. 8.20.** The members have EI equals to 1666.67 kN – m^2. The frequency analysis is carried out for lumped and consistent mass criteria by considering unit weight of each member as 1kN/m.

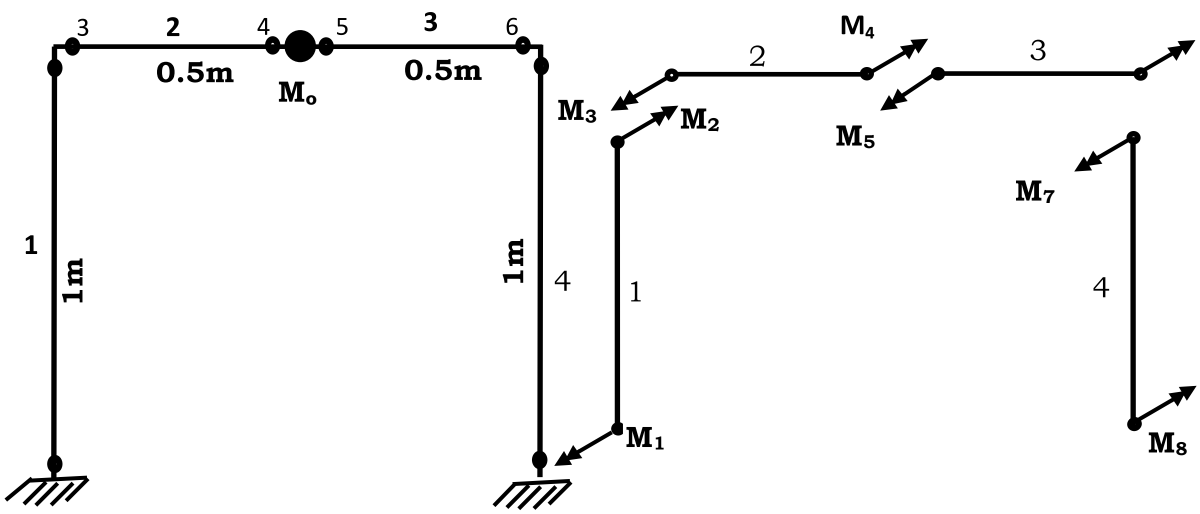


Fig. 8.20 Plane Frame Example with Free Body Diagram

1. Considering Direct Nodal Lumping Mass (DNLM Case)

The problem is having following five possible nodal displacements: (1) Horizontal displacement (Lateral sway) of beam member δ_{23H}. (2) Rotational displacement at junction of members 1-2 θ_{12},(3) Rotational displacement at junction of member 2-3 θ_{23} ,(4) Rotational displacement at joint 3-4 θ_{34} and (5) Vertical displacement at lumped mass δ_{23V}. Hence there are five possible frequencies one corresponding to each displacement. Eight internal moments are as shown in **Fig. 8.20.** The problem has basic equilibrium matrix [B] of size 5 x 8 and hence the problem is three degree statically indeterminate as per IFM.

The nodal lumping mass matrix can be written by referring Eq. (8.27) as

$$
\begin{matrix} & \delta_{23H} & \theta_{12} & \delta_{23V} & \theta_{23} & \theta_{34} \end{matrix}
$$

$$
[M_{DNLM}] = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ & 0 & 0 & 0 & 0 \\ & & 10 & 0 & 0 \\ & \text{Sym} & & 0 & 0 \\ & & & & 0 \end{bmatrix}
$$

Following the procedure used in the previous example, IFM and stiffness method based frequencies are worked out and are given in **Table 8.15**. After substituting IFM based first natural frequency value in basic equation of IFM formulation all the internal moments are worked out which are depicted in **Table 8.16**.

Table 8.15 Natural Frequencies (DNLM Case)

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method **[93]**
52.9151	52.9150
126.4912	126.4911

Table 8.16 Internal Moments and Nodal Displacements

Internal Moments (F)	Nodal Displacements (δ) x 10^{-3}
M_1 = 1.00	δ_{23H} = 0.125
M_2 =-0.75	$\Theta_{12} = -0.075$
M_3 = -0.75	δ_{23V} = 0.0375
M_4 = 0.00	Θ_{12} = 0.00
M_5 = 0.00	$\Theta_{34} = -0.075$
M_6 = 0.75	
M_7 = 0.75	
M_8 = -1.00	

(ii) Considering Lumping Mass Criteria (LM Case)

Considering mass per unit length of each member as 1kN/m and after suitable transformation the global lumped mass matrix is obtained as

$$[M_L] = \begin{array}{c} \begin{matrix} \delta_{23H} & \Theta_{12} & \delta_{23V} & \Theta_{23} & \Theta_{34} \end{matrix} \\ \begin{bmatrix} 1.0 & 0 & 0 & 0 & 0 \\ & 0 & 0 & 0 & 0 \\ & & 0.5 & 0 & 0 \\ & & & 0 & 0 \\ & & & & 0 \end{bmatrix} \end{array}$$

Results obtained for natural frequencies, internal force and nodal displacements are reported in Table 8.17 and 8.18.

Table 8.17 Natural Frequencies (LM Case)

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method
159.2038	159.2038
221.5824	221.5824

Table 8.18 Internal Forces and Nodal Displacements

Internal Moments (M)	Nodal Displacements (δ) x 10^{-3}
M_1 = 1.00	δ_{23H} = 0.1323
M_2 = -0.6788	$\Theta_{12} = -0.0969$
M_3 = -0.6788	δ_{23V} = 0.0924
M_4 = -0.5833	Θ_{12} = 0.00
M_5 = -0.5833	$\Theta_{34} = -0.0969$
M_6 = 0.6768	
M_7 = 0.6768	
M_8 = -1.00	

(iii) Considering Consistent Mass Criteria (CM Case)

The global consistent mass matrix by following the procedure outlined earlier is found as

δ_{23H} Θ_{12} δ_{23V} Θ_{23} Θ_{34}

$$[M_C] = \begin{bmatrix} 0.7428 & -0.0523 & 0 & 0 & -0.0261 \\ & 0.0119 & 0.01547 & -0.00178 & 0 \\ & & 0.7428 & 0 & -0.01547 \\ & \text{Symm} & & 0.00476 & -0.00178 \\ & & & & 0.00476 \end{bmatrix}$$

Results obtained considering consistent mass criteria are reported here in **Tables 8.19** and **8.20.**

Table 7.19 Natural Frequencies for Plane Frame

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method
152.88	152.00
208.11	208.1
1152.13	1152.0
2420.01	2420.0
11100.54	11100.0

Table 7.20 Internal Moments and Nodal Displacements

Internal Moments (M)	Nodal Displacements (δ)
M_1 = -0.5159	$\delta_{23H} = -0.0001$
M_2 = 0.2457	$\Theta_{12} = 0.0001$
M_3 = 0.3218	$\delta_{23V} = -0.0001$
M_4 = 1.00	$\Theta_{12} = 0.000$
M_5 = -1.00	$\Theta_{34} = 0.0001$
M_6 = -0.3218	
M_7 = -0.2457	
M_8 = 0.5081	

8.5 DYNAMIC ANALYSIS OF A GRID STRUCTURE

A grid structure with four members orthogonal to each other is shown in **Fig. 8.21**. Each member has flexural rigidity (EI) and torsional rigidity (GJ) equals to 1666.67 and 758.89 kN-m^2 respectively. It also has mass polar moment of inertia ($I\overline{m}$) and polar moment of inertia (J) about centroidal axis as 1250 kN-m^2 and 9.8174 x 10^{-6} m^4 respectively. The frequency analysis is to be carried out for direct nodal lumping of mass (Mo) equals to 10 kN-

Sec^2/m. It is also analysed for lumped and consistent mass criteria by considering unit weight of each member as 1kN/m.

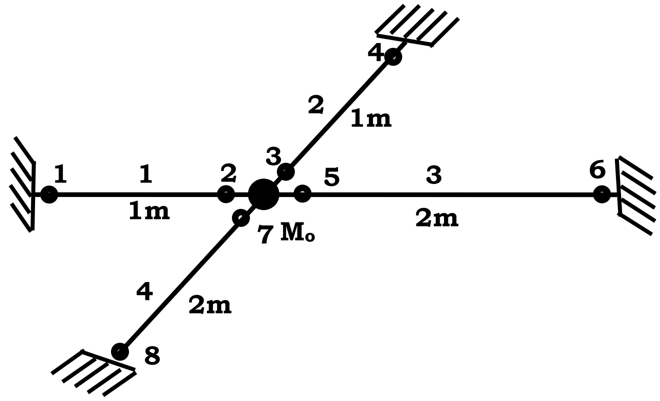


Fig. 8.21 Example of a Grid Structure

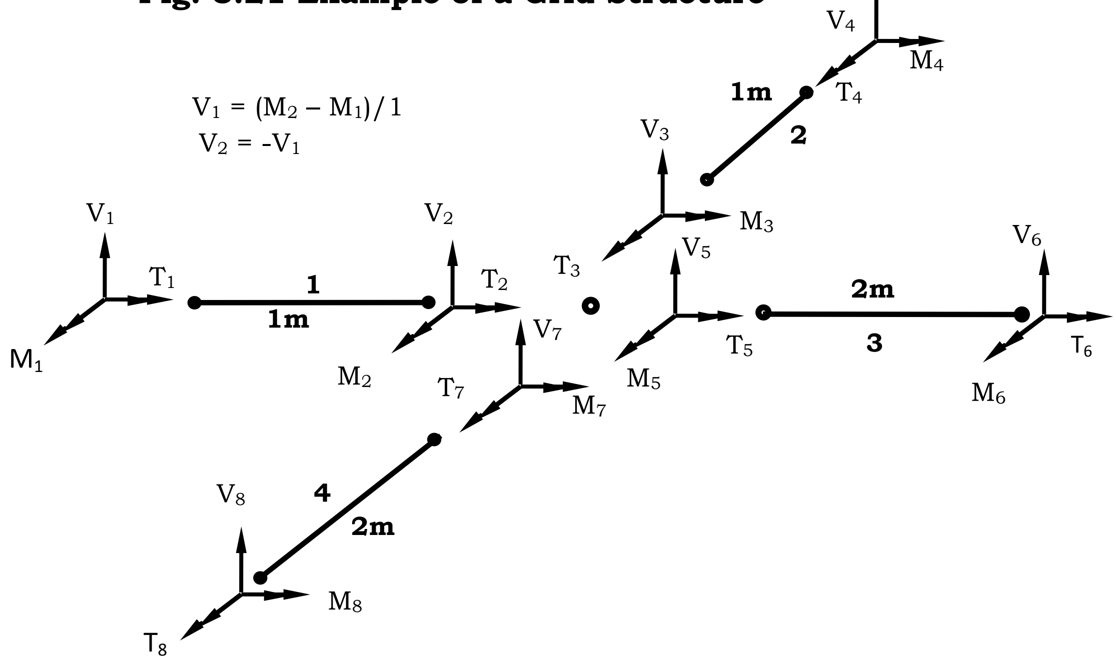


Fig. 8.22 Free Body Diagram

(i) Considering Direct Nodal Lumping Mass (DNLM Case)

The grid problem has total three possible nodal displacements at meeting point of all members i.e. Vertical displacement (δ_v), Bending rotation (θ_M) and Torsional rotation (θ_T). As there are total sixteen internal unknowns in terms of bending and torsional moments as shown in **Fig. 8.22**, the problem will have equilibrium matrix [B] of size 3 x16. Thus, the problem becomes thirteen degree statically indeterminate as per IFM.

The nodal lumping mass matrix can be written by referring Eq. (8.27) as

$$[M_{DNLM}] = \begin{array}{c} \begin{array}{ccc} \delta_v & \Theta_M & \Theta_T \end{array} \\ \begin{bmatrix} 10 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \end{array}$$

Based on the IFM the natural frequency of grid structure is found as 59.076 rad/sec, whereas using stiffness method the natural frequency is found as 59.073 rad/sec. Results obtained based on IFM for internal moments and nodal displacementare reported in **Table 8.21.**

Table 8.21 Internal Moments and Nodal Displacements

Flexural Moments (M)	Torsional Moments (T)	Nodal Displacements (δ) x 10^{-3}
M_1 = 1.00	T_1 = 0.00	δ_V = 2.72
M_2 =-0.7106	T_2 =-0.0659	Θ_M = 0.052
M_3 = 0.7106	T_3 = 0.0659	Θ_T = 0.054
M_4 = -1.00	T_4 = 0.00	
M_5 = -0.6118	T_5 = 0.0329	
M_6 = 0.4671	T_6 = 0.00	
M_7 = 0.6118	T_7 = -0.0329	
M_8 = -0.4671	T_8 = 0.00	

(ii) Considering Lumping Mass Criteria (LM Case)

Considering mass per unit length of each member as 1kN/m, based on lumping mass criteria the global lumped mass matrix can be calculated as

$$[M_L] = \begin{array}{c} \begin{array}{ccc} \delta_v & \Theta_M & \Theta_T \end{array} \\ \begin{bmatrix} 3.00 & 0 & 0 \\ & 1875.0 & 0 \\ & & 1875.0 \end{bmatrix} \end{array}$$

After deriving all the necessary matrices and after substituting in various equations the frequency values are found as reported in **Table 8.22**. After substituting each IFM based frequency value in eigen equation, the relative internal forces in each member are calculated using Matlab software. Once the internal moment in each member are calculated the relative nodal displacements are obtained as reported in **Table 8.23.**

Table 8.22 Natural Frequencies for Grid Structure

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method **[93]**
2.1463	2.1412
2.4363	2.4343
122.4798	122.4768

Table 8.23 Internal Moments and Nodal Displacements

Flexural Moments (M)	Torsional Moments (T)	Nodal Displacements (δ) x 10^{-4}
M_1 = 1.00	T_1 = 0.00	δ_V = 1.00
M_2 =-1.00	T_2 =-0.00	$\Theta_M = -0.0003$
M_3 = 1.00	T_3 = 0.00	Θ_T = 0.0003
M_4 = -1.00	T_4 = 0.00	
M_5 = 0.234	T_5 = 0.00	
M_6 = -0.234	T_6 = 0.00	
M_7 = 0.234	T_7 = -0.00	
M_8 = -0.234	T_8 = 0.00	

(iii) Considering Consistent Mass Criteria (CM Case)

Based on consistent mass concept, the global consistent mass matrix is as follows

$$[M_L] = \begin{bmatrix} 2.2286 & 0 & 0 \\ & 1250.0 & 0 \\ & & 1250.0 \end{bmatrix} \quad (\text{columns: } \delta_v,\ \Theta_M,\ \Theta_T)$$

Frequency analysis provides results as per **Table 8.24.** Moments are calculated using Matlab based module and are reported in **Table 8.25.**

Table 8.24 Natural Frequencies for Grid

Natural Frequency (ω) rad/sec	
IFM	Stiffness Method
2.6288	2.6210
2.949	2.942
142.11	142.11

Table 8.25 Internal Moments and Nodal Displacements

Flexural Moments (M)	Torsional Moments (T)	Nodal Displacements (δ) x 10^{-4}
M_1 = 1.00	T_1 = 0.00	δ_V = 0.9977
M_2 =-1.00	T_2 =-0.00	$\Theta_M = -0.0004$
M_3 = 1.00	T_3 = 0.00	Θ_T = 0.0004
M_4 = -1.00	T_4 = 0.00	
M_5 = 0.234	T_5 = 0.00	
M_6 = -0.234	T_6 = 0.00	
M_7 = 0.234	T_7 = -0.00	
M_8 = -0.234	T_8 = 0.00	

8.6 DISCUSSION OF RESULTS

Using IFM for frequency analysis and getting force mode shapes is one of the unexplored area in vibration theory of structural mechanics. After substituting each frequency value the relative internal forces are worked out and then the relative displacements are worked out by substituting the corresponding internal forces into IFM based equations. Thus, by utilizing the concept of force mode shape a direct design is feasible for the structural members which preferably vibrate under controlled frequency constraints. Some of the important observations of this chapter are as follows:

- Eigen value analysis is carried out by attempting various types of beam problems with varying boundary condition. In all the cases fixed central point mass is considered to enable comparison with the standard cases. By keeping same span, flexural rigidity and vibrating mass for all the cases the overall behavior in terms of maxima/minima frequency values, relative moments and nodal displacements are studied. For the same relative internal moment values the nodal displacements in simply supported beam are found to be higher compared to fixed one under first modal pattern. A propped cantilever beam which exhibits behavior between the two cases indicates approximately average values in moments and nodal displacement with non zero value of slope at centre of beam. A continuous beam with two equal span length having extreme ends as simply supported is also studied under same criteria. Using IFM based formulation two frequency values are worked out. In the first case, whenever the vibrating mass is on one side of the beam base line, the frequency value of propped cantilever beam and first modal frequency is found to be matching. Thus, a good agreement is found with respect to structural vibration theory. During the second vibratory motion when the both masses are on the opposite side of beam base line, the frequency value of simply supported beam case and the second modal frequency values are found equal. The values for internal moments and nodal displacements are also found to match with the standard theory of vibration.

- All the skeletal framed structures are studied under various types of mass calculations keeping all other properties identical such as: (1) Direct Nodal Lumping Mass (DNLM) where constant intensity of 10kN is lumped at necessary joints, (2) Lumped Mass (LM) is calculated as per its mass per unit length as 1kN/m for all the structural members and (3) Consistent Mass (CM) is calculated as per same mass per unit length. The frequency values for lumped mass criteria are higher than the frequency values calculated using consistent mass criteria. Relative values of internal moments and nodal displacement are also worked out for the first frequency values under LM and CM cases.
- Two pin jointed structures are solved for frequency analysis. One is having three members triangulated panel and other is of eleven

members having two rectangular panels. Both the structures are analysed for eigen values under all the three categories of mass. IFM based analysis for triangulated type panel truss structure is found to give frequency values higher than the conventional stiffness based eigen value approach. Second structure with rectangular panel behaves very rigidly in its own plane and hence requires large amount of inertial force to yield higher natural frequencies.

- For fixed footed portal frame two natural frequency values are found to be matching with the standard stiffness based eigen values corresponding to two lateral displacements for the DNLM and LM Cases. While considering consistent mass criteria, total five values of natural frequencies are calculated by including mass corresponding to rotation in the mass matrix. The relative values of moments and nodal displacements, for all the cases, are also verified for approximate shape of deflection pattern as per the frame vibration criteria as per sequential number of frequencies.
- In case of frequency analysis of unsymmetrical grid structure, all the values corresponding to three displacements are found matching with the stiffness based solutions.

CHAPTER 9

STATIC ANALYSIS OF 2D CONTINUUM STRUCTURES

9.1 COMPUTER IMPLEMENTATION OF SOLUTION STEPS

Pre- and main-processors are developed in Visual Basic to facilitate analysis of continuum structures based on integrated force method. Most of post-processing part is carried out using the Matlab software. Selection of type problem i.e. Plane stress, Plane strain or Plate bending is facilitated through a main menu which also has provision for selecting the various sub menus as per the requirement. Once the selection is done, next screen depicts Form 1, which is developed for giving an input data related to structural geometry using different GUI features. The selection of material is done by user through a menu in which provision is made for entering the value of Modulus of Elasticity (E) and Poisson's ratio (ν). Once geometrical and material input is over it is depicted on form2 with various command buttons in addition to option button to enter the type of discretization scheme. Depending on discretization pattern, number of elements and number of nodes are calculated automatically and Key Diagram is plotted using first command button named as "Key Diagram". Global degrees of freedom are then calculated. Once the various operation related to form2 are over, control of the program is transferred to the form3 which is developed for selecting the type of load. On the next form i.e, form 4 two command buttons are provided in which first one is used to draw the figure of selected element. The other things related to elemental degrees of freedom are automatically plotted using graphical commands of VB 6 with Matlab 7.4 using COM connection, which initiates active command window on which various further necessary operations are carried out. Before connection, following matrices are externally developed using Matlab editor facility:

1. Shape function matrix [N].
2. Strain displacement matrix [Z] by differentiating [N] which is directly carried out using “diff (N, x)” of Matlab command.
3. Stress linking matrix [Y] can be directly typed where the rows of the matrix correspond to number of stress components for the given problem and columns of the matrix correspond to the number of unknowns per element which represents directly internal forces.
4. The basic elemental Equilibrium matrix [Be] is worked out by integration of product of $[z]^T$ and [Y] using “int (function, lower limit, upper limit, with respect to x or y)” of Matlab command button.
5. The elemental flexibility matrix [Ge] is also worked out using same integration function. The matrix is obtained by multiplication of $[Y]^T$, material matrix [D] and [Y].
6. The assembly procedure for global equilibrium and flexibility matrices is carried out as per IFM based simplified concatenated approach. Once these matrices are assembled, they are transferred into notepad using .txt extension, which are directly called by Matlab command by giving proper path.
7. The load matrix {P} is directly developed or read from .txt file using note pad.

The analysis procedure is demonstrated here with the help a deep cantilever beam having span of 1m with cross-sectional dimensions as 0.015m x 0.15m as shown in **Fig. 9.1.** It is subjected to a point load of 10 kN at free end. Considering mild steel as material with Modulus of elasticity (E) and

Poisson's ratio (ν) as 2.01×10^8 kN/m^2 and 0.3 respectively, it is analysed here using 1 x 2 discretization.

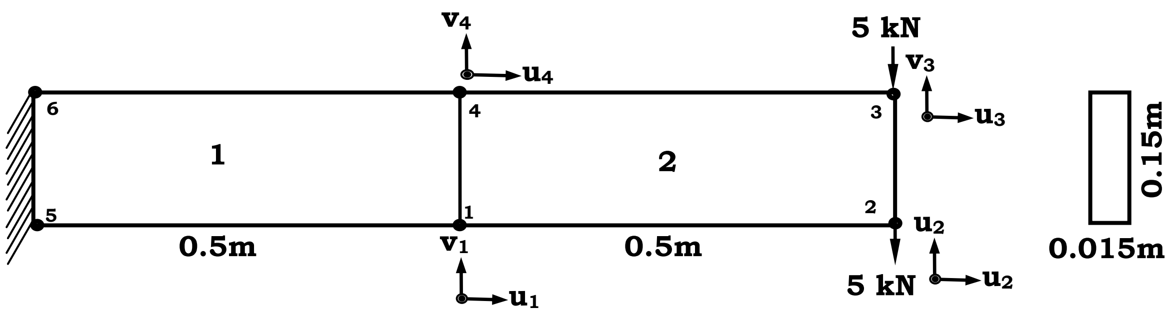


Fig 9.1 A Deep Cantilever Beam Example

Step1: Using MDI form the type of problem is selected from the provisions made for Plane stress, Plane strain and Bending of plates. **Figure 9.2** depicts a selection of deep cantilever beam from the Plane stress option.

Fig 9.2 Main Menu for Type of Structure

Step 2: Once the type of structure is selected, control is transferred to next form in which geometrical dimensions are entered and material type selection is done by using option button and text box facility as depicted in **Fig. 9.3**.

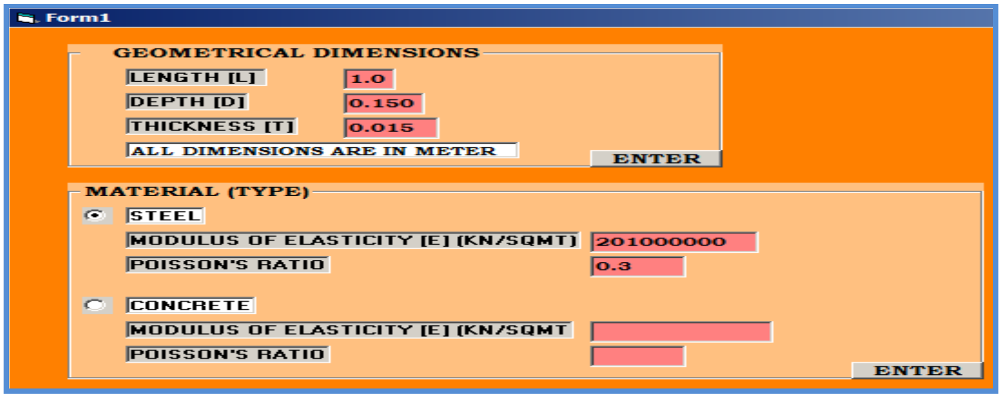


Fig. 9.3 Selection of Geometry and Material

Step 3: A diagram is depicted as shown in **Fig. 9.4** by pressing key diagram command button. Element numbering and node numbering is carried out and total eight global degrees of freedom are also shown in figure.

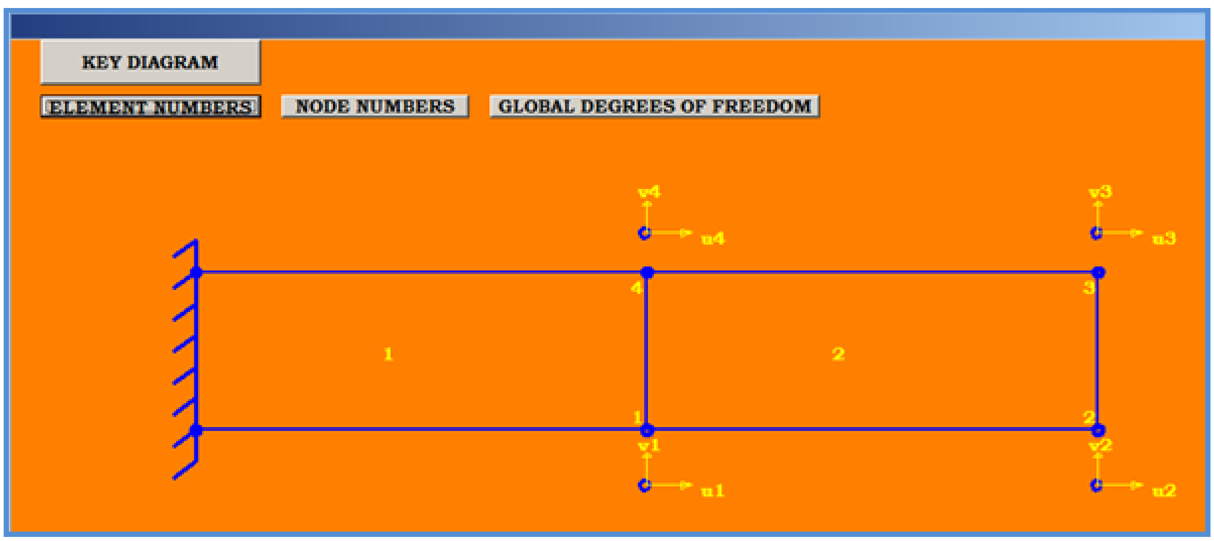


Fig 9.4 Key Diagram of Deep Cantilever Beam

Step 4: Next, by clicking any point on the form3, a form4 is depicted on screen which shows a rectangular element with five independent forces (F_1 to F_5) and eight displacement degree of freedom (**Fig. 9.5**).

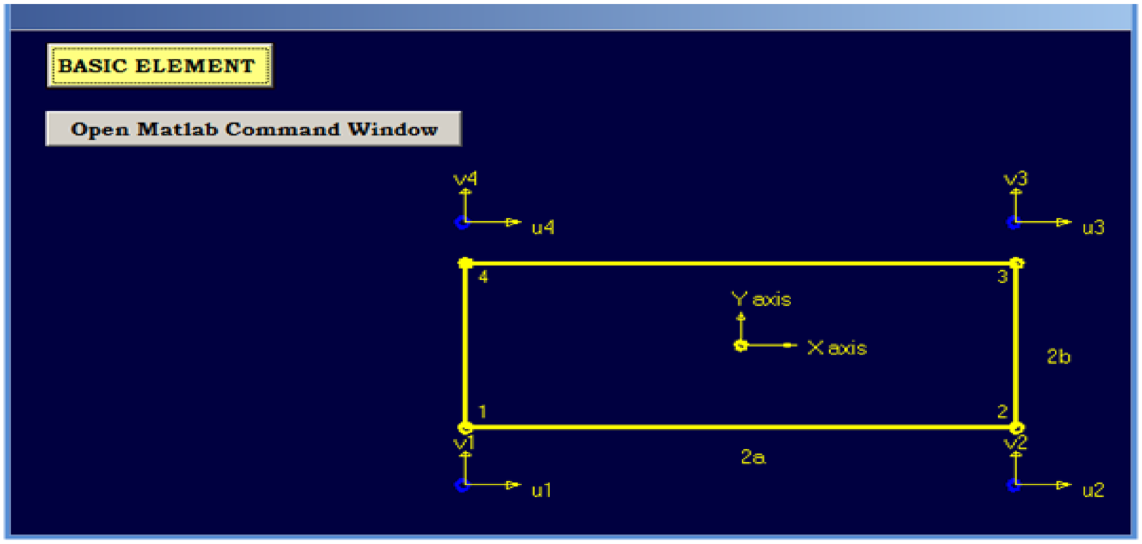


Fig 9.5 RECT_5F_8D

Step 5: Once basic element is depicted, VB program is interfaced with Matlab using COM automation server. By clicking the command button, Matlab command window editor can be seen on the form4 as depicted in **Fig. 9.6.**

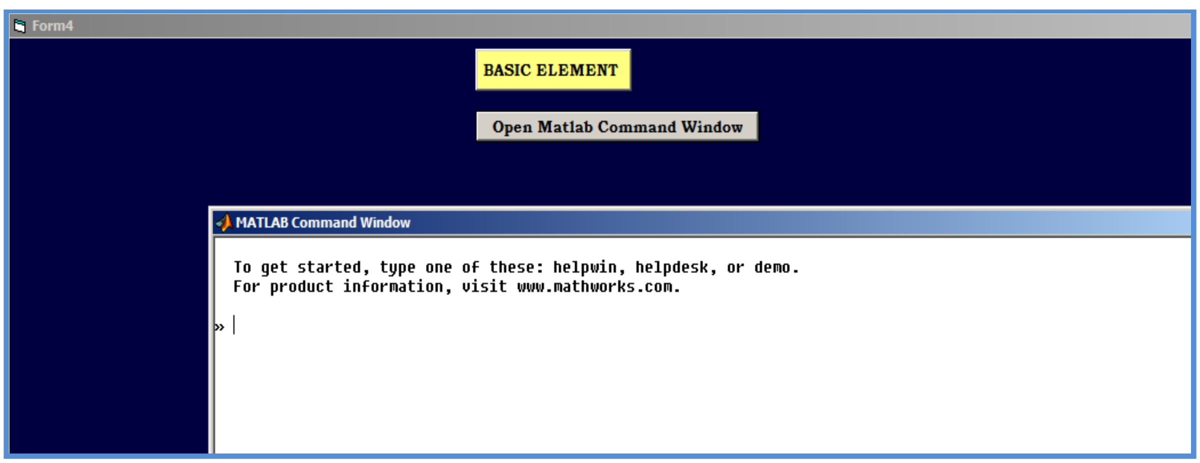


Fig. 9.6 Matlab Command Window

Step 6: Matlab command window is used to carry out the mathematical operations. By writing proper path the assembled equilibrium matrix [B] and global flexibility matrix [G] are called in the matlab editor as shown in **Fig. 9.7.**

```
>>  B = load('C:\Users\Ganesh\Desktop\CANT_BEAM.txt')

B =

    0.0750   -0.0250         0         0   -0.2500   -0.0750    0.0250         0         0   -0.2500
         0         0   -0.2500   -0.0833    0.0750         0         0   -0.2500    0.0833   -0.0750
         0         0         0         0         0    0.0750   -0.0250         0         0   -0.2500
         0         0         0         0         0         0         0   -0.2500   -0.0833    0.0750
         0         0         0         0         0    0.0750    0.0250         0         0    0.2500
         0         0         0         0         0         0         0    0.2500    0.0833    0.0750
    0.0750    0.0250         0         0    0.2500   -0.0750   -0.0250         0         0    0.2500
         0         0    0.2500    0.0833    0.0750         0         0    0.2500   -0.0833   -0.0750

>>  Gmatrix = load('C:\Users\Ganesh\Desktop\GMATRIX.txt')

Gmatrix =

    0.2488         0   -0.0746         0         0         0         0         0         0         0
         0    0.0821         0         0         0         0         0         0         0         0
   -0.0746         0    0.2488         0         0         0         0         0         0         0
         0         0         0    0.0821         0         0         0         0         0         0
         0         0         0         0    0.6468         0         0         0         0         0
         0         0         0         0         0    0.2488         0   -0.0746         0         0
         0         0         0         0         0         0    0.0821         0         0         0
         0         0         0         0         0   -0.0746         0    0.2488         0         0
         0         0         0         0         0         0         0         0    0.0821         0
         0         0         0         0         0         0         0         0         0    0.6468
```

Fig. 9.7 [B] and [G] Matrices on Matlab Editor

Step 7: Once Matlab command window is available, all the matrix operations are carried out on the command prompt. By writing proper path for the .m file it is connected to folder "CCPROG" on the desktop. Operating the module of "mtechexamplemod(B)" demands number of compatibility conditions for making the global equilibrium matrix square as depicted in **Fig. 9.8.** Thus, out of the total 10 unknowns, 2 unknowns are selected as "CodedepB" while the rest as "CodeindepB" as per the LIUT procedure.

```
>> z = mtechexamplemod(B)

codeindB =

     1     2     3     5     6     7     8    10


codedepB =

     4     9

Enter the no. of solutions wanted:2
```

Fig. 9.8 Operations Related to Compatibility Conditions

Step 8: 'z.cMatrix' of size 2 x 10 is generated which consists of the coefficients of the {β} vector. The null property CCs is checked by using the module 'z.cTransposeB'. Checking of the developed matrix is depicted in **Fig. 9.9.**

```
>> z.cMatrix

ans =

         0         0    3.3333    8.0000         0         0   -0.0000   -3.0000    9.0000    0.0000
         0         0    5.3333    2.0000         0         0   -0.0000   -3.0000    9.0000    0.0000

>> z.cTransposeB

ans =

  1.0e-016 *

  -0.8327   -0.2498   -0.8327    0.2498    0.8327    0.2498    0.8327   -0.2498
  -0.8327   -0.2498   -0.8327    0.2498    0.8327    0.2498    0.8327   -0.2498
```

Fig. 9.9 [C] Matrix and Null Property Check for [C] and [B]^T

Step 9: The multiplication of [C] matrix with global flexibility matrix [G] gives a matrix with force coefficients, which is concatenated into [B] matrix. It makes the global equilibrium matrix after normalization a square one. It is is denoted as [Smatrix] and is depicted in **Fig.9.10.**

```
>> CCmatrix = z.cMatrix*GMATRIX*1000000
CCmatrix =
   -0.0249         0    0.0829    0.0657         0    0.0224   -0.0000   -0.0746    0.0739    0.0000
   -0.0398         0    0.1327    0.0164         0    0.0224   -0.0000   -0.0746    0.0739    0.0000
>> Smatrix = [B;CCmatrix]
Smatrix =
    0.0750   -0.0250         0         0   -0.2500   -0.0750    0.0250         0         0   -0.2500
         0         0   -0.2500   -0.0833    0.0750         0         0   -0.2500    0.0833   -0.0750
         0         0         0         0         0    0.0750   -0.0250         0         0   -0.2500
         0         0         0         0         0         0         0   -0.2500   -0.0833    0.0750
         0         0         0         0         0    0.0750    0.0250         0         0    0.2500
         0         0         0         0         0         0         0    0.2500    0.0833    0.0750
    0.0750    0.0250         0         0    0.2500   -0.0750   -0.0250         0         0    0.2500
         0         0    0.2500    0.0833    0.0750         0         0    0.2500   -0.0833   -0.0750
   -0.0249         0    0.0829    0.0657         0    0.0224   -0.0000   -0.0746    0.0739    0.0000
   -0.0398         0    0.1327    0.0164         0    0.0224   -0.0000   -0.0746    0.0739    0.0000
```

Fig. 9.10 Global Equilibrium Matrix [Smatrix]

Step 10: Sinv as the inversion of [Smatrix], and Jmatrix as the transpose of Sinv are calculated. The load vector is multiplied with the Sinv to calculate {F} vector as also depicted in **Fig. 9.11.**

```
>> Jmatrix = transpose(Sinv)
Jmatrix =

    6.6667  -20.0000    1.1463   -0.8613         0         0         0   -0.4296    1.2887         0
   -0.0000  -66.6667   -0.8531   -0.8618    6.6667         0         0   -0.4298    1.2894         0
    6.6667  -20.0000    0.7167   -1.2953         0    6.6667  -20.0000   -0.1425    0.4274         0
   -0.0000 -200.0000    0.4233    0.4276    6.6667   -0.0000  -66.6667   -1.2829   -2.1512    6.6667
    6.6667   20.0000    0.7167   -1.2953         0    6.6667   20.0000   -0.1425    0.4274         0
   -0.0000 -200.0000   -0.4233   -0.4276    6.6667   -0.0000  -66.6667    1.2829    2.1512    6.6667
    6.6667   20.0000    1.1463   -0.8613         0         0         0   -0.4296    1.2887         0
   -0.0000  -66.6667    0.8531    0.8618    6.6667         0         0    0.4298   -1.2894         0
         0         0   -3.8462   16.4152         0         0         0   -0.8128    2.4383         0
         0         0    6.7254  -13.5068         0         0         0   -1.1116    3.3347         0

>> Pmatrix = load('C:\Users\Ganesh\Desktop\PMATRIX.txt')
Pmatrix =

     0
     0
     0
    -5
     0
    -5
     0
     0
     0
     0

>> Fmatrix = Sinv*Pmatrix
Fmatrix =

  1.0e+003 *

    0.0000
    2.0000
    0.0000
    0.0000
   -0.0667
    0.0000
    0.6667
   -0.0000
    0.0000
   -0.0667
```

Fig. 9.11 Loading Vector [Pmatrix], [Jmatrix] and Vector {F}

Step 11: The nodal displacement vector {δ} is worked out by multiplication of [Jmatrix], [Gmatrix] and {F} vector as depicted in **Fig. 9.12**.

```
Fmatrix =

  1.0e+003 *

    0.0000
    2.0000
    0.0000
    0.0000
   -0.0667
    0.0000
    0.6667
   -0.0000
    0.0000
   -0.0667

>> Displacement = Jmatrix*GMATRIX*Fmatrix

Displacement =

   -0.0003
   -0.0011
   -0.0004
   -0.0037
    0.0004
   -0.0037
    0.0003
   -0.0011
    0.0000
   -0.0000
```

Fig 9.12 Nodal Displacement Vector {δ}Step 12:

Step 12: The support reactions are calculated by direct multiplication of corresponding portion of globa lequilibrium matrix [Smatrix] and components of {F} vector as shown in **Fig. 9.13**.

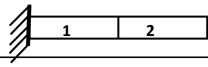
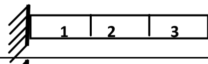
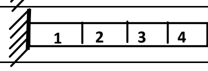
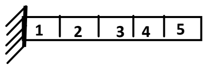
```
Support_reaction =

    0.0750   -0.0250         0         0   -0.2500
         0         0   -0.2500   -0.0833    0.0750
         0         0         0         0         0
         0         0         0         0         0
         0         0         0         0         0
         0         0         0         0         0
    0.0750    0.0250         0         0    0.2500
         0         0    0.2500    0.0833    0.0750

>> Reaction = Support_reaction*Fmatrix
??? Error using ==> mtimes
Inner matrix dimensions must agree.

>> Reaction = Support_reaction*[Fmatrix(1,1);Fmatrix(2,1); Fmatrix(3,1);Fmatrix(4,1) ;Fmatrix(5,1)]

Reaction =

  -33.3333
   -5.0000
         0
         0
         0
         0
   33.3333
   -5.0000
```

Fig 9.13 Support Reactions {R}

With finer discretization the values of nodal deflections and internal stresses are found to converge to exact solution **[94].** Results of the convergence to the exact study are reported in **Table 9.1.**

Table 9.1 Convergence of Maximum Stress and deflection

Discretization Pattern	$\frac{\sigma_{x(IFM)}}{\sigma_{x(EXACT)}}$	$\frac{\delta_{MAX(IFM)}}{\delta_{MAX(EXACT)}}$

	0.755	0.9414
	0.833	0.9669
	0.912	0.9898
	0.991	0.9988

9.2 DIFM BASED ANALYSIS OF A DEEP CANTILEVER

The same problem is analysed **(Fig.9.1)** now by using the Dual Integrated Force Method (DIFM) by using the following steps:

Step 0 – Solution strategy: The problem is divided into two RECT_5F_8D elements, thus each element is having five internal unknowns i.e. F_1 to F_5 for element number 1 and F_6 to F_{10} for element number 2 which corresponds to internal stress components N_x, N_y and N_{xy}. The problem has total 8 displacement degrees of freedom (u_1, v_1, u_2, v_2.........v_4) and 10 force degrees of freedom.

Step 1- Development of Elemental Matrices: The elemental equilibrium matrix [Be] and elemental flexibility matrix [Ge] are calculated by substituting values of a and b as 0.25m and 0.075m respectively and are obtained as

$$[Be] = \begin{bmatrix} -0.075 & 0.025 & 0 & 0 & -0.25 \\ 0 & 0 & -0.25 & 0.0833 & -0.075 \\ 0.075 & -0.025 & 0 & 0 & -0.25 \\ 0 & 0 & -0.25 & -0.833 & 0.075 \\ 0.075 & 0.025 & 0 & 0 & 0.25 \\ 0 & 0 & -0.25 & 0.0833 & 0.075 \\ -0.075 & -0.025 & 0 & 0 & 0.25 \\ 0 & 0 & -0.25 & -0.0833 & -0.075 \end{bmatrix}$$

$$[Ge] = 2.48756 \text{ x } 10^{-8} \begin{bmatrix} 1.0 & 0 & -0.3 & 0 & 0 \\ 0 & 0.33 & 0 & 0 & 0 \\ & & 1.0 & 0 & 0 \\ & & & 0.33 & 0 \\ & & & & 2.6 \end{bmatrix}$$

The Dual matrix $[D]_{difm}$ for both the elements will be same due to the same geometrical dimensions and is worked out using the following formula

$$[D]_{difm(e)} = [B_e][G_e]^{-1}[B_e]^T$$

Substituting necessary matrices in above equation for the element 1 the complete matrix is as follows

$$[D]_{difm(1)} = 10^6 \begin{bmatrix} 1.291 & 0.5384 & 0.6417 & -0.0414 & -1.1387 & -0.5384 & -0.794 & 0.0414 \\ 0 & 3.6939 & 0.0414 & 1.8281 & -0.5384 & -2.002 & -0.0414 & -3.52 \\ & & 1.291 & -0.5384 & -0.794 & -0.0414 & -1.1387 & 0.5384 \\ & & & 3.6939 & 0.0414 & -3.52 & 0.5384 & -2.002 \\ & & & & 1.292 & 0.5384 & 0.6417 & -0.0414 \\ & & \text{Symm} & & & 3.6939 & 0.0414 & 1.8281 \\ & & & & & & 1.291 & -0.5384 \\ & & & & & & & 3.6939 \end{bmatrix}$$

Similarly $[D]_{difm}$ is calculated for element – 2.

Step 2 – Solution of equations: The basic DIFM equation based on stiffness analogy is given by

$$[D]_{difm}\{\delta\} = \{P\}$$

Where $[D]_{difm}$ is the global pseudo stiffness matrix, $\{\delta\}$ is the unknown vector of size 8 x 1 i.e. $\{u_1, v_1, u_{2\ldots\ldots}v_4\}$ and $\{P\}$ is the load vector of size 8 x 1.

After substituting all the necessary matrices into above equation and solving for displacements, one gets

$$\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} -0.0003 \\ -0.0011 \\ -0.0004 \\ -0.0037 \\ +0.0004 \\ -0.0037 \\ +0.0003 \\ -0.0011 \end{Bmatrix} m$$

The solution obtained using DIFM for nodal displacements is fully matching with solution obtained by IFM approach.

Step 3 – Solution of equations: Once the displacement vector is available, the internal forces moments are worked out using either global matrix approach or individual element approach using the following equation

$$\{F\} = [G]^{-1}[B]^T\{\delta\}$$

Thus, the internal unknown force vector {F} is found as

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \\ F_7 \\ F_8 \\ F_9 \\ F_{10} \end{Bmatrix} = \begin{Bmatrix} 0.00 \\ 2000.0 \\ 0.00 \\ 0.00 \\ -66.67 \\ 0.00 \\ 666.67 \\ 0.00 \\ 0.00 \\ -66.67 \end{Bmatrix}$$

9.3 A PLANE STRESS PROBLEM OF PURE BENDING OF BEAM

A 2D plane stress example of pure bending of beam is considered here with dimensions as of 228.6 mm x 152.4 mm (9” x 6”) and thickness as 25.4 mm (1”), E = 30 x 10^6 N/mm^2, ע = 0.3 and subjected to a force with a maximum intensity of 1000 N/mm^2 as shown in **Fig. 9.14**. Because of the symmetry and antisymmetry about the y and x axes respectively only the shaded quadrant of the beam is considered for the analysis. **Fig. 9.15** depict a key diagram of discretized continuum with necessary details

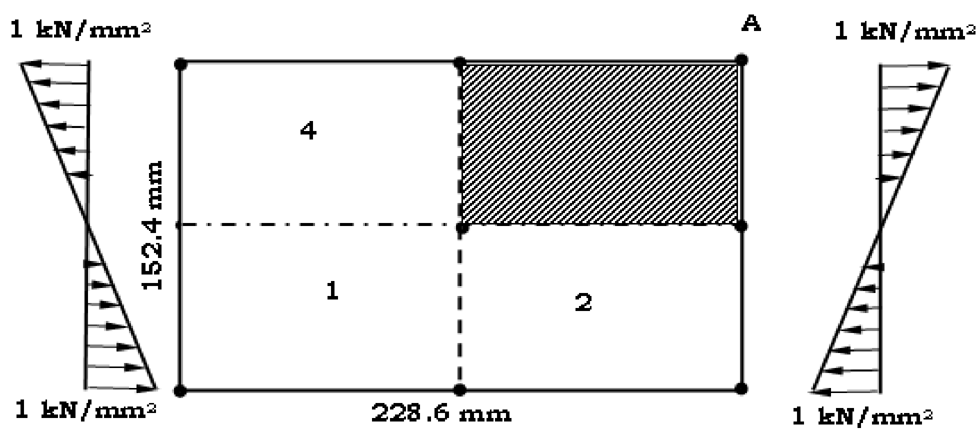


Fig. 9.14 Pure Bending of Beam

Following the steps outlined above given above results obtained for nodal displacements using IFM are compared with the exact **[96]** and FEM solutions **[96]** in **Table 9.2.**

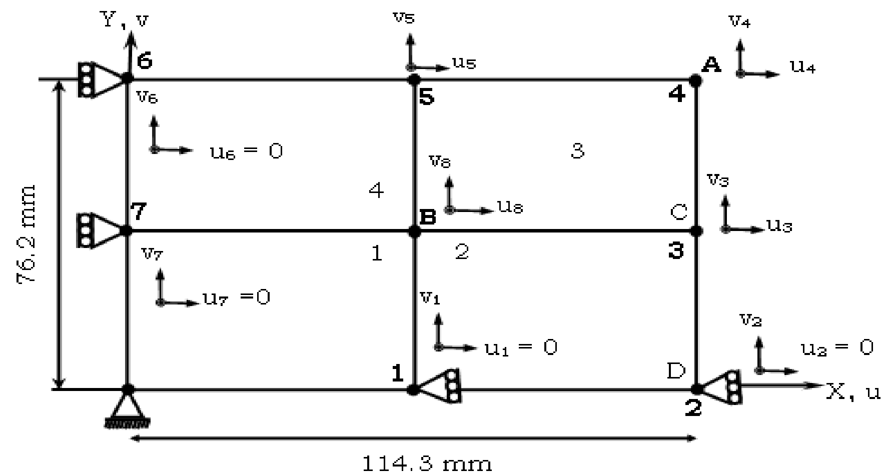


Fig. 9.15 (2 x 2) Discretization for Hatched Portion

Following the steps outlined above results obtained for nodal displacements using IFM are compared with the exact **[96]** and FEM solutions **[96]** in Table 9.2.

Table 9.2 Comparison of Nodal Displacements

Node	Horizontal /Vertical Displacement (mm)	IFM	EXACT	FEM
A	Horizontal	0.003759	0.00381	0.0034581
	Vertical	-0.003073	-0.003238	-0.002949
B	Horizontal	0.000922	0.0009525	0.0008844
	Vertical	-0.000767	- 0.0008095	-0.0007457

9.4 A PROPPED CANTILEVER STEEL PLATE EXAMPLE

A 2D plane stress example is considered here with the dimensions of plate as 750mm x 500mm x 15mm, E = 2.01 x 10^8 KN/m^2, ν= 0.25. It is subjected to a nodal force of 50 kN as shown in **Fig. 9.16.**

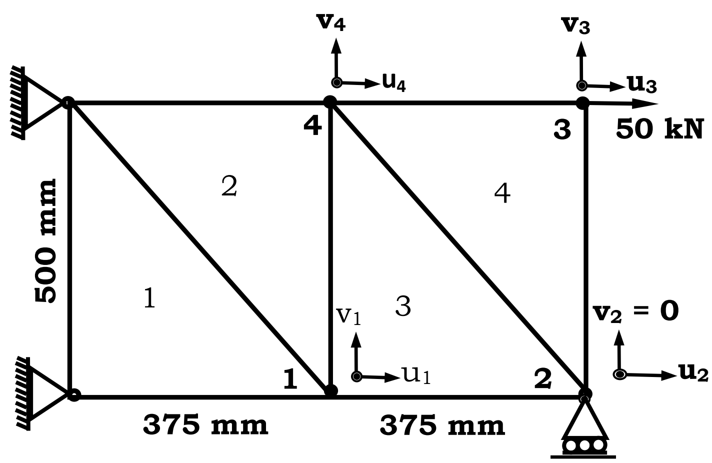


Fig. 9.16 Propped Cantilever Steel Plate Example

The propped cantilever plate example is solved here by using two different types of 2D plane stress elements i.e. (1) Triangular Element and (2) Rectangular Element.

The triangular element has six displacement degrees of freedoms (u and v at each node) and three number of independent internal unknowns (F_1, F_2 and F_3). The plate problem has total twelve force unknowns (F_1 to F_{12}) and seven displacement unknowns. Thus the complete problem requires five compatibility conditions. Discretized continuum with necessary details for triangular type of element is shown in **Fig. 9.16.**

Table 9.3 shows the values of internal force unknowns corresponding to in-plane stresses in each element, while **Table 9.4** shows the displacement values at nodes 2, 3 and 5.

Table 9.3 Internal Force Unknowns (TRI_3F_6D)

Element Number	Values of Internal Unknowns
1	F_1 = 54.33, F_2 = 13.58 , F_3 = 13.48
2	F_4 = 145.66, F_5 = 15.27 , F_6 = 29.48
3	F_7 = 21.83, F_8 = -15.67 , F_9 = 13.78
4	F_{10} = 178.11, F_{11} = -38.90 , F_{12} = 29.17

Table 9.4 Nodal Displacements (TRI_3F_6D)

Displacement (m) x 10^{-4}	IFM	FEM
u_2	-0.0178	--
u_3	0.566	0.5366
v_3	-0.166	-0.1182

Rectangular element RECT_5F_8D is also used here for the solution. As the domain has globally seven displacement degrees of freedom and ten force degrees of freedom, three compatibility conditions are sufficient for solving the problem.

Table 9.5 shows the values of internal unknowns while **Table 9.6** shows the displacement ratio between IFM based values and available FEM solution **[97]** at node 2 and 3.

Table 9.5 Internal Force Unknowns (RECT_5F_8D)

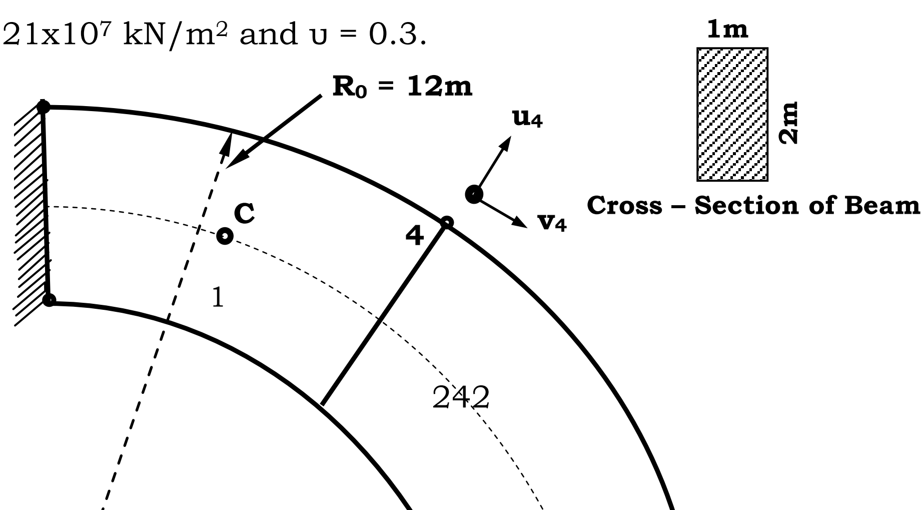
Element Number	Values of Internal Unknowns
1	$F_1 = 100.0$, $F_2 = 44.427$, $F_3 = 19.64$, $F_4 = -5.40$, $F_5 = 37.86$
2	$F_6 = 100.0$, $F_7 = 214.80$, $F_8 = -34.16$, $F_9 = -48.95$, $F_{10} = 37.86$

Table 9.6 Nodal Displacement Ratio (RECT_5F_6D)

Displacement	IFM	FEM
u_2	-0.163	--
u_3	0.5720	0.5366
v_3	-0.136	-0.1182

9.5 A CURVED BEAM EXAMPLE

A cantilever curved member shown in **Fig. 9.17** clamped at $\alpha = 0^o$ is subjected to a moment equals to 600 kN-m at $\alpha = 90^o$. The cross section of beam is 1m x 2m with an inner radius R_i = 10m and outer radius R_o = 12m. with E = $21x10^7$ kN/m^2 and υ = 0.3.



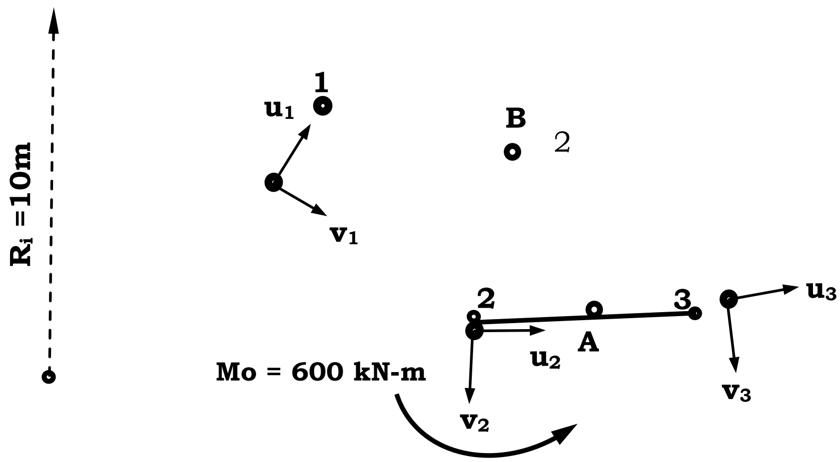


Fig 9.17 A Curved Beam Example

Step 0 – Solution strategy: The curved beam which is subjected to a pure bending moment is attempted using rectangular element having two opposite edges (1-2, and 3-4) as curved and the remaining two (1-4, 2-3) as straight. Polar coordinate system (r, α) is used here. The basic element has eight displacement degrees of freedom. Deducting three rigid body rotations, five independent unknowns are considered here and therefore the element is named as CURV_5F_8D.

The problem has total eight displacement degrees of freedom ($u_1, v_1 \ldots\ldots u_4, v_4$) and ten force degrees of freedom (F_1, $F_2 \ldots.. F_{10}$). Two additional compatibility conditions are developed using matlab based module named as "mtechexamplemod(B)". Following the standard IFM procedure [Smatrix] is generated , inverted and multiplied by the {P} vector to get the stresses and using the same the nodal displacements are calculated. Stresses σ_r amd σ_α are calculated at point C while nodal displacements δ_u and δ_v are worked out at point A. Results are compared in **Table 9.7** with the exact solution available in the literature **[98]**.

Table 9.7 Stress and Displacement Ratio

Point	Stress Or Displacement	$\frac{IFM}{EXACT}$

C	σ_r	0.913
	σ_α	0.933
A	δu	0.973
	δv	0.966

9.6 A PLANE STRAIN EXAMPLE OF BOX CULVERT

A culvert shown in **Fig. 9.18,** is now analysed by using plane strain criteria for its major principal stress and maximum displacement. The modulas of elasticity E = 210 Gpa and Poisson's ratio $\nu = 0.3$.

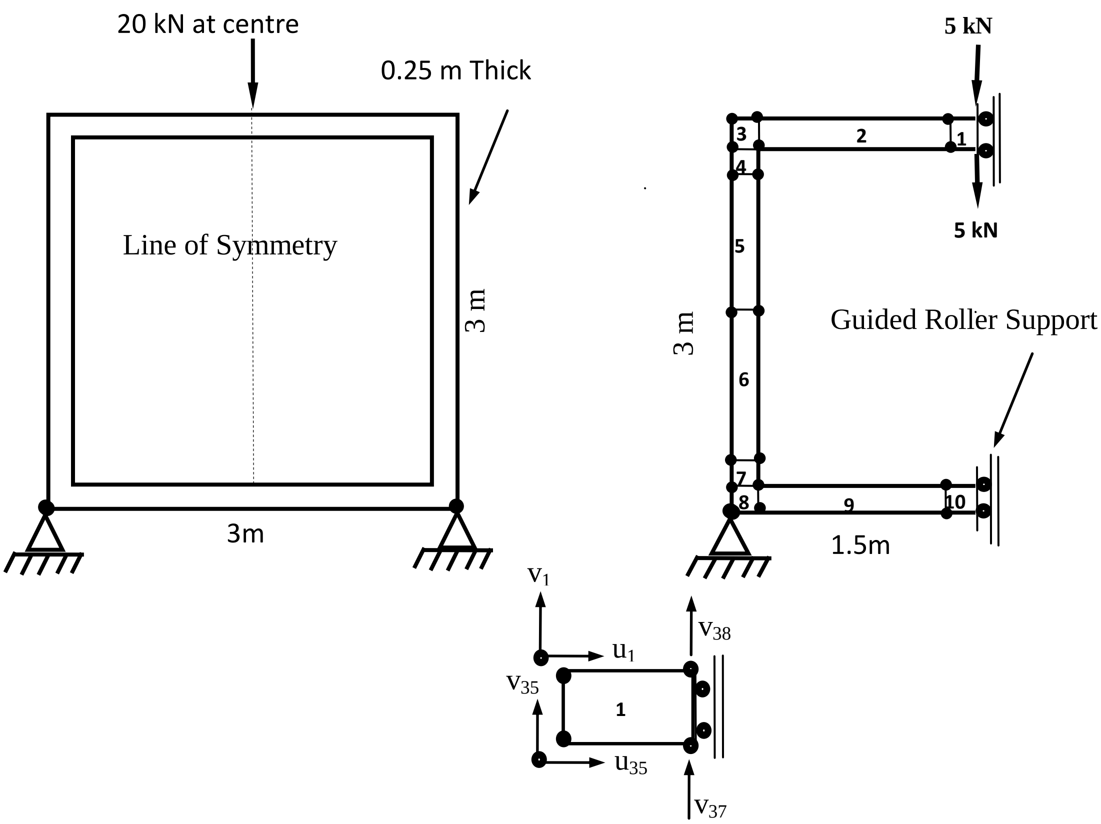


Fig 9.18 A Plane Strain Culvert Example

Due to symmetry only half of the problem is discretized into 10 rectangular elements as shown in Fig. 9.18. It results in total 38 global displacement degrees of freedom and 50 force degrees of freedom. The change in formulation is due to material matrix [D], which is required in calculation of flexibility matrix.

The elemental flexibility matrix is obtained by discretizing the complementary strain energy.

$[G^e_{ps}]=\int_s [Y]^T[D_{ps}][Y]dxdy$ ... (9.1)

where [Y] is the force interpolation function matrix and $[D_{ps}]$ is the material property matrix which for a plane strain condition is given below.

$$[D_{ps}] = \frac{1-\nu^2}{E}\begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{-2}{1-\nu} \end{bmatrix} \quad \text{... (9.2)}$$

Where E and ν are the modulus of elasticity and Poisson ratio respectively.

The solution procedure requires development of elemental matrices, development of global matrices, solution of equations and calculation of nodal displacement from forces.

The major principal stress is under point load in element number 1. Comparing with FEM solution **[83],** the ratio of IFM/FEM is found as 1.11. Nodal displacements of element 1 are reported in **Table 9.8.**

Table 9.8 Nodal Displacements for Element 1

Displacement	IFM (m)
u_1	9.36E-06
v_1	0.0002
u_{35}	-9.8E-06
v_{35}	-0.00019
v_{36}	-0.00021
v_{37}	-0.00021

9.7 STRATEGY ADOPTED FOR PLATE BENDING PROBLEMS

Various types of rectangular plate bending problems are solved here by changing support and loading conditions. Simply supported and fixed plate problems are considered under different loading conditions i.e. Central point load (CPL), Uniform pressure loading (UPL), Patch loading (PL) and Uniformly varying pressure loading (UVPL). Nodal displacements and internal moments are calculated at different points in plate domain. Use of symmetry is made in IFM based analysis. After convergence study, 5 x 5 discretization is considered for solving most of the problems. RECT_9F_12D having nine force and 12 displacement degrees of freedom is employed here for finding the solution. The element has important element properties such as no shear locking, full row rank and repeatating component behavior in each column at equal interval. All the necessary matrices are derived using direct integration technique available in matlab module as “int(function, lower limit, upper limit, with respect to x or y)” in element domain ((-a,-b) to (a, b)). The necessary compatibility conditions are worked out by using Matlab Auto compatibility condition development program. These CCs are concatenated into [B] matrix to make it a square matrix. After inverting the [S] matrix, the {F} vector and internal moments (M_x, M_y and M_{xy}) are worked out followed by the calculation of nodal displacements. Finally, by using matlab 2D moment contours and 3D deformed shapes are drawn.

9.8 SQUARE PLATE BENDING EXAMPLES

Total four examples of mild steel plate bending are considered here to validate the proposed formulation. Two examples of simply supported plate and others of clamped boundary condition are considered under CPL and ULP cases as shown in **Figs. 9.19** and **9.20**. Each plate is subjected to point load of 1kN and uniformly distributed loading of 1kN/m^2. The geometrical dimensions of isotropic plate are considered as 4000 mm x 4000 mm x 200 mm with E = 2.01 x 10^{11} N/m^2 and ν = 0.23.

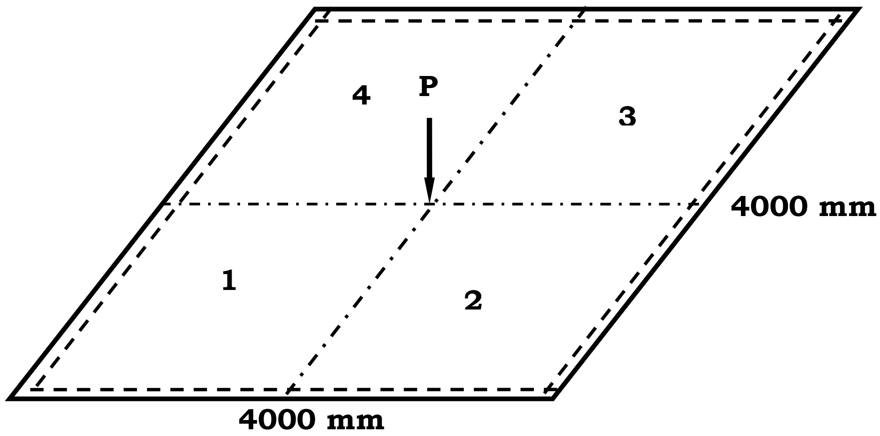


Fig 9.19 A Simply Supported Plate Example(SQR_SS_PTL)

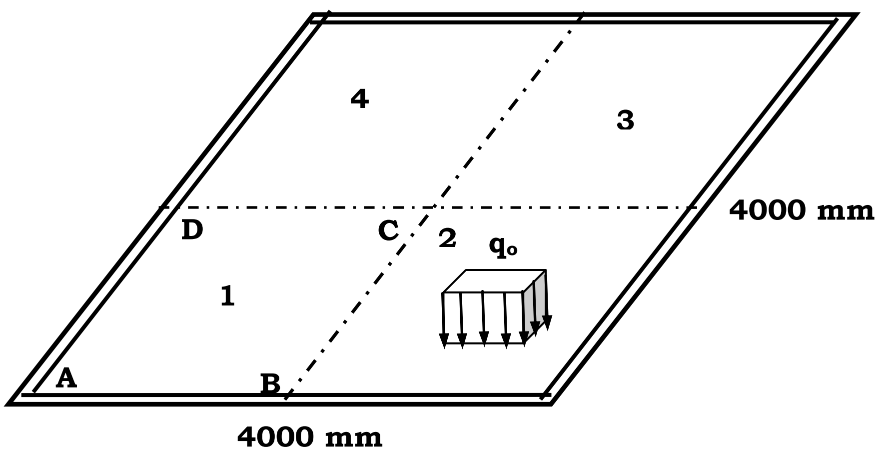


Fig 9.20 A Clamped Plate Example (SQR_CLAMPED_UDL)

Due to two way symmetry only left bottom most quadrant ABCD is analysed by considering appropriate boundary conditions at symmetry lines. Convergence study for deflection at the centre of the plate is carried out for SQR_SS_PTL case with 2 x 2, 3 x 3, 4 x 4 and 5 x 5 discretization schemes. **Fig. 9.21** depicts 2 x 2 discretization with possible displacements.

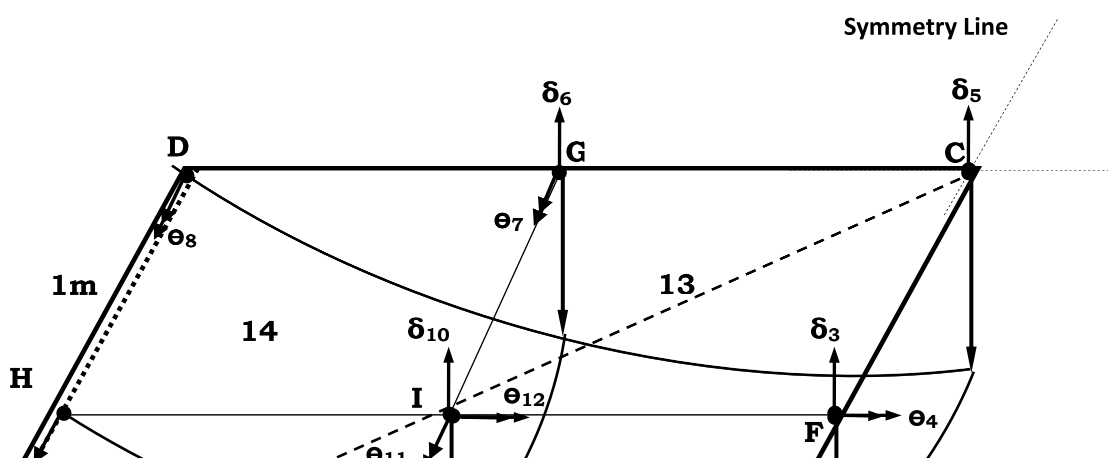


Fig 9.21 A 2 x 2 Discretization Scheme For SS Plate

Step 0 - Solution Strategy: The square plate with simply supported boundary condition is divided into four main elements 1, 2, 3 and 4 as depicted in **Fig 9.20**. The quadrant 1 is discretized into four sub-elements such as 11, 12, 13 and 14 as depicted in **Fig 9.21.** The discretized domain has total 12 non zero displacements ($\theta_1, \theta_2 \ldots \delta_5 \ldots\ldots \theta_{12}$). As RECT_9F_12D element has total twelve displacement and nine force degrees of freedom, the problem needs total 24 compatibility conditions for solution.

Step 1-Development of elemental matrices: By referring rectangular element (RECT_9F_12) formulation given in Chapter5, elemental equilibrium matrix [Be] and elemental flexibility matrix [Ge] are calculated by substituting values of a and b equal to 0.5m as

$$[Be] =$$

$$\begin{bmatrix} 0 & 0.5 & 0 & -0.083 & 0 & 0 & 0.5 & -0.0833 & 2 \\ -0.5 & 0.25 & 0.0833 & -0.04167 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -0.5 & 0.0833 & 0.25 & -0.04167 & 0 \\ 0 & -0.5 & 0 & 0.0083 & 0 & 0 & 0.5 & 0.0833 & -2 \\ 0.5 & 0.25 & -0.083 & -0.0416 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -0.5 & -0.083 & 0.25 & 0.04167 & 0 \\ 0 & -0.5 & 0 & -0.0833 & 0 & 0 & -0.5 & -0.0833 & 2 \\ 0.5 & 0.25 & 0.083 & -0.0416 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5 & 0.0833 & 0.25 & -0.4167 & 0 \\ 0 & 0.5 & 0 & 0.0833 & 0 & 0 & -0.5 & 0.0833 & -2 \\ -0.5 & 0.25 & -0.083 & 0.04167 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5 & -0.083 & 0.25 & -0.04167 & 0 \end{bmatrix}$$

$$[Ge_1] =$$

$$10^{-9}\begin{bmatrix} 7.5 & 0 & 0 & 0 & -1.72 & 0 & 0 & 0 & 0 \\ & 0.625 & 0 & 0 & 0 & -0.143 & 0 & 0 & 0 \\ & & 0.625 & 0 & 0 & 0 & -0.143 & 0 & 0 \\ & & & 0.0525 & 0 & 0 & 0 & -0.01197 & 0 \\ & & & & 7.5 & 0 & 0 & 0 & 0 \\ & & & & & 0.625 & 0 & 0 & 0 \\ & \text{Symm} & & & & & 0.625 & 0 & 0 \\ & & & & & & & 0.0525 & 0 \\ & & & & & & & & 18.45 \end{bmatrix}$$

Step 2- Development of Global Matrices: The compatibility matrix for the four elements is obtained from the displacement deformation relations (DDR) i.e. $\beta = [B]^T\{\delta\}$. In the DDR, 36 deformations which correspond to 36 force variables are expressed in terms of 12 displacements ($\theta_1, \theta_2 ...... \delta_5 ...... \theta_{12}$). The problem requires 24 compatibility conditions [C] that are obtained by eliminating the 12 displacements from the 36 DDR's, which are calculated by using auto-generated matlab based computer program by giving input as upper part of global equilibrium matrix [B].

The Global Flexibility matrix for the problem is obtained by diagonal concatenation of the four elemental flexibility matrices as

$$[G] = \begin{bmatrix} Ge_1 & & & \\ & Ge_2 & & \\ & & Ge_3 & \\ & & & Ge_4 \end{bmatrix}$$

Step 3 Calculations of forces {F}: By multiplying compatibility matrix [C] and global flexibility matrix [G] the bottom most part of the global equilibrium matrix is obtained. Assembling both gives complete [S] matrix of size 36 x 36, which comprises of EEs and CCs. The internal forces corresponding to first sub-element number 11 are obtained by using MatLab's inverting procedure and are found as follows:

$$\{F\} = [15.41\ 30.87\ 30.87\ 61.74\ 15.41\ 30.87\ 30.87\ 61.74\ 60.4522]^T$$

After substituting the values of all internal unknowns in moment equations (**Fig 9.22),** one can calculate moments at corners A, E, H and I by substituting values of coordinates. These moments are compared with the standard small deflection theory of plate results **[99]**. A good agreement is found.

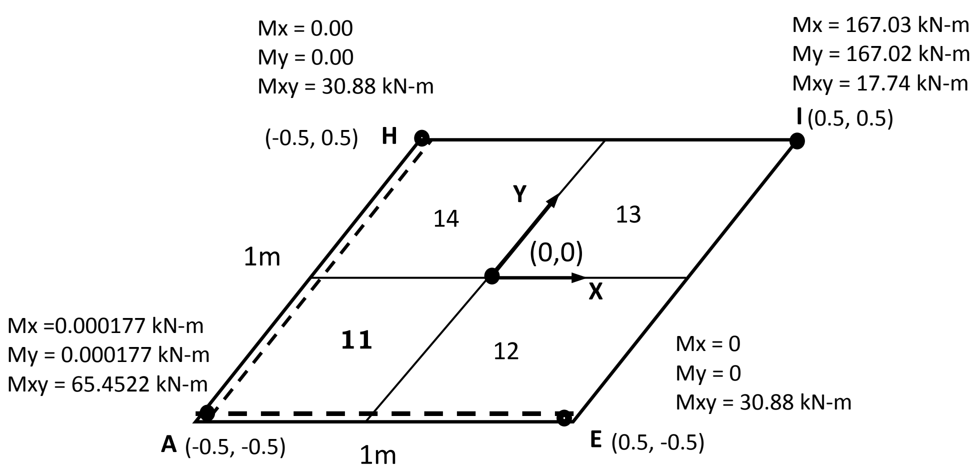


Fig. 9.22 Moment Values at Corners A, E, I, H of Element 1

Step 4 Calculation of displacements {δ}: The nodal displacements are calculated by using relation {δ} = [J][G]{F}, where [J] = m rows of matrix $[[S]^{-1}]^{T}$. Values obtained using are compared with those given by the standard classical plate theory **[99]**. A comparison of those given vertical displacement is presented in **Table 9.9.**

Table 9.10 Comparison of Results of Nodal Displacements

Displacement (m)	IFM	Exact
δ_3	0.8879×10^{-7}	0.8144×10^{-7}
δ_5	1.3911×10^{-6}	1.1843×10^{-6}
δ_6	0.8879×10^{-7}	0.8144×10^{-7}
δ_{10}	0.6911×10^{-7}	0.6616×10^{-7}

After convergence study **(Fig 9.23),** all the related plate bending examples are solved using finer discretization as depicted in **Fig. 9.24.**

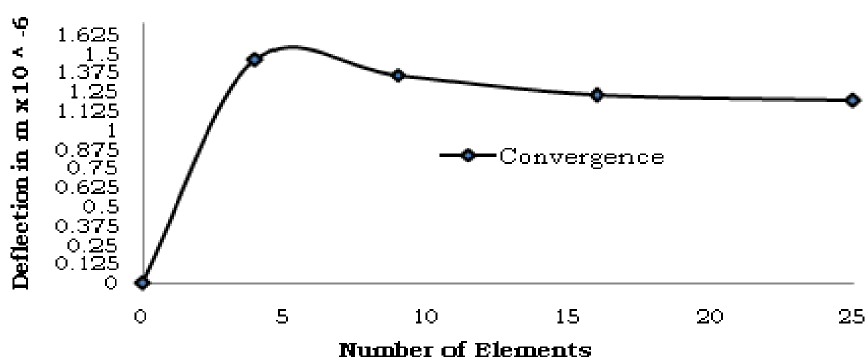


Fig. 9.23 Plot for Convergence

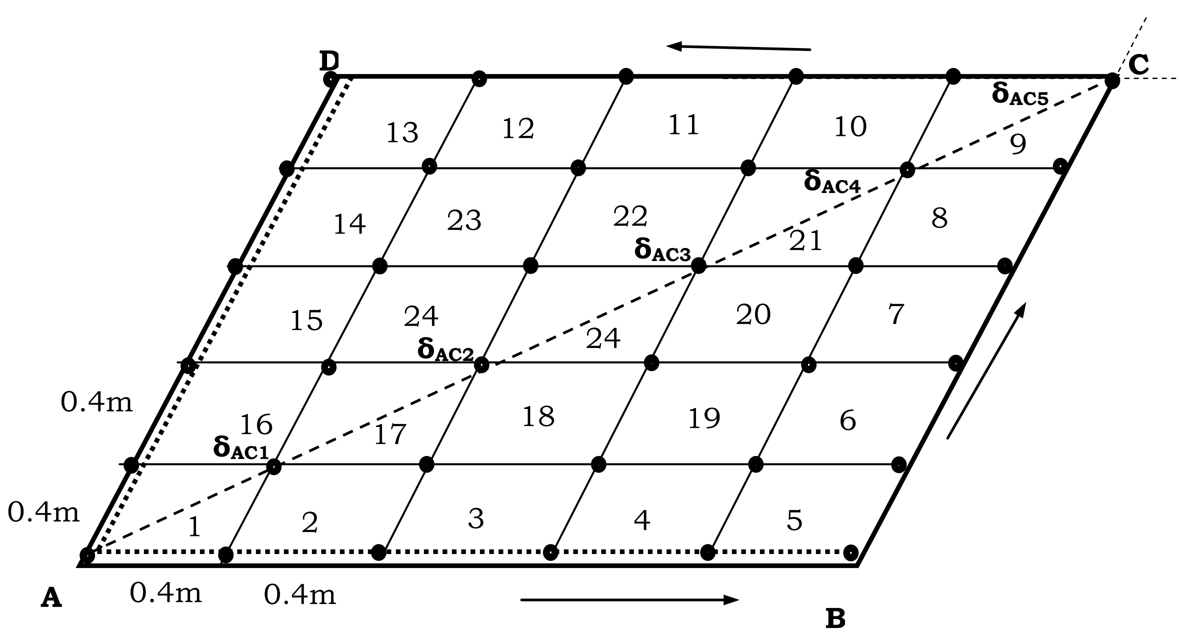


Fig 9.23 Numbering of Nodes and Elements of (5x5) Discretization

Using matlab software, 2D moment contours and 3D deflected shape are drawn using the values of nodal moments Mx and Mxy and and deflections at all the 36 nodes of the left bottom most quadrant as shown in **Figs 9.24, 9.25** and **9.26** respectively.

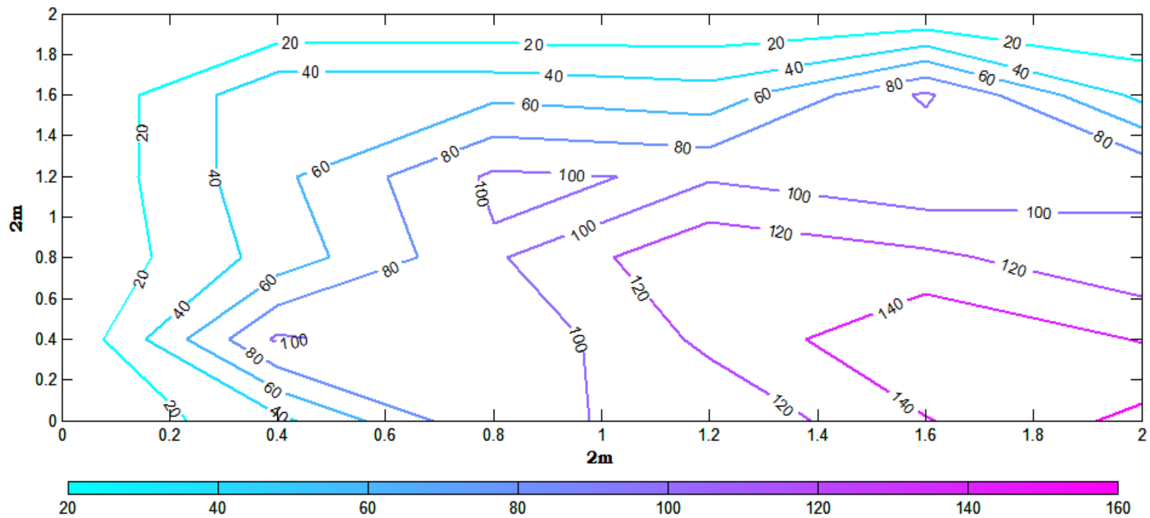


Fig. 9.24 Moment Contours for Mx in SS plate (CPL Case)

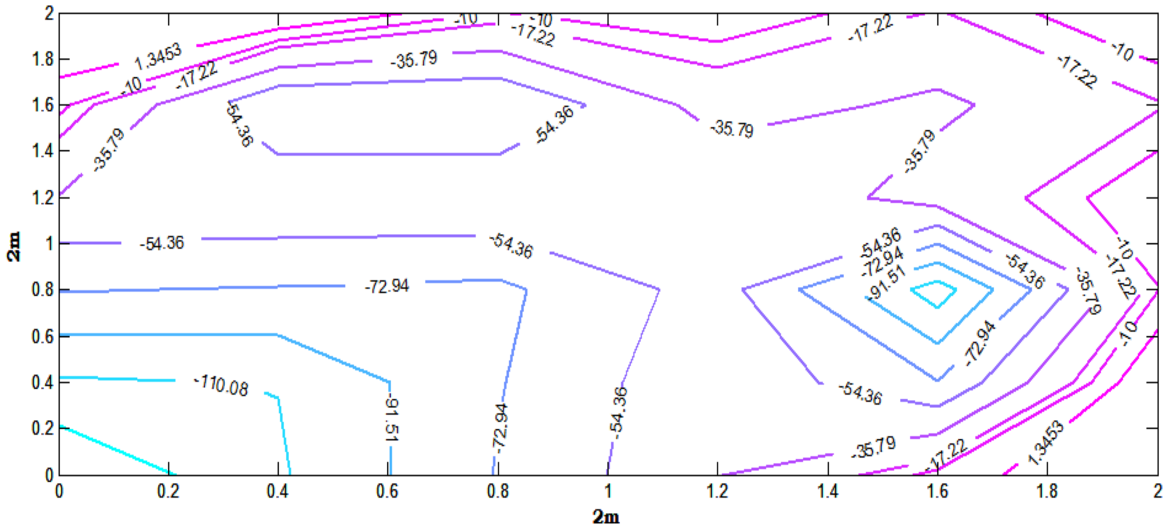


Fig. 9.25 Moment Contours for Mxy in SS plate (CPL Case)

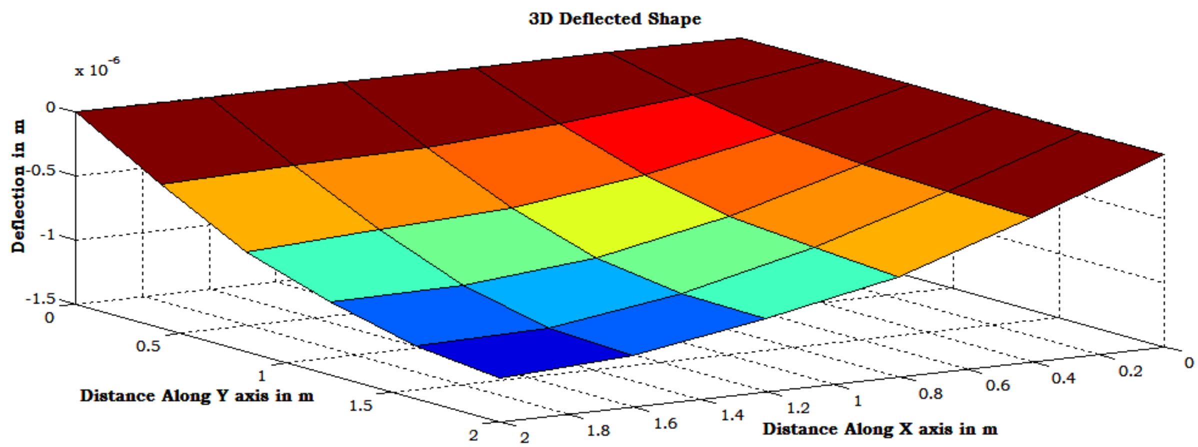


Fig. 9.26 3D Deflected Shape for Square Plate Under CPL Case

Next, a square plate with simply supported boundary condition is analysed under UDL of intensity 1kN/m^2. The IFM based displacement values are obtained and compared with those given by the standard classical plate theory **[99]** in **(Table 9.11).**

Table 9.11 Comparison of Results of Nodal Displacements

Displacement (m)	IFM	Exact
δ_3	5.681×10^{-6}	5.345×10^{-6}
δ_5	7.887×10^{-6}	7.581×10^{-6}
δ_6	5.681×10^{-6}	5.345×10^{-6}
δ_{10}	4.104×10^{-6}	3.909×10^{-6}

Using matlab software 2D moment curves and 3D deflected shape are drawn by calculating the values of nodal moments and deflections at all the 36 nodes of the left bottom most quadrant for 5 x 5 discretization as shown in **Figs 9.27, 9.28 and 9.29.**

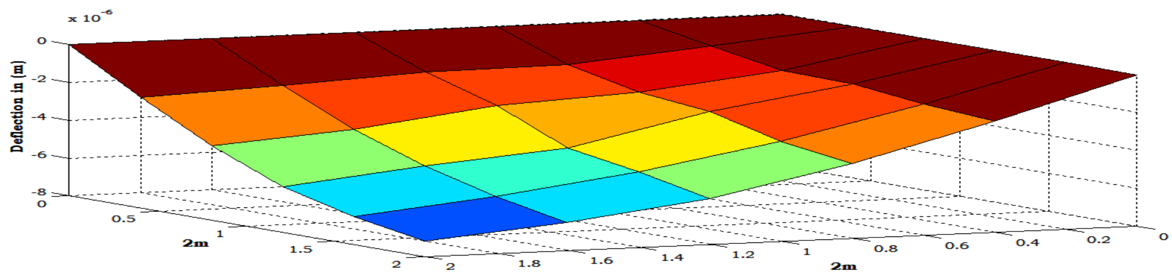


Fig. 9.27 Deformed Shape of SS Quarter Plate Subjected to UDL

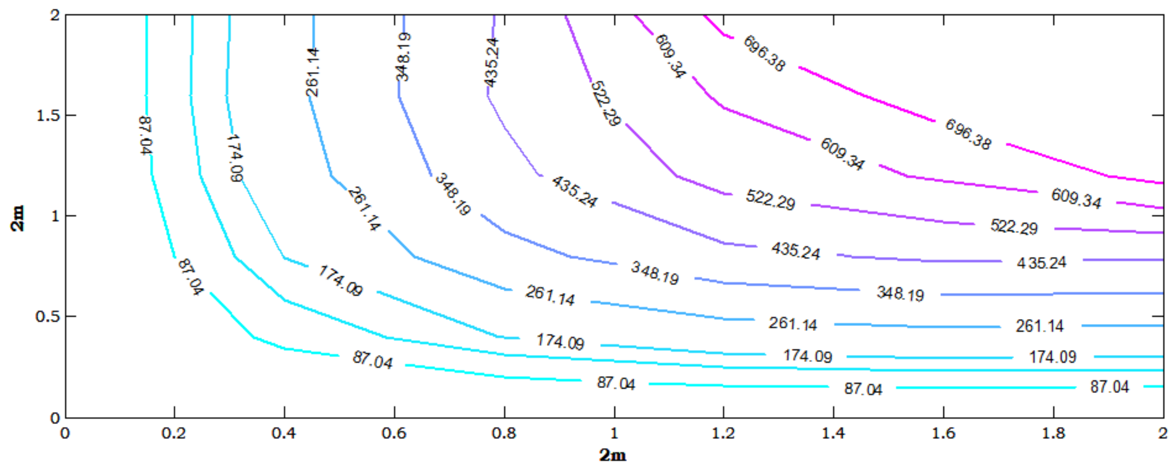


Fig. 9.28 Moment Contours for Mxx in SS Plate Under UDL

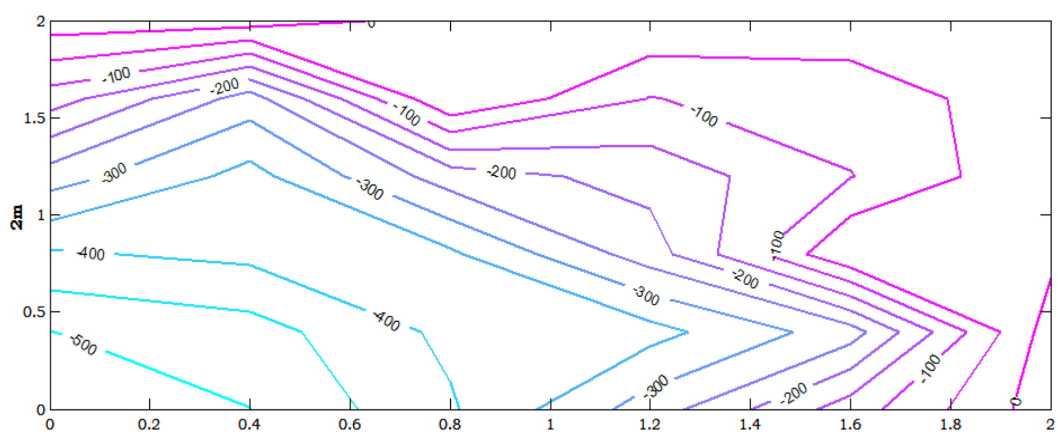


Fig. 9.29 Moment Contours for Mxy in SS Plate Under UDL

Following the same steps a square plate with clamped boundary conditions is analysed for Central Point Load (CPL) of 1kN and UDL of intensity 1kN/m^2 by considering 5 x 5 discretization scheme.

Displacement values at the centre of plate are obtained and are compared with those given by the standard classical plate theory. The deflection ratio (IFM/Exact) is found as 1.101 for CPL and 1.09 for UDL. The moment contours for Mxx are depicted in **Fig. 9.30.**

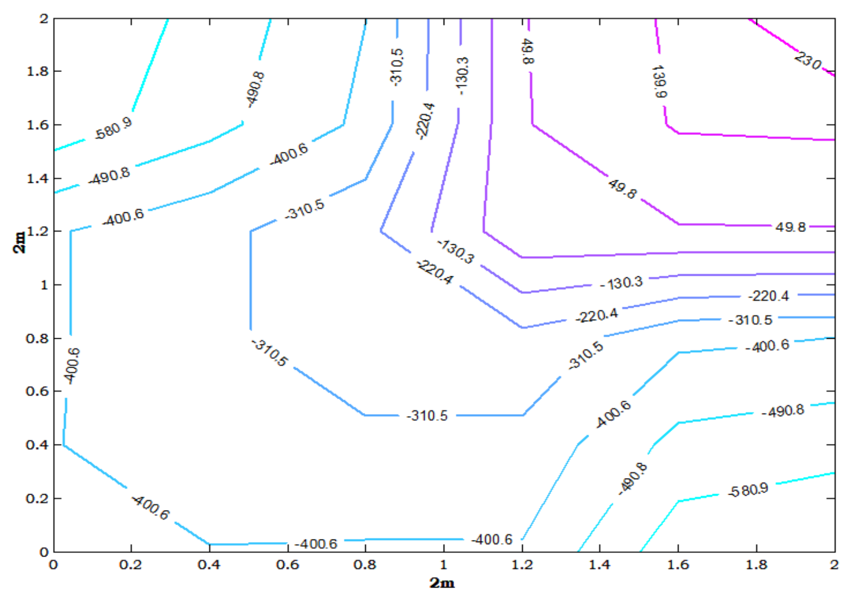
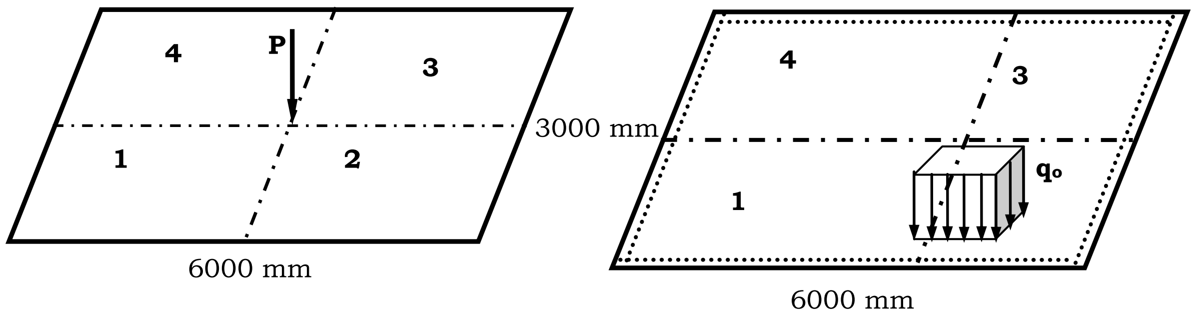


Fig. 9.30 Moment Contours for Mxx in Clamped Plate Under UDL

9.9 RECTANGULAR PLATE BENDING EXAMPLES

Total four examples of rectangular plate are considered here to validate the proposed method. Two examples are of simply supported and others are of clamped boundary conditions **(Fig.9.31).** Each plate is subjected to a point load of 10kN and uniform distributed loading of 10kN/m^2. The geometrical dimensions isotropic steel plate are considered as 6000 mm x 3000 mm x 200 mm with E as 2.01 x 10^{11} N/m^2 ande, ν = 0.23.



(a) Simply Supported Plate **(b) Fixed Plate**

Fig. 9.31 Rectangular Plate Bending Examples

Using two way symmetry with 5 x5 discretization for quarter plate IFM based solutions for rectangular plate problems are obtained. Three dimensional plots for deformed shape and two dimensional plots for moment contours are included here for simply supported plate under CPL and UPL cases in **Figs 9.32** to **9.33.**

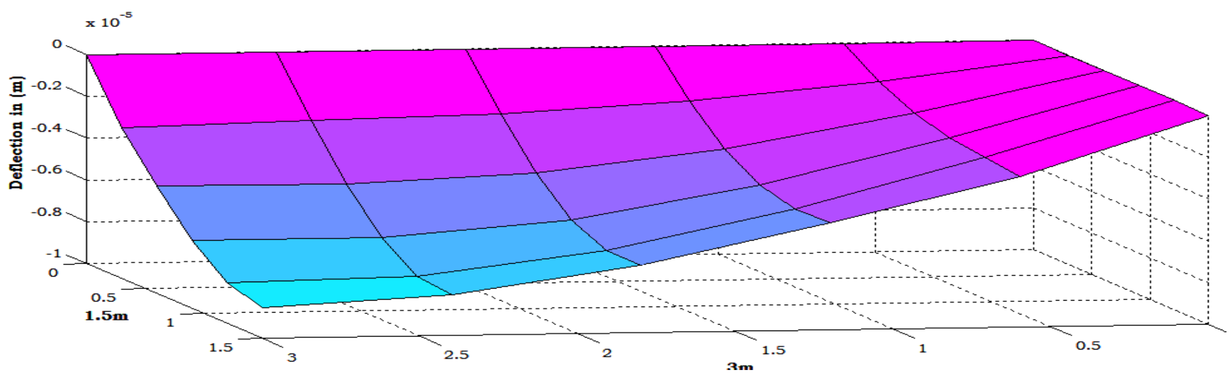


Fig. 9.32 Deformed Shape of SS Quarter Plate Under CPL

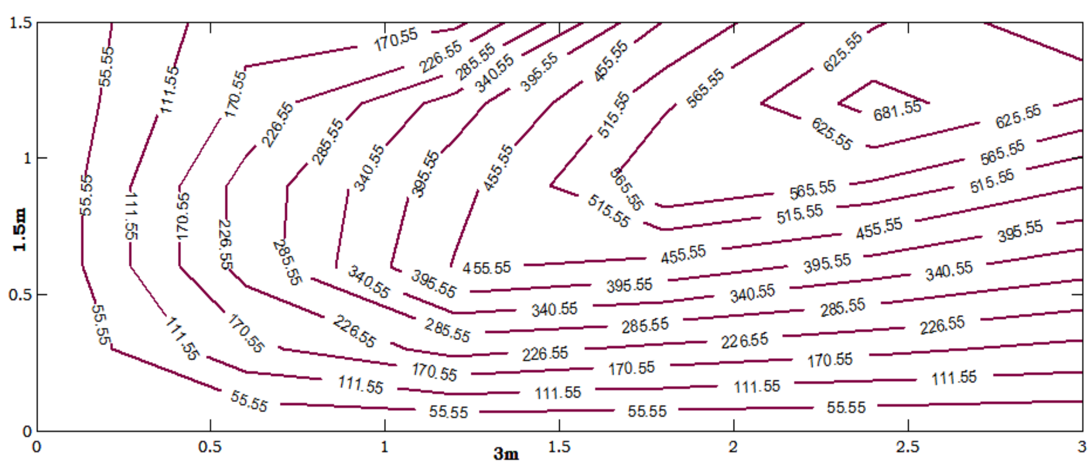


Fig 9.32 Moment Contour for Mx for SS plate under CPL

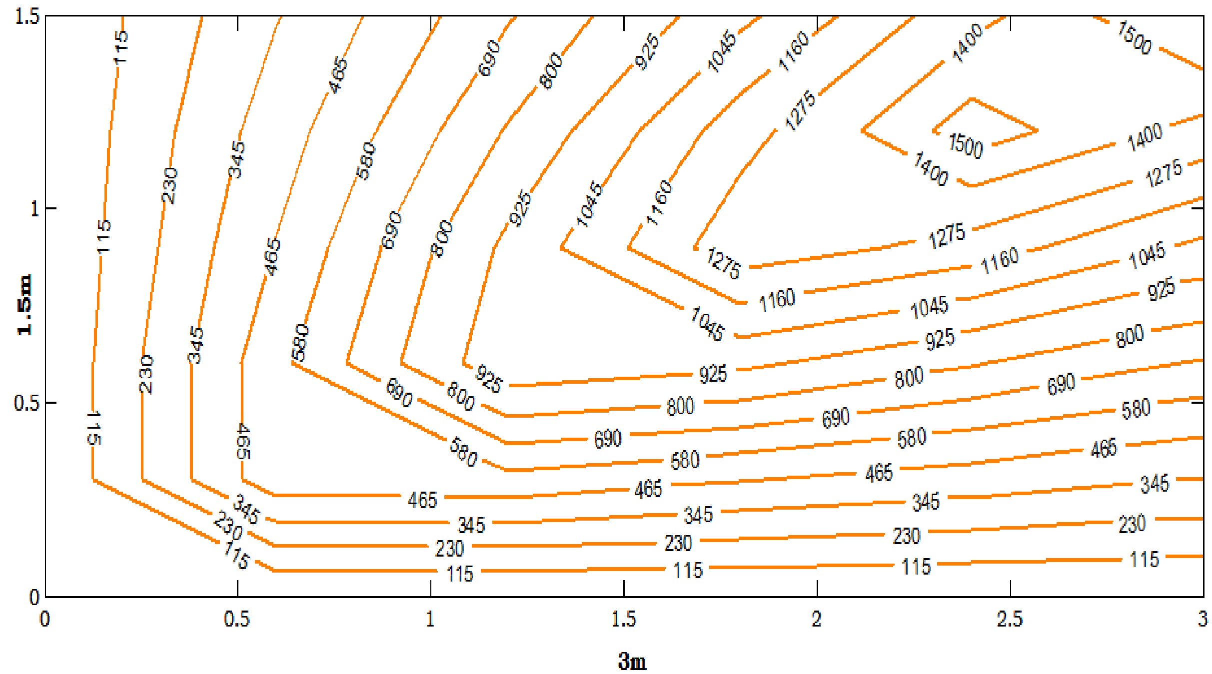


Fig 9.33 Moment Contour for Myy in SS plate under CPL

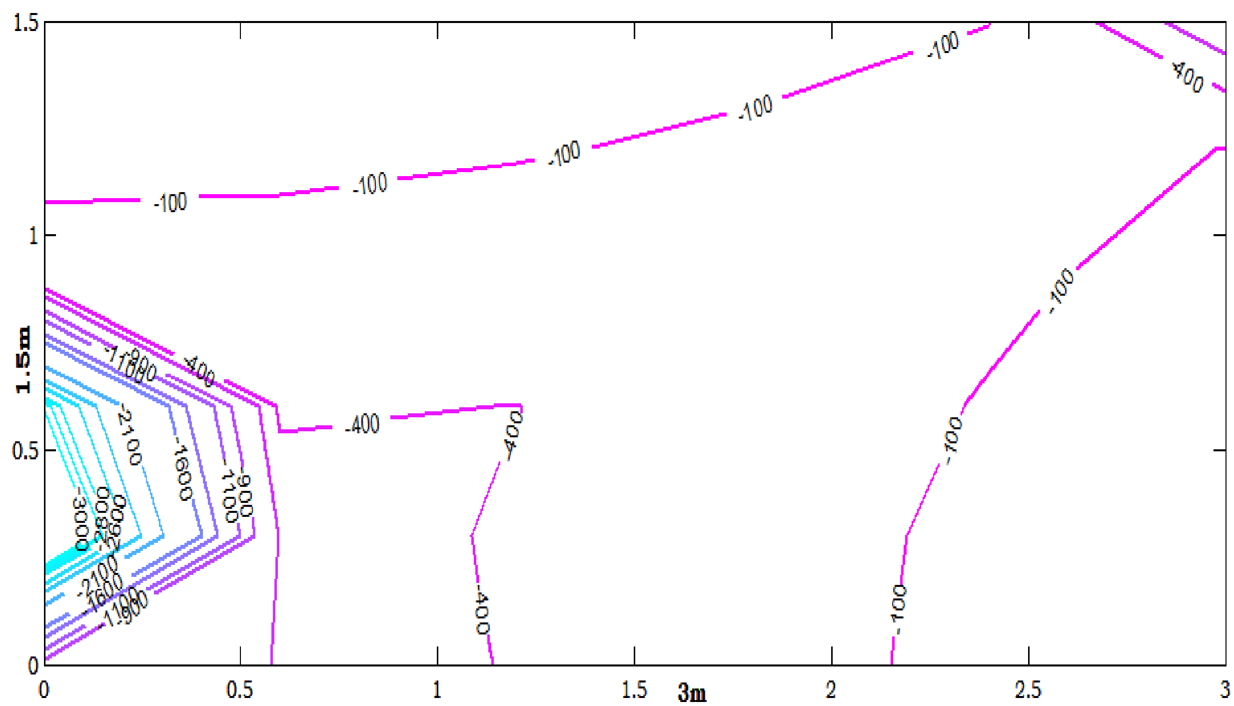


Fig. 9.35 Moment Contours for Mxy in SS plate under CPL

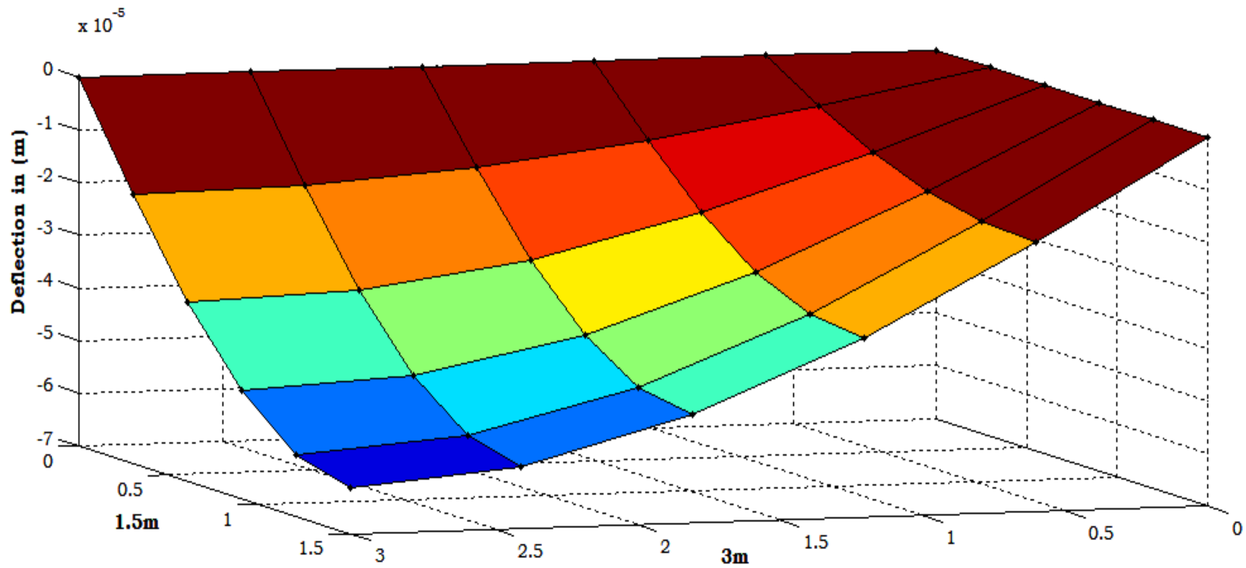


Fig. 9.36 Deformed Shape of SS Quarter Plate under UPL

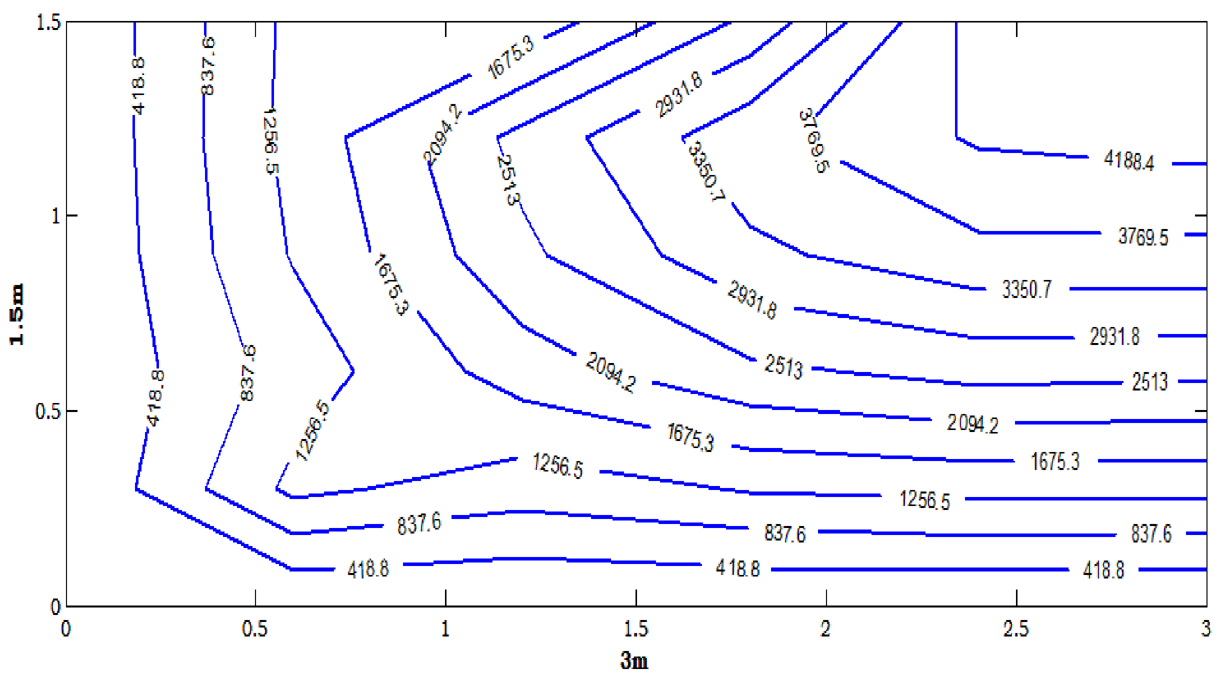


Fig. 9.37 Moment Contours for Mx in SS Plate Under UPL

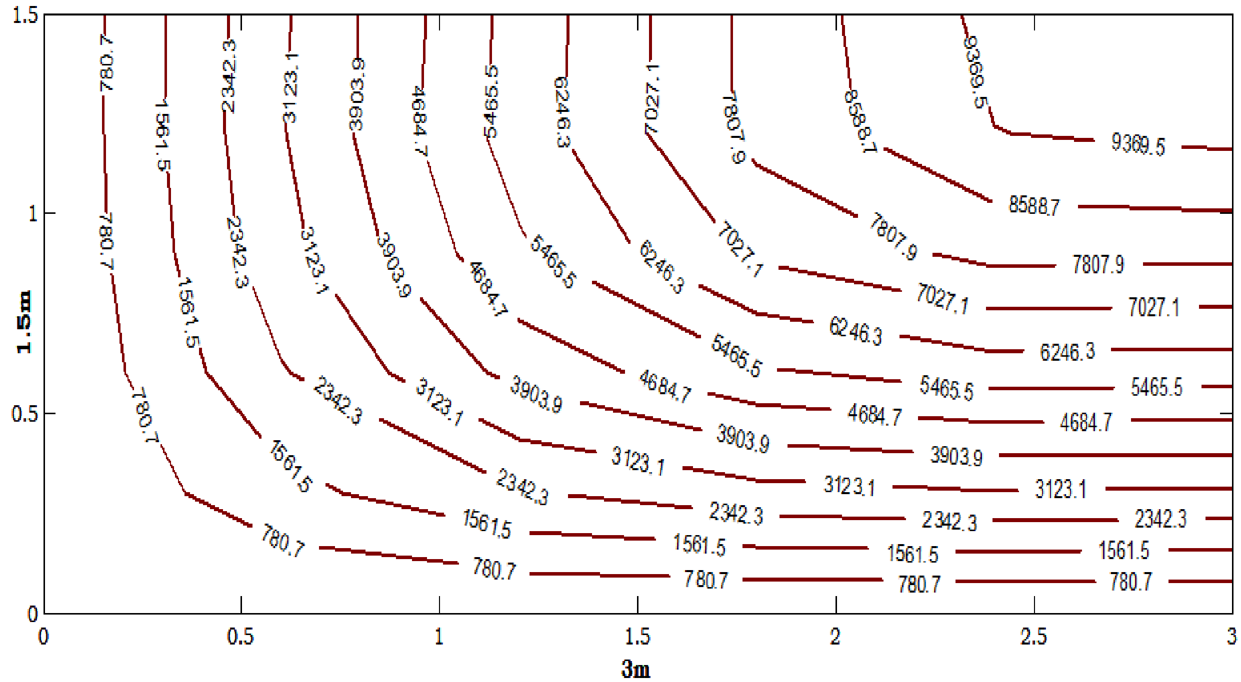


Fig 9.38 Moment Contoura for Myy for SS Plate Under UPL

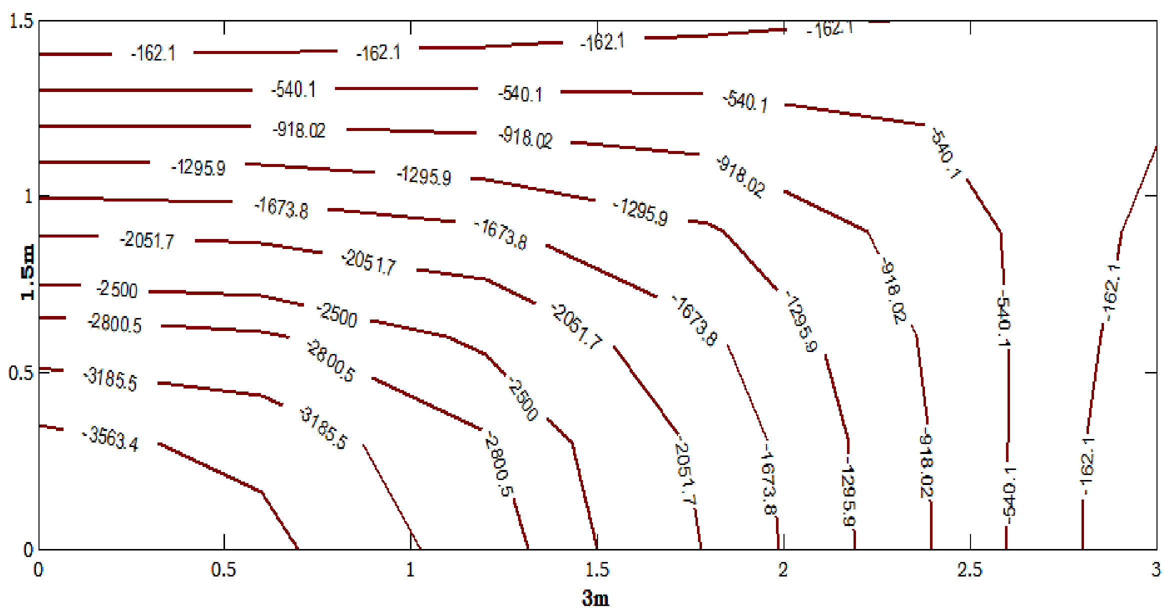


Fig 9.39 Moment Contours for Mxy for SS Plate Under UPL

Next, solutions for the clamped rectangular plate subjected to CPL and UPL type of loading are obtained by using IFM based formulation. The internal moments and deflections for UPL case at the nodal points starting from A to C of AC line (δ_{AC1}, δ_{AC2}....... δ_{AC5}) of **Fig 9.23** for the 5x5 discretization scheme are calculated and compared with the Ritz method **[100]** based on doubly cosine curve series. Table **9.12** depicts the Nodal Deflection Ratio (NDR) and Nodal Moment Ratio (NMR) for the clamped CPL case.

Table 9.12 Nodal Deflection and Nodal Moment Ratio (CPL)

Point	$\frac{IFM}{RITZ}$ (NDR) (Nodal Deflection Ratio)	$\frac{IFM}{RITZ}$ (NMR) (Nodal Moment Ratio)		
		M_{xx}	M_{yy}	M_{xy}
AC_1	1.034	1.021	1.034	1.083
AC_2	1.046	1.025	1.054	1.087
AC_3	1.076	1.045	1.063	1.089
AC_4	1.091	1.066	1.069	1.092
AC_5	1.106	1.091	1.088	1.105

The results obtained by IFM for a fixed rectangular plate problem under ULP case are included here in **Table 9.13**

Table 9.13 Results for Clamped Plate under ULP

Values at 'C'	IFM
δ(m)	9.897×10^{-06}
M_{xx} (N-m)	110.55
M_{yy} (N-m)	357.665
M_{xy} (N-m)	3.522

9.10 PLATE UNDER PATCH LOADING EXAMPLES

A simply supported square plate of size 4000 mm x 4000 mm x 200 mm is studied under a central patch loading of intensity 10 kN/m^2 over an area 1600 mm x 1600 mm as shown in **Fig 9.40.** Due to two way symmetry, only quarter of the plate is analysed by discretizing into 5 x 5 mesh. Results are presented in terms of nodal deflection ratio (NDR) for w and nodal moment ratio (NMR) for Mx, My and Mxy in **Table 9.14.** The contours of Mx and deflection profile are included here in **Figs. 9.41** and **9.42** respectively.

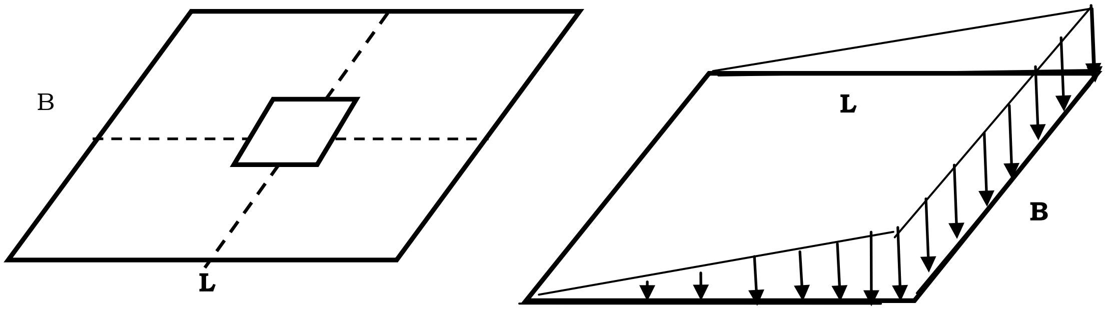


Fig. 9.38 Plate under Patch Loading

Table 9.14 Nodal Deflection and Nodal Moment Ratios

Node	IFM/EXACT **[101]**	
	NDR	NMR
AC_1	0.924	0.988
AC_2	0.980	0.998
AC_3	0.956	0.987
AC_4	0.983	0.989
AC_5	0.988	0.990

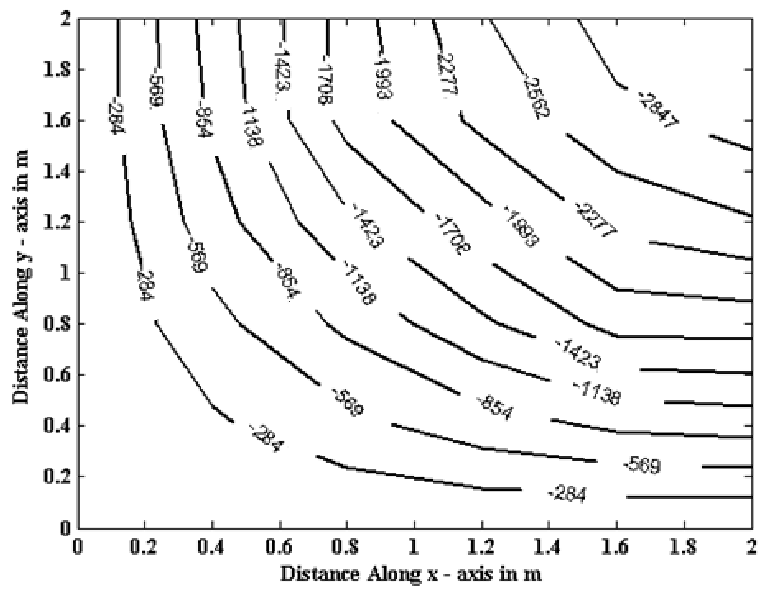


Fig. 9.41 Contours of Mx for Plate under Patch Loading

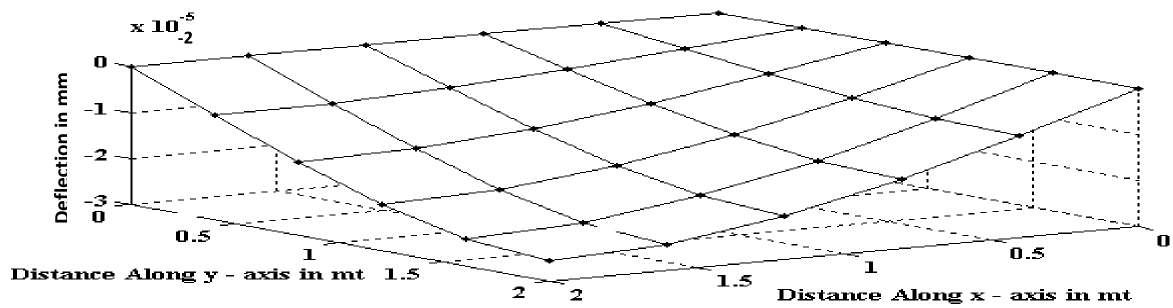


Fig. 9.42 Deflection Profile of Quarter Plate under Patch Loading

Next, a rectangular plate of size 6000 mm x 4000 mm x 200 mm simply supported along all edges is considered under patch loading of intensity of 10kN/m^2 over a size 2400 mm x 1600 mm. Results obtained in terms of NDR and NMR for w displacement and moment Mx at centre are presented in **Table 9.15**. Variation of deflection w along the centre line of the quarter rectangular and square plates is depicted in **Fig. 9.42.**

Table 9.15 Deflection and Moment Ratios For Rectangular Plate

Node	$\frac{IFM}{EXACT}$	
	NDR	NMR
AC_1	0.869	0.950
AC_2	0.828	0.985
AC_3	0.888	0.993
AC_4	0.911	0.988
AC_5	0.919	0.994

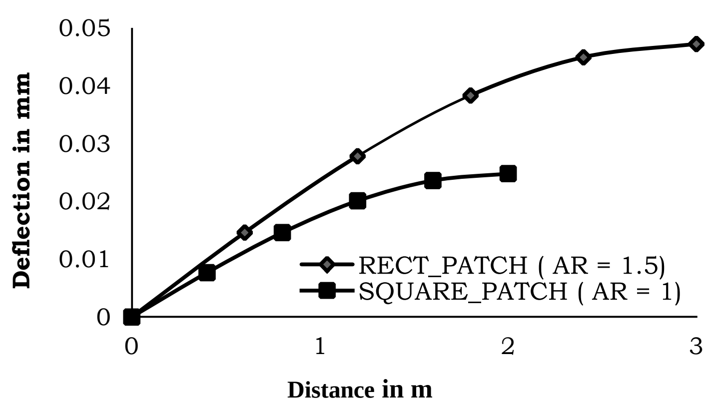


Fig. 9.43 Variation of Deflection (w) Along Central Line

9.11 PLATE UNDER HYDROSTATIC LOADING EXAMPLES

A simply supported square plate of size 4000 x 4000 x 200 mm subjected to a hydrostatic loading (uniformly varying lateral load) of intensity 10 kN/m^2 is analysed now. Due to one way symmetry, only half of the plate is analysed by discretizing it into 10 x 5 grid. Results are presented in Table 9.16 in terms of nodal deflection ratio and nodal moment ratio for lateral displacement w and moment Mx at the nodes lying on the symmetry line from the centre of the plate. Moment Mx contours are depicted in **Fig. 9.44** whereas deflection profile is shown in **Fig 9.45.**

Table 9.16 Deflection and Moment Ratios for Square Plate

Node	IFM/EXACT **[101]**	
	NDR	NMR
1	1.00	1.00
2	0.969	0.970
3	0.968	0.969
4	0.962	0.983
5	0.972	0.989
6	0.971	0.986
7	0.940	0.996
8	0.971	0.992
9	0.972	0.993
10	0.969	0.986
11	1.00	1.00

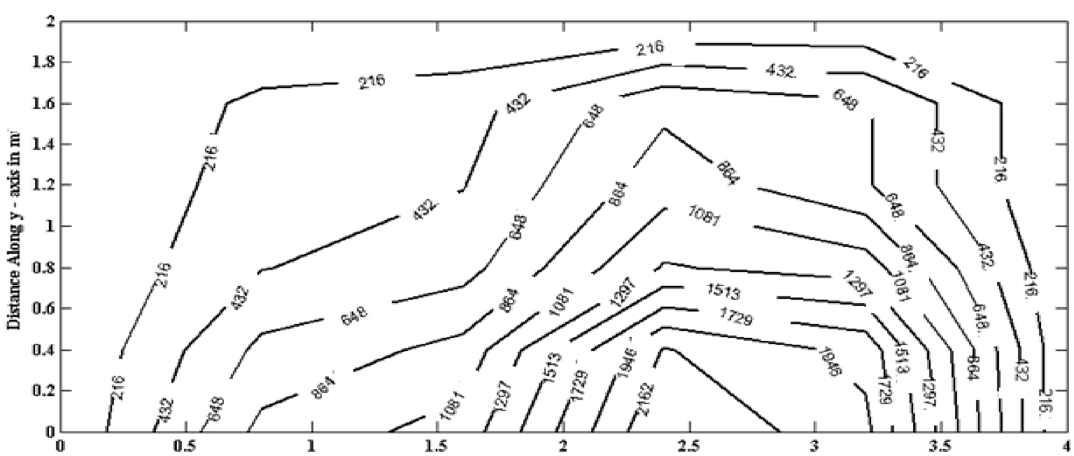


Fig. 9.44 Contours of Mx for Square Plate Under Hydrostatic Loading

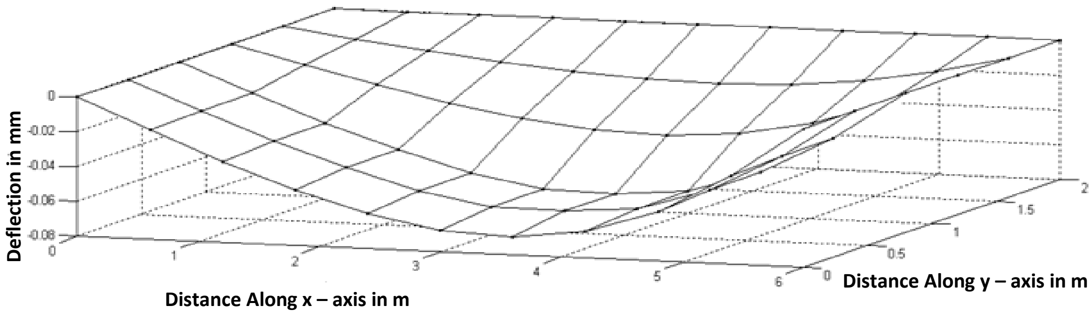


Fig. 9.45 Deflection Profile of Half Plate Under Hydrostatic Loading

Finally, a rectangular plate of size 6000 x 4000 x 200 mm is solved under a linearly varying load intensity 10kN/m^2 in the x direction. One way symmetry is used to discretize into 6000 mm x 2000 mm portion of the plate into 10 x 5 grid. Some of the results obtained using IFM are compared in terms of NDR and NMR in **Table 9.17**. The values of lateral deflection w and moment Mx reported in the table are for nodes 1 to 11 lying on the central line of the plate in the x direction. Also, variation of deflection w along the centre line is depicted in

Fig. 9.46 for square and rectangular plates having aspect ratio as 1.0 and 1.5 respectively.

Table 9.17 Nodal Deflection and for Rectangular Plate

Node	IFM/ EXACT [101]	
	NDR	NMR
1	1.00	1.00
2	0.9588	0.967
3	0.9577	0.980
4	0.957	0.968
5	0.958	0.960
6	0.958	0.976
7	0.966	0.942
8	0.961	0.967
9	0.956	0.979
10	0.955	0.969
11	1.00	0.0000

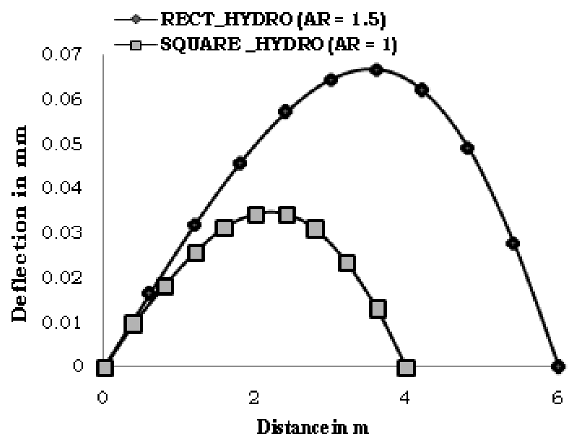


Fig. 9.46 Variation of Displacement w Along Central Line

9.12 DISCUSSION OF RESULTS

- For the deep cantilever beam problems results obtained for nodal deflections and maximum stresses are found to converge towards the exact solution with finer discretization. The discretization with 5 elements is found to give exact solution thus no further discretization is required for similar problems.
- Pure bending example of beam is solved using symmetry and antisymmetry in which only upper right quadrant is considered. Results obtained using IFM are compared with those obtained using the exact method and finite element method. The IFM based solution is found more closer to the exact solution compared to the FEM solution.
- A propped deep cantilever beam subjected to horizontal point load is solved using two different IFM based Elements i.e. RECT_5F_8D and TRI_3F_6D. Stress variation seems to be appropriate but could not be compared because of the non availability of the solution. Results for nodal displacements using TRI_3F_6D and RECT_5F_8D are found in good agreement with those available in the literature based on finite element method.
- IFM is also successfully applied to a curved beam problem using CURV_5F_8D element having five internal forces and eight nodal

displacement degrees of freedom. With two element discretization results for nodal displacements at A and Stresses at C are compared with the results of theory of elasticity; a good agreement is indicated.

- Various types of plate bending problems are attempted using IFM based RECT_9F_12D. Simply supported and clamped, square and rectangular plate bending problems are attempted under Central Point Load (CPL) and Uniform Lateral Pressure (ULP). By using dual symmetry only lower left quadrant is discretized. Convergence study indicated 5 x 5 discretization result quite close to the exact value
- At junction 'I" of the elements 11, 12, 13, and 14 (Fig. 9.21) the values of moments M_x and M_y are found same from all the elements in magnitude and nature. These moments are calculated by taking respective values of internal unknowns with their applicable nodal distance from the centre of the elements.
- Node and Element numbering is done in anticlockwise direction starting from left bottom most point and reaching to the centre of the domain with 25 number for 5 x 5 discretization. The displacement numbering is also done in the same manner. Thus, maximum possible numerical values are oriented along diagonal direction of the global equilibrium matrix of size 75 x 225. This helps in developing relevant global compatibility conditions which depends upon proper input value of [B] matrix in "mtechexamplemod(B)" file of matlab. The other numbering patterns for elements and nodal ddofs may produce more sparsity in global equilibrium matrix which sometimes may lead to singular matrix and thus one cannot find the solution in such cases.
- Use of 2 x 2 discretization gives higher values of deflections compared to exact values for square plate under point load and uniformly distributed loading cases. While the moments for the same shows an increasing nature from lower values to higher with coarser to finer discretization patterns. With 5 x 5 discretization, for the plate under both types of loading, the results are found quite close to the exact

solution. Results for clamped rectangular plate subjected to uniform lateral pressure are also found in good agreement with the energy based Ritz approach results.

- Special cases of loading i.e. patch loading and hydrostatic loading could be easily tackled by IFM for square and rectangular plate cases. Two dimensional moment contours and three dimensional deformed shape included for different example gives an immediate glimpse of the variation of moments and deflection.

CHAPTER 10

DYNAMIC ANALYSIS OF RECTANGULAR PLATE PROBLEMS

10.1 COMPUTER IMPLEMENTATION

Software development of plate bending problems is mainly collaborative work between Visual Basic 6 (Programming tool) and Matlab 7.4 (Mathematical tool). Forms are developed using an advanced GUI based facility of VB to facilitate interactive input. First of all the plate geometry, square or rectangular shape, is selected by using the option button. Once selection is done of shape, by using text box facility other geometrical parameters and material properties are supplied and accepted. After that, control of the program is transferred to the next form to select the types of boundary conditions are through check box facility. Next, discretization pattern is to be selected by user thru next form depicted on screen. Once it is done, discretized continuum is visible on the screen for left-bottom quadrant of the plate or bottom half of the plate depending upon the type of symmetry of the problem. Clicking of the accept button transfers the control to the next form where IFM based RECT_9F_12D is shown with its degrees of freedom. Next, the following operations are carried out:

1. By using Hermitian shape functions, strain linking matrix [Z] of size 3 x12 is developed for rectangular element of size 2a x 2b.
2. Stress linking matrix [Y] is directly written in terms of x and y which is of size 3 x 9, where the rows of the matrix correspond to number of moment components and columns of the matrix correspond to the number of unknown internal moments per element.
3. The elemental equilibrium matrix [Be] of size 12 x 9 is worked out by direct integration of product of $[z]^T$ and [Y] using “int (function, lower imit, upper limit, with respect to x or y)” command of Matlab.

4. The elemental flexibility matrix [Ge] of size 9 x 9 for the rectangular element is worked out. The calculation of matrix requires multiplication of $[Y]^T$, material matrix [D] and [Y] and the numerical integration with respect to x and y.
5. The assembly procedure to get the global equilibrium and flexibility matrices is carried out. It gives the global equilibrium matrix [B] of size 12 x 36 and global flexibility matrix [G] of size 36 x 36 for a 2 x 2 discretization scheme. Once these matrices are assembled, they are transferred into notepad files using .txt extension, which are directly called by matlab command by names RECT_9F_12D_B.txt and RECT_9F_12D_G.TXT through proper path.
6. The lumped mass matrix [M] which is of size 12 x12 is directly read from a .txt file named as RECT_9F_12D_M.txt.

Once all the above steps are performed the connection between VB and Matlab is established using COM Automation server facility where Matlab acts as a mathematical server to continue the remaining procedure required for the calculation of unknowns.

10.2 ILLUSTRATIVE EXAMPLE WITH SOLUTION STEPS

Frequency analysis of a simply supported plate having dimension of 2m x 2m x 0.01m is carried out by considering dual symmetry. Bottom left quadrant is discretized into 2 x 2 grid and lumped mass approach is considered. Using E and ν as 2.01×10^8 kN/m^2 and 0.3 respectively, the results are obtained in terms of natural frequencies, normalized moments, and nodal displacements as follows.

Step1: Using MDI feature of VB 6 different basic forms are attached with a menu editor, where user can select dynamic as a sub-menu from the main menu of Plate analysis as depicted in **Fig. 10.1.**

Fig. 10.1 Main Menu with option for Dynamic Analysis

Step 2: Once dynamic analysis is selected, the control is transferred to the next form where shape of the plate is selected and dimensions and material properties are entered as depicted in **Fig. 10.2**.

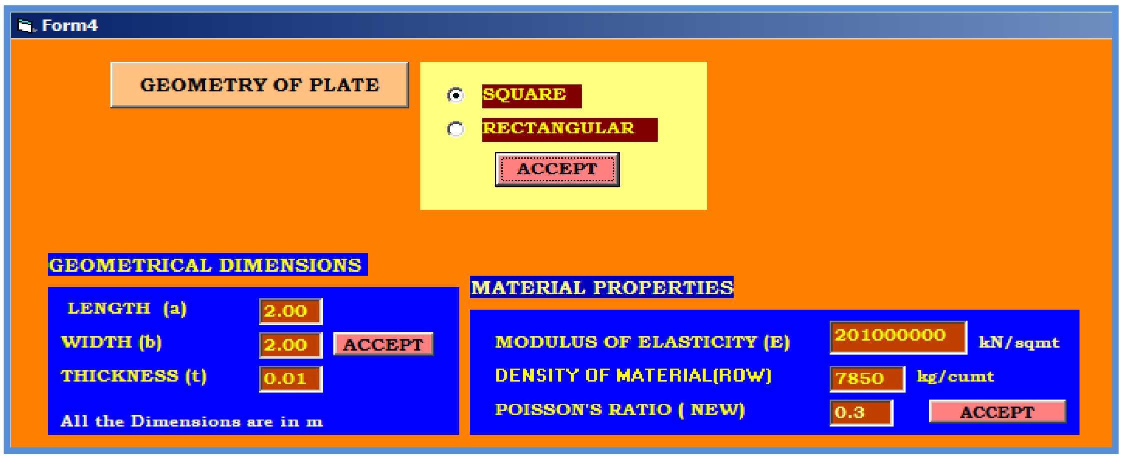


Fig. 10.2 Form for Geometry and Material Properties

Step 3: The boundary condition of plate is selected using check box facility in the next form in which for each side of the plate a provision is made to specify separate condition as shown in **Fig. 10.3**.

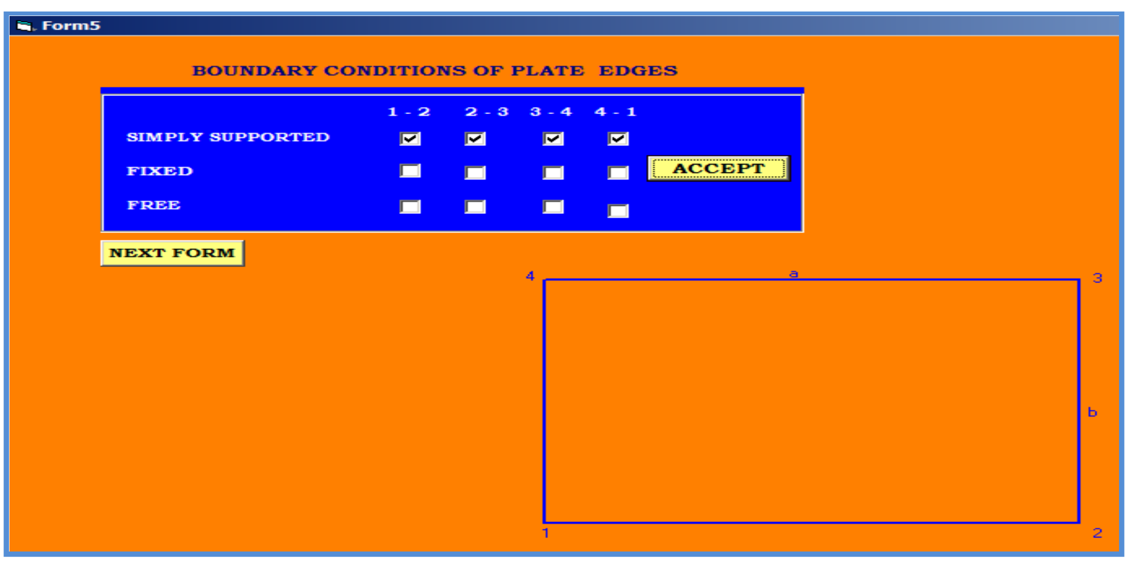


Fig.10.3 Form for Boundary Conditions

Step 4: Depending upon the specified boundary conditions, single or double symmetry are auto selected with appropriate boundary conditions. The discretization pattern is then selected from the available options as shown in **Fig. 10.4.**

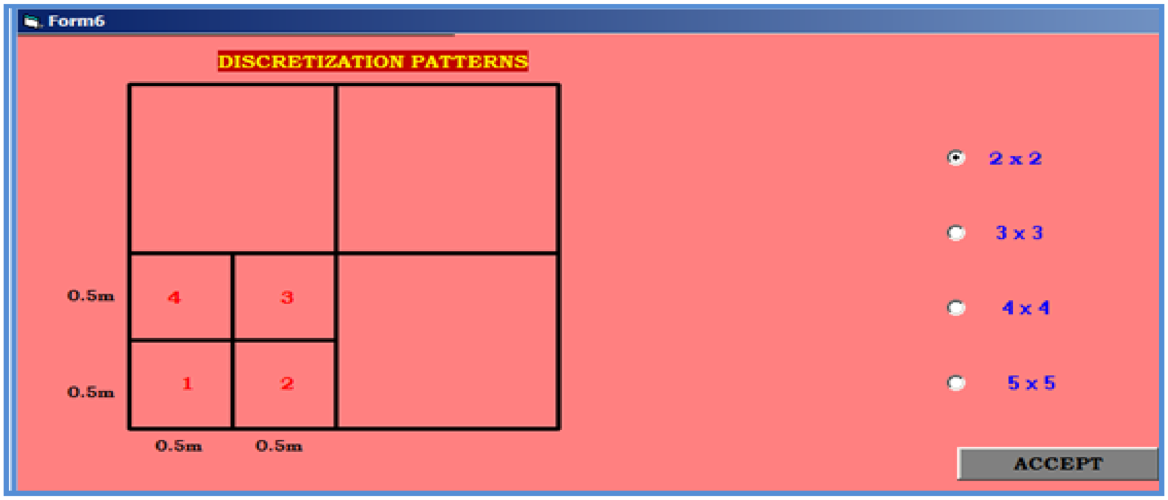


Fig. 10.4 Discretization Scheme

Step 5: Clicking button of Basic element in the next form a rectangular plate bending element having total nine internal unknowns and twelve displacement degrees of freedom is drawn as depicted in **Fig. 10.5.** Connection between Visual basic and Matlab is then established by using COM automation server as explained earlier in Chapter 6 and a Com based matlab command window is depicted on screen as shown in **Fig. 10.6.**

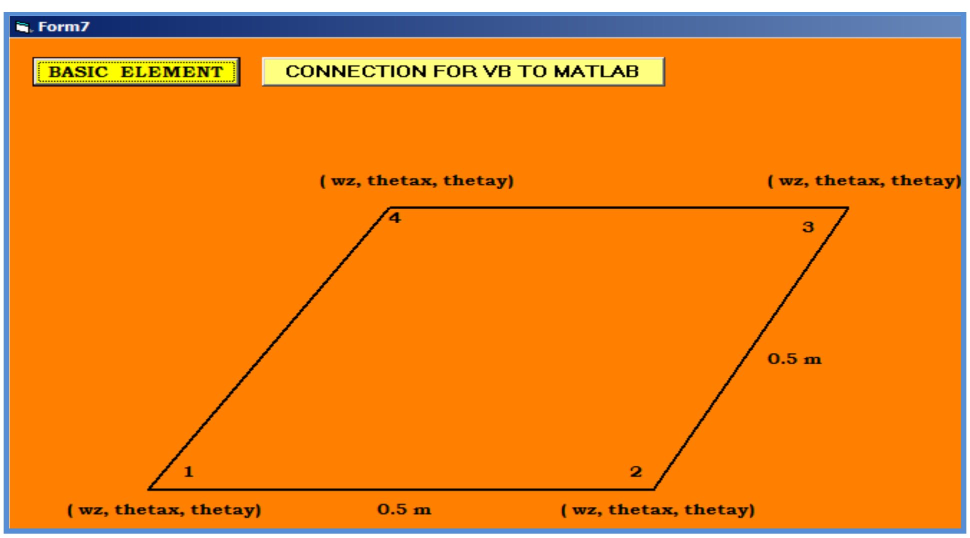


Fig. 10.5 Rectangular Plate Element (RECT_9F_12D)

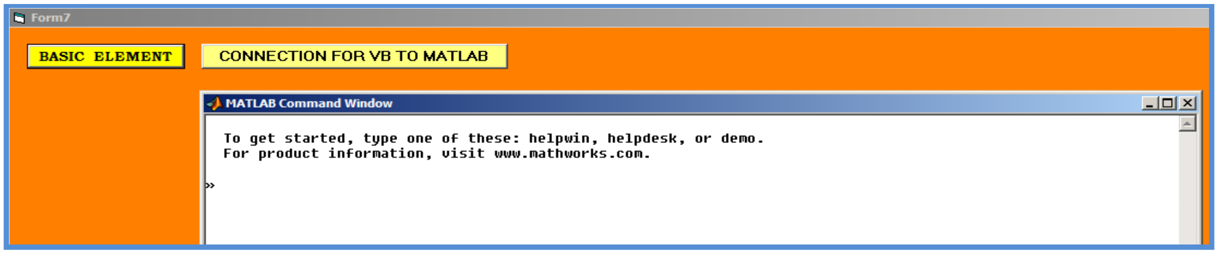


Fig. 10.6 Com Based Matlab Command Window

Step 6: Using the matlab command editor the necessary mathematical operations are carried out. The assembled global equilibrium matrix [B] is depicted in **Fig. 10.7**.

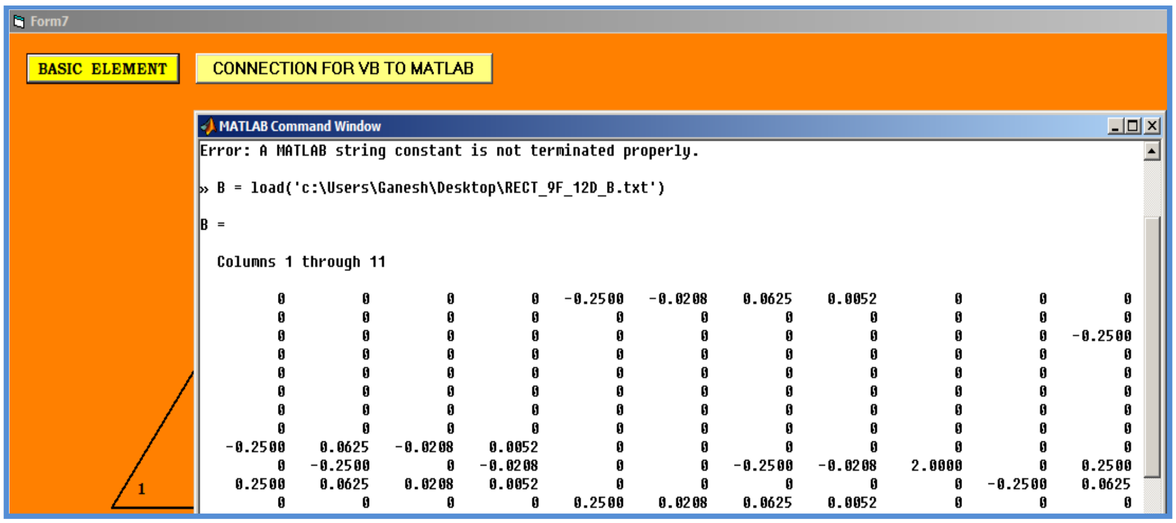


Fig. 10.7 [B] Matrix in Matlab Command Window

Step 7: The global flexibility matrix is also developed with Matlab editor using sub-matrix function, where a basic flexibility matrix for the given element [Ge] is directly called and using concatenation of zero matrix of size (9 x 9) at position other than the diagonal a complete global matrix is formulated. The element matrix is shown in **Fig. 10.8** whereas the global matrix is depicted in **Fig. 10.9.**

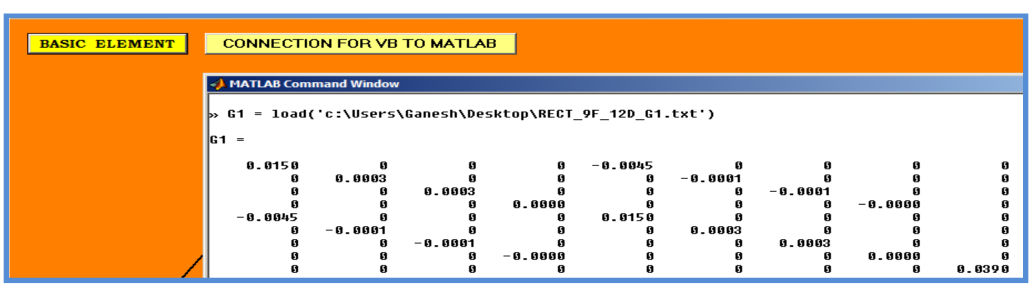
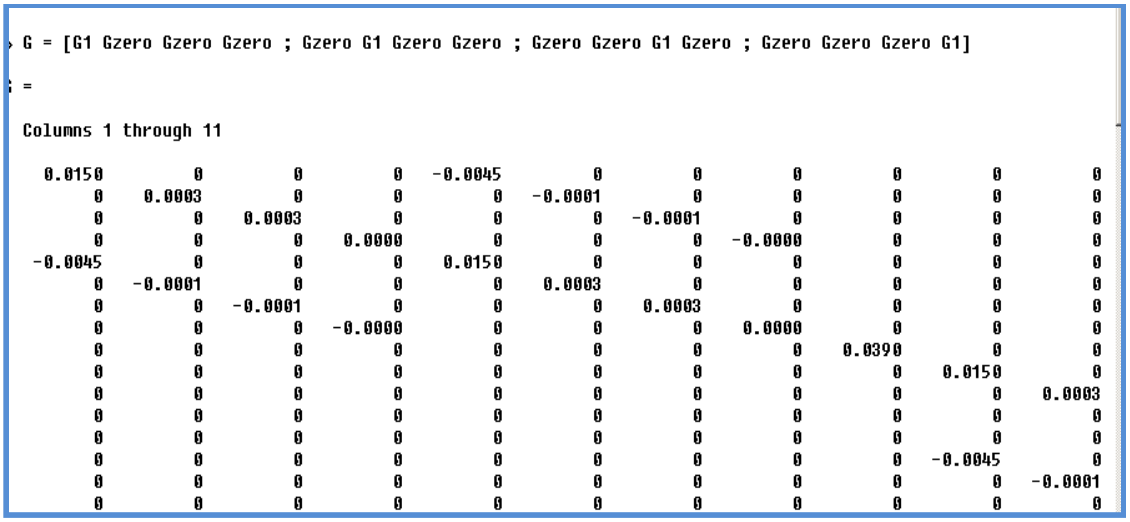


Fig. 10.8 Elemental Flexibility Matrix [Ge_1]



```
G = [G1 Gzero Gzero Gzero ; Gzero G1 Gzero Gzero ; Gzero Gzero G1 Gzero ; Gzero Gzero Gzero G1]
```

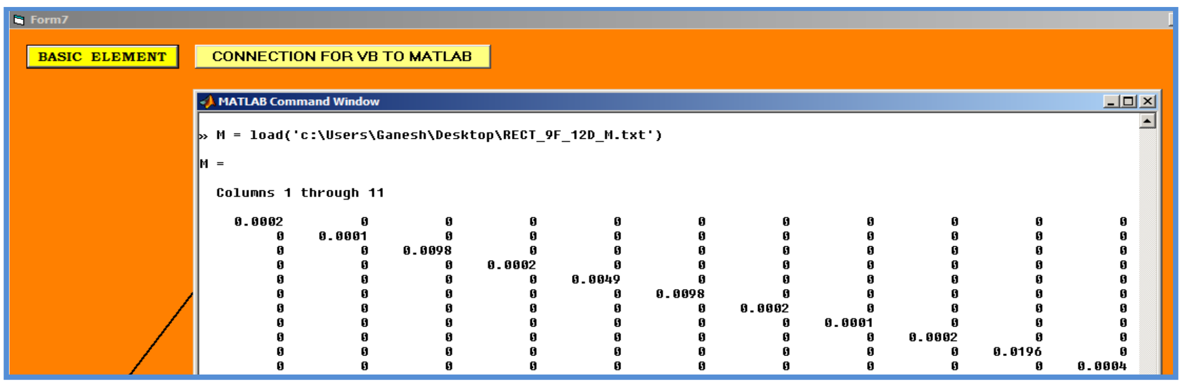
G =

Columns 1 through 11

0.0150	0	0	0	-0.0045	0	0	0	0	0	0
0	0.0003	0	0	0	-0.0001	0	0	0	0	0
0	0	0.0003	0	0	0	-0.0001	0	0	0	0
0	0	0	0.0000	0	0	0	-0.0000	0	0	0
-0.0045	0	0	0	0.0150	0	0	0	0	0	0
0	-0.0001	0	0	0	0.0003	0	0	0	0	0
0	0	-0.0001	0	0	0	0.0003	0	0	0	0
0	0	0	-0.0000	0	0	0	0.0000	0	0	0
0	0	0	0	0	0	0	0	0.0390	0	0
0	0	0	0	0	0	0	0	0	0.0150	0
0	0	0	0	0	0	0	0	0	0	0.0003
0	0	0	0	0	0	0	0	0	0	0
0	0	0	0	0	0	0	0	0	0	0
0	0	0	0	0	0	0	0	0	-0.0045	0
0	0	0	0	0	0	0	0	0	0	-0.0001
0	0	0	0	0	0	0	0	0	0	0

Fig.10.9 Global Flexibility Matrix [G]

Step 8: The global lumped mass matrix is lumped diagonally which presently include components of rotary inertia **[100]** as shown in **Fig. 10.10**. The terms of mass matrix [M] are worked out by multiplying mass per unit area by tributary area of each element under consideration.




```
>> M = load('c:\Users\Ganesh\Desktop\RECT_9F_12D_M.txt')
```

M =

Columns 1 through 11

0.0002	0	0	0	0	0	0	0	0	0	0
0	0.0001	0	0	0	0	0	0	0	0	0
0	0	0.0098	0	0	0	0	0	0	0	0
0	0	0	0.0002	0	0	0	0	0	0	0
0	0	0	0	0.0049	0	0	0	0	0	0
0	0	0	0	0	0.0098	0	0	0	0	0
0	0	0	0	0	0	0.0002	0	0	0	0
0	0	0	0	0	0	0	0.0001	0	0	0
0	0	0	0	0	0	0	0	0.0002	0	0
0	0	0	0	0	0	0	0	0	0.0196	0
0	0	0	0	0	0	0	0	0	0	0.0004

Fig. 10.10 Global Lumped Mass Matrix [G]

Step 9: Next, the change directory operation is performed and matlab server is connected to folder named as "CCPROG", which consists of a program for auto-generation of compatibility conditions. Thus by using "z= mtechexamplemod(B)", the complete [B] matrix is read by the .m file of matlab. The number of the compatibility conditions is specified to make the global equilibrium matrix a square matrix which for the present problem of is of size 24 x 24 , (**Fig. 10.11).**

```
>>
>> z = mtechexamplemod(B)

codeindB =

     1     2     5     7     9    14    16    18    20    21    29    36


codedepB =

  Columns 1 through 23

     3     4     6     8    10    11    12    13    15    17    19    22    23    24    25    26    27    28    30    31    32    33    34

  Column 24

    35

Enter the no. of solutions wanted:24
```

Fig. 10.11 Development of CC using .m File ("mtechexamplemod(B))"

Step 10: Once the number of compatibility conditions is specified as 24 to matlab editor, it depicts a complete compatibility matrix (z.cMatrix) of size 24 x 36 on screen as shown in **Fig. 10.12.**

```
  Columns 1 through 11

    1.5833    4.6667    7.0000    2.0000    0.3854    8.0000    2.7917    9.0000    6.9323    5.0000    8.0000
  -18.1771  -73.2083    8.0000    6.0000    2.3750    9.0000   12.5833    7.0000    3.4010    6.0000    8.0000
  -15.3958  -61.4167    1.0000    6.0000    2.0208    6.0000    4.5000    7.0000    2.0417    9.0000    9.0000
   -3.0729  -14.7917    2.0000    2.0000    1.6979    2.0000    3.4583    2.0000    1.0208    1.0000    6.0000
    1.3854    4.5417    1.0000    2.0000    4.6562    2.0000   14.2917    6.0000   10.3438    6.0000    1.0000
  -21.8854  -87.7083    5.0000    6.0000    1.8021    1.0000    6.5417    8.0000    2.5313    6.0000    8.0000
   -5.8646  -24.1250    2.0000    4.0000    4.6979    1.0000   14.8750    5.0000    4.9792    4.0000    2.0000
   -9.5938  -36.6250    4.0000    1.0000    5.3125    9.0000   18.5833    2.0000   -0.5052    1.0000    3.0000
  -15.4271  -63.4583    4.0000    3.0000    0.0104    7.0000   -1.2083    1.0000   -0.5937    1.0000    7.0000
  -18.2396  -74.7917    9.0000    8.0000    3.0729    7.0000   10.2083    1.0000   -0.6875    1.0000    1.0000
   -9.3646  -37.9583    4.0000    4.0000    2.1562    7.0000    6.1250    6.0000   -1.6250    8.0000    4.0000
  -14.6563  -60.4583    2.0000    8.0000    3.2500    3.0000   11.8333    6.0000   -1.7552    9.0000    4.0000
    2.3437    8.4583    1.0000    3.0000    0.9688    7.0000    3.9583    7.0000    4.1354    2.0000    2.0000
  -12.6146  -49.2083    9.0000    7.0000    2.3750    6.0000    7.8333    2.0000    6.1719    2.0000    1.0000
   -8.1667  -33.7500    2.0000    7.0000    0.9271    8.0000    2.7083    4.0000    1.6406    7.0000    3.0000
  -23.1979  -94.5417    9.0000    5.0000    4.6979    3.0000   15.3750    7.0000   -0.5625    7.0000    7.0000
  -25.7917 -103.2500    8.0000    7.0000    3.0313    2.0000   11.5417    5.0000   -2.0990    3.0000    5.0000
  -14.0000  -59.7500    7.0000    7.0000    3.3958    6.0000   11.2500    4.0000    3.0313    4.0000    8.0000
```

Fig. 10.12 The Compatibility Matrix

Step 11: Once "z.cMatrix" is ready its null property is readily checked in the module named as 'z.cTransposeB', which actually represents product of [C] and $[B]^T$.(**Fig. 10.13).**

```
>> z.cTransposeB

ans =

  1.0e-014 *

  Columns 1 through 11

    0.0038   -0.0038   -0.0444   -0.0010         0   -0.0888   -0.0687   -0.0298    0.0520   -0.0888    0.1464
    0.0021    0.0035         0   -0.0139         0         0   -0.2130    0.1089    0.0243    0.3553   -0.0534
    0.0056    0.0056   -0.1776    0.0203         0    0.1776   -0.1287    0.1176    0.0378   -0.3553   -0.1377
   -0.0052    0.0031         0    0.0035         0         0   -0.0555    0.0222         0    0.3553    0.0111
    0.0010    0.0101         0    0.0173         0   -0.1776   -0.1252   -0.0322   -0.0122    0.3553    0.1918
   -0.0203   -0.0130   -0.1776   -0.0160    0.1776   -0.5329   -0.1731    0.1655    0.1454    0.7105   -0.3598
```

Fig. 10.13 [C] Matrix and Null Property Check for [C] and $[B]^T$

Step 12: The multiplication of [C] matrix with global flexibility matrix [G] gives a global compatibility matrix with the coefficients which converts the compatibility condition into forces. It is concatenated into [B] matrix from bottom side where no normalization is needed. The numerical coefficients of z.cMatrix and [G] matrices are modified after multiplication and there is no need of normalization. The CCmatrix and Smatrix are depicted in **Figs. 10.14** and **10.15** respectively.

```
>> CCmatrix = z.cMatrix*G

CCmatrix =

  Columns 1 through 11

    0.0220    0.0007    0.0019   -0.0000   -0.0013    0.0021    0.0002    0.0001    0.2704    0.0600    0.0023
   -0.2833   -0.0237    0.0013    0.0000    0.1174    0.0097    0.0032    0.0000    0.1326    0.0828    0.0024
   -0.2400   -0.0198   -0.0001    0.0000    0.0996    0.0076    0.0013    0.0000    0.0796    0.1240    0.0020
   -0.0537   -0.0048    0.0003    0.0000    0.0393    0.0020    0.0009    0.0000    0.0398    0.0171    0.0012
   -0.0002    0.0012   -0.0010    0.0000    0.0636    0.0002    0.0044    0.0000    0.4034    0.0756   -0.0003
   -0.3364   -0.0275    0.0009    0.0000    0.1255    0.0085    0.0016    0.0000    0.0987    0.0750    0.0024
   -0.1091   -0.0076   -0.0008    0.0000    0.0969    0.0026    0.0045    0.0000    0.1942    0.0490   -0.0000
   -0.1678   -0.0123   -0.0005    0.0000    0.1229    0.0062    0.0054    0.0000   -0.0197    0.0146    0.0001
   -0.2315   -0.0205    0.0014    0.0000    0.0696    0.0081   -0.0008    0.0000   -0.0232    0.0030    0.0015
   -0.2874   -0.0240    0.0019    0.0001    0.1282    0.0092    0.0023   -0.0000   -0.0268    0.0140   -0.0003
```

Fig. 10.14 Global Compatibility Matrix [CCmatrix]

```
>> Smatrix =[B;CCmatrix]
Smatrix =
  Columns 1 through 11
         0         0         0         0   -0.2500   -0.0208    0.0625    0.0052         0         0         0
         0         0         0         0         0         0         0         0         0         0         0
         0         0         0         0         0         0         0         0         0         0   -0.2500
         0         0         0         0         0         0         0         0         0         0         0
         0         0         0         0         0         0         0         0         0         0         0
         0         0         0         0         0         0         0         0         0         0         0
         0         0         0         0         0         0         0         0         0         0         0
         0         0         0         0         0         0         0         0         0         0         0
   -0.2500    0.0625   -0.0208    0.0052         0         0         0         0         0         0         0
         0   -0.2500         0   -0.0208         0         0   -0.2500   -0.0208    2.0000         0    0.2500
    0.2500    0.0625    0.0208    0.0052         0         0         0         0         0   -0.2500    0.0625
         0         0         0         0    0.2500    0.0208    0.0625    0.0052         0         0         0
    0.0220    0.0007    0.0019   -0.0000   -0.0013    0.0021    0.0002    0.0001    0.2704    0.0600    0.0023
   -0.2833   -0.0237    0.0013    0.0000    0.1174    0.0097    0.0032    0.0000    0.1326    0.0828    0.0024
   -0.2400   -0.0198   -0.0001    0.0000    0.0996    0.0076    0.0013    0.0000    0.0796    0.1240    0.0020
   -0.0537   -0.0048    0.0003    0.0000    0.0393    0.0020    0.0009    0.0000    0.0398    0.0171    0.0012
   -0.0002    0.0012   -0.0010    0.0000    0.0636    0.0002    0.0044    0.0000    0.4034    0.0756   -0.0003
   -0.3364   -0.0275    0.0009    0.0000    0.1255    0.0085    0.0016    0.0000    0.0987    0.0750    0.0024
   -0.1091   -0.0076   -0.0008    0.0000    0.0969    0.0026    0.0045    0.0000    0.1942    0.0490   -0.0000
```

Fig 10.15 Global Equilibrium Matrix [Smatrix]

Step 13: The [Sinv] matrix is worked out by inverting the [Smatrix]. It is depicted in **Fig. 10.16**. The transpose the [Sinv] matrix is named as [Jmatrix]**.**

```
>> Sinv = inv(Smatrix)
Sinv =
  1.0e+004 *
  Columns 1 through 11
   -0.0000    0.0000   -0.0000   -0.0000   -0.0000   -0.0000   -0.0000    0.0000   -0.0001   -0.0000    0.0000
   -0.0002    0.0000   -0.0000   -0.0001   -0.0000   -0.0000   -0.0001   -0.0003    0.0003   -0.0000    0.0002
    0.0000   -0.0001   -0.0000   -0.0001   -0.0000   -0.0000   -0.0001    0.0001   -0.0003   -0.0000    0.0002
    0.0002   -0.0003   -0.0000   -0.0002   -0.0001   -0.0001   -0.0004   -0.0011    0.0014   -0.0001    0.0006
   -0.0001    0.0000   -0.0000   -0.0000   -0.0000   -0.0000   -0.0000    0.0000   -0.0000   -0.0000    0.0000
   -0.0003    0.0001   -0.0000   -0.0001   -0.0000   -0.0000   -0.0001   -0.0001    0.0000   -0.0000    0.0000
```

Fig. 10.16 Inversion of [Smatrix]

Step 14: The global mass matrix [M] is developed as per free displacement degrees of freedom while for multiplying the same matrix with [Jmatrix] and [G], two null sub-matrices are concatenated towards right and from bottom side. Thus, the complete compatible Mass matrix [M_1] becomes of size 36 x 36. A part of it is depicted in **Fig. 10.17.**

```
M11 =

  Columns 1 through 14

    0.0002         0         0         0         0         0         0         0         0         0         0         0
         0    0.0001         0         0         0         0         0         0         0         0         0         0
         0         0    0.0098         0         0         0         0         0         0         0         0         0
         0         0         0    0.0002         0         0         0         0         0         0         0         0
         0         0         0         0    0.0049         0         0         0         0         0         0         0
         0         0         0         0         0    0.0098         0         0         0         0         0         0
         0         0         0         0         0         0    0.0002         0         0         0         0         0
         0         0         0         0         0         0         0    0.0001         0         0         0         0
         0         0         0         0         0         0         0         0    0.0002         0         0         0
         0         0         0         0         0         0         0         0         0    0.0196         0         0
         0         0         0         0         0         0         0         0         0         0    0.0004         0
         0         0         0         0         0         0         0         0         0         0         0    0.0004
```

Fig 10.17 Mass Matrix

Step 15: The product of global modified mass matrix [M_1], [Jmatrix] and Global flexibility matrix [G] is named as MJG. it is the denominator part of the eigen analysis (**Fig. 10.18)**.

```
MATLAB Command Window
>> MJG = M11*Jmatrix*G

MJG =

  1.0e-003 *

  Columns 1 through 14

   -0.0004   -0.0000   -0.0000   -0.0000   -0.0023   -0.0002    0.0002    0.0000    0.0027   -0.0004    0.0000
    0.0000    0.0000    0.0000    0.0000    0.0002    0.0000   -0.0001   -0.0000    0.0016   -0.0004   -0.0000
   -0.0006   -0.0000   -0.0000   -0.0000   -0.0066   -0.0006   -0.0006   -0.0000    0.0670   -0.0164   -0.0013
   -0.0002   -0.0000   -0.0000   -0.0000   -0.0006   -0.0000   -0.0000   -0.0000    0.0021   -0.0004    0.0000
   -0.0029   -0.0002   -0.0002   -0.0000   -0.0029   -0.0002   -0.0002   -0.0000    0.0455   -0.0095   -0.0003
   -0.0066   -0.0006   -0.0006   -0.0000   -0.0006   -0.0000   -0.0000   -0.0000    0.0670   -0.0123    0.0001
   -0.0006   -0.0000   -0.0000   -0.0000   -0.0002   -0.0000   -0.0000   -0.0000    0.0021   -0.0001   -0.0001
```

Fig. 10.18 [MJG] matrix

Step 16: The frequency analysis is carried out by writting the eigen form at command prompt of Matlab editor as "[Fmatrix, Freq11] = eig(Smatrix, MJG)", in which Freq11 represents a diagonal matrix of size (36 x 36) having 12 possible values corresponding to each displacement degrees of freedom as depicted in **Fig 10.19.** After having square root [Freq11] matrix, a diagonal matrix of FREQUNCY is developed which is of the same size, while [Fmatrix] is a square matrix of size 36 x 36 (Fig. 10.29), in which each column represents values of internal unknowns {F_1, F_2, F_3......F_{36}}, obtained after substituting value of each frequency in a IFM based eigen equation.**(Fig.10.21)**

```
MATLAB Command Window

Freq11 =

  1.0e+005 *

  Columns 1 through 11

    4.8305         0         0         0         0         0         0         0         0         0         0
         0    4.3777         0         0         0         0         0         0         0         0         0
         0         0    2.4890         0         0         0         0         0         0         0         0
         0         0         0    0.0464         0         0         0         0         0         0         0
         0         0         0         0    0.9985         0         0         0         0         0         0
         0         0         0         0         0    0.6984         0         0         0         0         0
         0         0         0         0         0         0    2.9868         0         0         0         0
         0         0         0         0         0         0         0    2.9868         0         0         0
         0         0         0         0         0         0         0         0    2.0908         0         0
         0         0         0         0         0         0         0         0         0    2.0908         0
         0         0         0         0         0         0         0         0         0         0    0.7467
         0         0         0         0         0         0         0         0         0         0         0
```

Fig. 10.19 Freq11matrix

```
MATLAB Command Window
    0.0408    0.0464   -0.1394   -0.0396    0.0138    0.0727    0.0189   -0.0377
    0.2672    0.0681   -0.6456   -0.4200    0.1245    0.5282   -0.2791   -0.3295

>> FREQUENCY = sqrt(Freq11)

FREQUENCY =

  Columns 1 through 11

  695.0195         0         0         0         0         0         0         0         0         0         0
         0  661.6419         0         0         0         0         0         0         0         0         0
         0         0  498.8975         0         0         0         0         0         0         0         0
         0         0         0   68.1478         0         0         0         0         0         0         0
         0         0         0         0  315.9950         0         0         0         0         0         0
         0         0         0         0         0  264.2690         0         0         0         0         0
         0         0         0         0         0         0  546.5151         0         0         0         0
         0         0         0         0         0         0         0  546.5151         0         0         0
         0         0         0         0         0         0         0         0  457.2478         0         0
         0         0         0         0         0         0         0         0         0  457.2478         0
         0         0         0         0         0         0         0         0         0         0  273.2574
```

Fig. 10.20 FREQUENCY matrix

```
>> Fmatrix

Fmatrix =

  Columns 1 through 11

    0.0032   -0.0036   -0.0039    0.0777   -0.0208    0.1371   -0.0055    0.0127   -0.0014    0.0038    0.0896
   -0.1893    0.3633    0.0294    0.2641    0.0970    0.1015   -0.0274   -0.0750   -0.1086   -0.0800    0.1226
   -0.0478   -0.1276   -0.0020    0.2969   -0.1374    0.4142   -0.0235    0.0130   -0.0119    0.0530    0.2877
   -1.0000    1.0000    0.1716    1.0000    0.1716   -0.1303   -0.1163   -0.4506   -0.4604   -0.1694    0.2073
    0.0032    0.0036   -0.0039    0.0777    0.0208    0.1371    0.0064    0.0077   -0.0037   -0.0026    0.0637
   -0.0478    0.1276   -0.0020    0.2969    0.1374    0.4142    0.0238   -0.0067   -0.0458   -0.0390    0.1841
   -0.1893   -0.3633    0.0294    0.2641   -0.0970    0.1015    0.0199   -0.0946   -0.0367    0.1152    0.0190
   -1.0000   -1.0000    0.1716    1.0000   -0.1716   -0.1303    0.0730   -0.5293   -0.2705    0.3462   -0.2073
    0.0011   -0.0000   -0.0016   -0.2552    0.0000   -0.0112    0.0001    0.0022    0.0000   -0.0000   -0.0248
    0.0078   -0.0086    0.0016    0.1877    0.0086   -0.0568    0.0126   -0.0015    0.0013   -0.0011   -0.0000
```

Fig. 10.21 Internal Unknowns [Fmatrix]

Step 17: The nodal displacements {δ} are worked out by the direct product of [Jmatrix], Global flexibility matrix [G] and coefficients corresponding to internal moments i.e., [Fmatrix]. **Figure.10.22** depicts the nodal displacements corresponding to each frequency value. The normalization procedure is carried out for calculating the relative or corresponding values only possible displacement degrees of freedom shows the possible values of frequency while all other 24 values are fully restrained.

```
MATLAB Command Window

>> Displacement = Jmatrix*G*Fmatrix

Displacement =

  Columns 1 through 14

   -0.0005   -0.0010    0.0001   -0.0111   -0.0010   -0.0034   -0.0002   -0.0003    0.0000    0.0004   -0.0022
   -0.0007   -0.0013   -0.0001   -0.0158    0.0013    0.0048   -0.0006    0.0001    0.0004   -0.0003    0.0001
    0.0000    0.0000    0.0000   -0.0070    0.0000    0.0003    0.0001   -0.0000    0.0000    0.0000    0.0003
   -0.0005   -0.0010    0.0001   -0.0111   -0.0010   -0.0034    0.0002    0.0003   -0.0000   -0.0004    0.0022
    0.0000    0.0000   -0.0001   -0.0100   -0.0000   -0.0004   -0.0000   -0.0001   -0.0000    0.0000    0.0010
    0.0000    0.0000    0.0000   -0.0070   -0.0000    0.0003   -0.0001    0.0000   -0.0000   -0.0000   -0.0003
   -0.0005    0.0010    0.0001   -0.0111    0.0010   -0.0034   -0.0001    0.0004    0.0002    0.0003    0.0029
   -0.0007    0.0013   -0.0001   -0.0158   -0.0013    0.0048    0.0006   -0.0004    0.0005    0.0001   -0.0026
   -0.0005    0.0010    0.0001   -0.0111    0.0010   -0.0034    0.0001   -0.0004   -0.0002   -0.0003   -0.0029
    0.0000   -0.0000   -0.0000   -0.0050    0.0000   -0.0002    0.0000    0.0000    0.0000   -0.0000   -0.0005
   -0.0003    0.0007   -0.0001   -0.0079   -0.0007    0.0024   -0.0003    0.0002   -0.0002   -0.0000    0.0013
   -0.0003   -0.0007   -0.0001   -0.0079    0.0007    0.0024    0.0003   -0.0000   -0.0002    0.0001   -0.0000
```

Fig. 10.22 Nodal Displacements {δ}

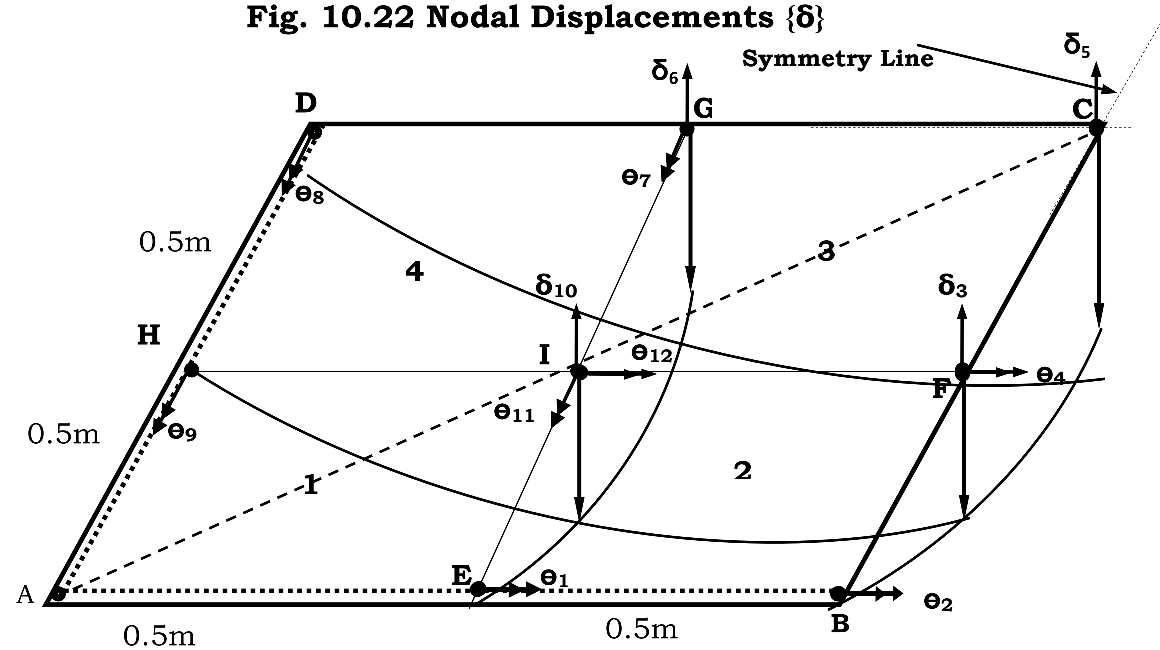


Fig. 10.23 Bottom Left Quadrant of Plate (1-2-3-4)

Table 10.1 shows the Frequency Ratio of IFM based natural frequency and exact solution **[99, 100]** available for the first four modes.

Table 10.1 Frequency Ratio

Mode Number	First	Second	Third	Fourth
$\omega_{IFM}/\omega_{EXACT}$	0.841	1.432	1.432	1.035

In **Fig. 10.21,** fourth column represents all the internal unknowns from F_1 to F_9 for the Element 1, F_{10} to F_{18} for Element 2, F_{19} to F_{27} for Element 3 and F_{28} to F_{36} for Element 4 for the lowest frequency value respectively. Substituting all the values of corresponding internal unknowns in M_x, M_y and M_{xy} one can calculate the moments. Normalized moments with respect to values at point F **(Fig. 10.23)** for the fundamental first frequencies are given in **Table 10.2.** The normalized nodal displacements **(Fig.10.22)** are reported in **Table 10.3.**

Table 10.2 Normalized Moments For (ω_1=64.1478rad/seconds)

Normalized Moments	Element 1 At Point I	Element 2 At Point F	Element 3 At Point C	Element 4 At Point G
M_{xx}	0.6612	1.0	1.438	0.9612
M_{yy}	0.6912	1.0	1.497	1.0406
M_{xy}	2.414	1.0	0.414	-1.000

Table 10.3 Normalized Nodal Displacements

θ_1	θ_2	δ_3	θ_4	δ_5	δ_6	θ_7	θ_8	θ_9	δ_{10}	θ_{11}	θ_{12}
1.1	1.58	0.7	1.00	0.7	0.7	1.1	1.58	1.1	0.5	0.79	0.79

Convergence study is also carried out. **Figure 10.24** shows that the first fundamental frequency converges towards the exact solution **[100].**

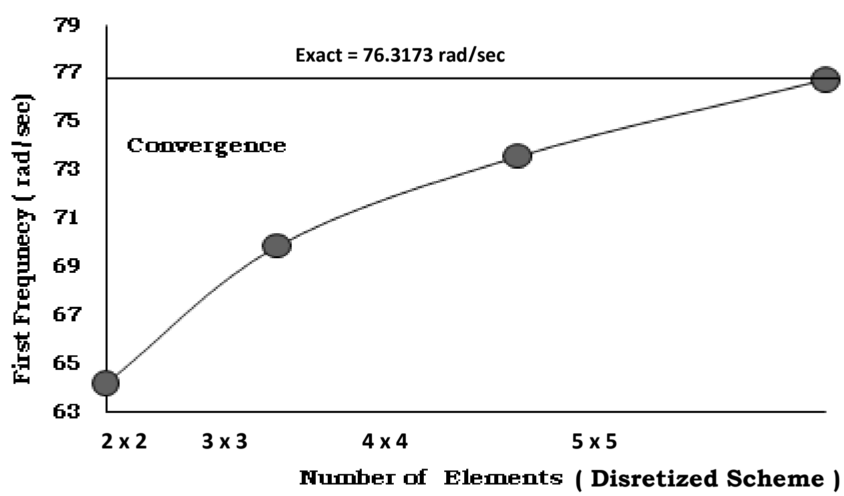


Fig. 10.24 Plot for Convergence

The remaining problems are attempted using 5 x 5 discretization scheme **(Fig.10.25)** where in total 25 elements with 225 internal force components are considered as primary unknowns. The global displacement degrees of freedom for the 5 x 5 discretization are 75. Thus, for making the global equilibrium matrix square, one needs 150 compatibility conditions. Eigen value analysis is carried out only for the first four frequencies out of the total 75 values. The unknown force vector [Fmatrix] is of size (225 x 1) which is readily available from the matlab based matrix operation, where the moments and nodal displacements are calculated along the points lying on the diagonal line by considering element numbers as 1, 17, 25 21 and 9 for the first lowest frequency value.

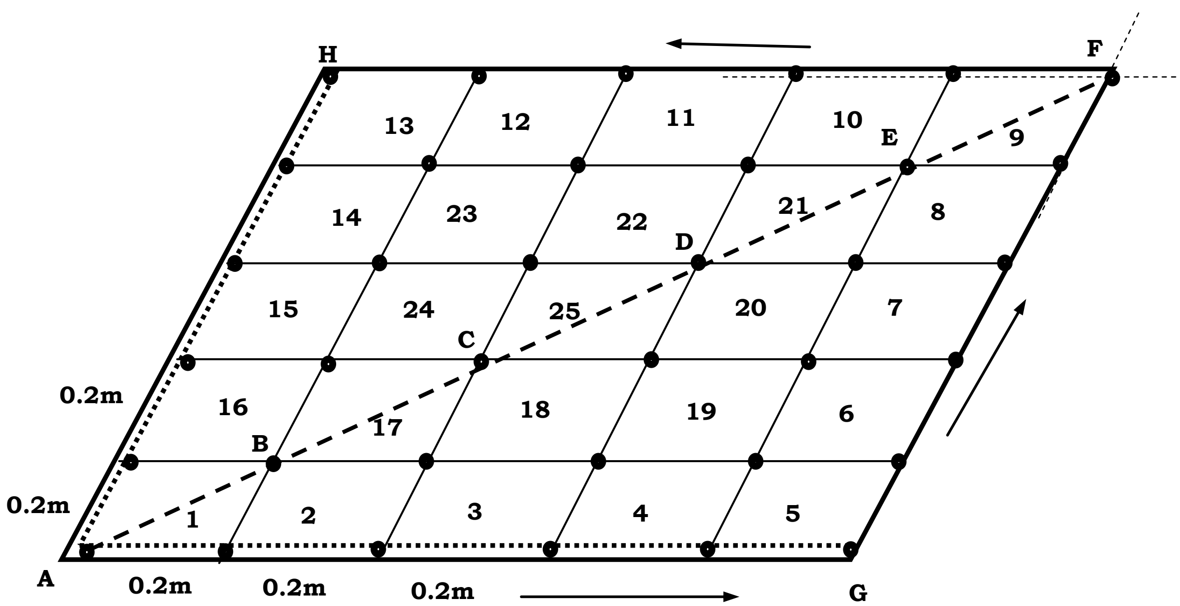


Fig. 10.25 Discretization Scheme 5 x 5

Table 10.4 shows the values of normalized moments (Mx) and vertical deflection with respect to value corresponding to unity at the centre of plate i.e. at Node F. Deformed shapes for the full plate based on the results obtained for the first four frequencies are drawn using Matlab facility and are depicted here in **Figs. 10.26 to 10.28.**

Table 10.4 Normalized Nodal Moments and Deflections

Points	M_x	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00
E	0.947	1.016	0.984
D	0.792	1.087	0.852
C	0.642	1.187	0.532
B	0.231	1.277	0.221
A	0.00	1.311	0.00

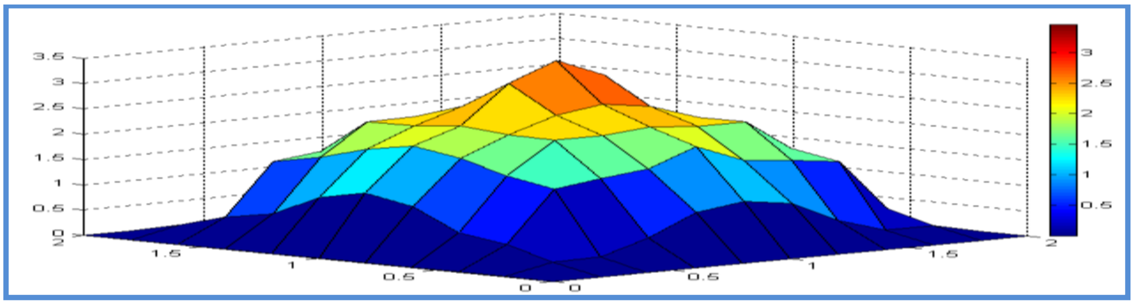


Fig. 10.26 First Mode Deformation Pattern (ω_{11})

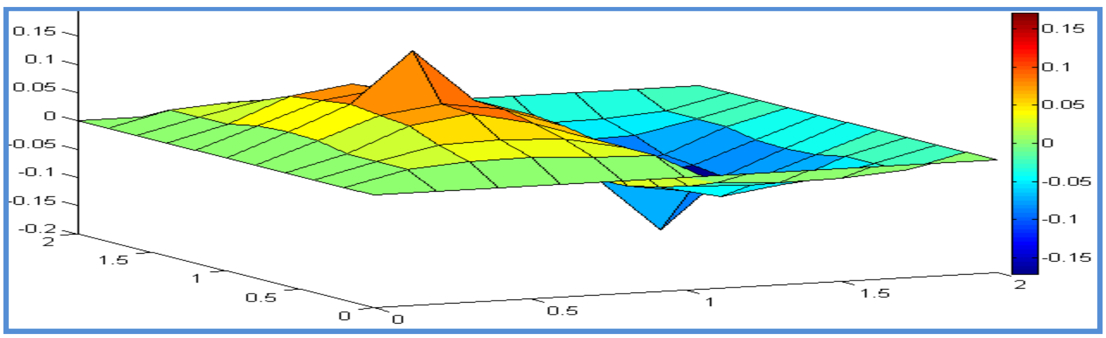


Fig. 10.27 Second and Third Mode Deformation Pattern (ω_{12} or ω_{21})

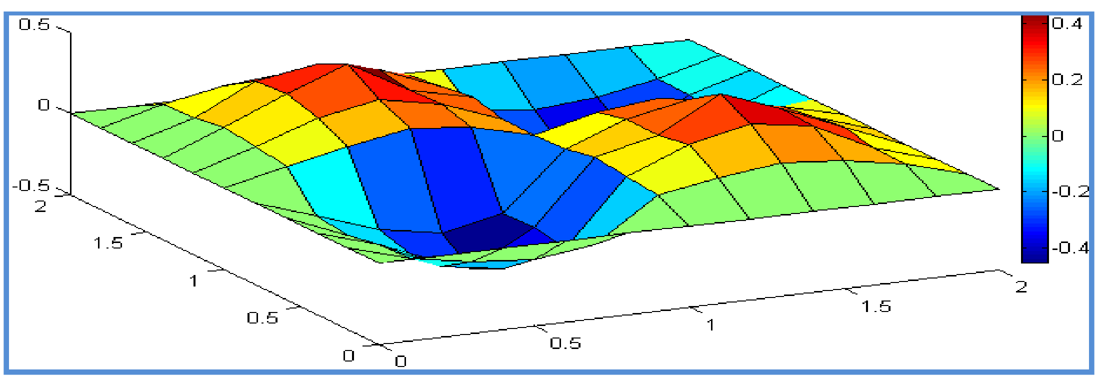


Fig. 10.28 Fourth Mode Deformation Pattern (ω_{22})

Consistent mass approach is also used here for frequency analysis of simply supported plate problem where the basic consistent mass matrix for an element is of size 12 x 12. Using standard stiffness based assembly procedure a global mass matrix of size 75 x 75 is generated. IFM based eigen value analysis is carried out by replacing the lumped mass matrix by consistent mass matrix. The Frequency ratios (FR) worked out based on the consistent mass are reported in **Table 10.5** for the first four modes.

Table 10.5 Frequency Ratio

Mode Number	First	Second	Third	Fourth
$\omega_{IFM}/\omega_{EXACT}$	0.882	0.911	0.911	0.998

10.3 CLAMPED SQUARE PLATE EXAMPLE

A clamped square plate of the same dimensions and material properties is now considered for the frequency analysis by considering dual symmetry and discretizing the left most quadrant into 5 x 5 mesh pattern by using lumped mass approach. **Table 10.6** shows frequency ratios for first four mode based on lumped and consistent mass approaches.

Table 10.6 Frequency Ratio

Mode Number	$\omega_{IFM\,(L)}/\omega_{EXACT}$	$\omega_{IFM(C)}/\omega_{EXACT}$
First	0.9122	0.925
Second	0.9517	0.966
Third	0.9517	0.966
Fourth	0.9828	0.992

Table 10.7 shows the normalized nodal moments, and deflections corresponding to unit value at the centre of the plate.

Table 10.7 Normalized Nodal Moments and Deflections

Points	M_x	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00
E	0.843	0.944	0.766
D	0.622	0.846	0.611
C	0.387	0.288	0.411
B	0.102	0.122	0.218
A	-0.002	0.006	0.000

Figs. 10.29 to 10.31 depict the deformation patterns for different modes.

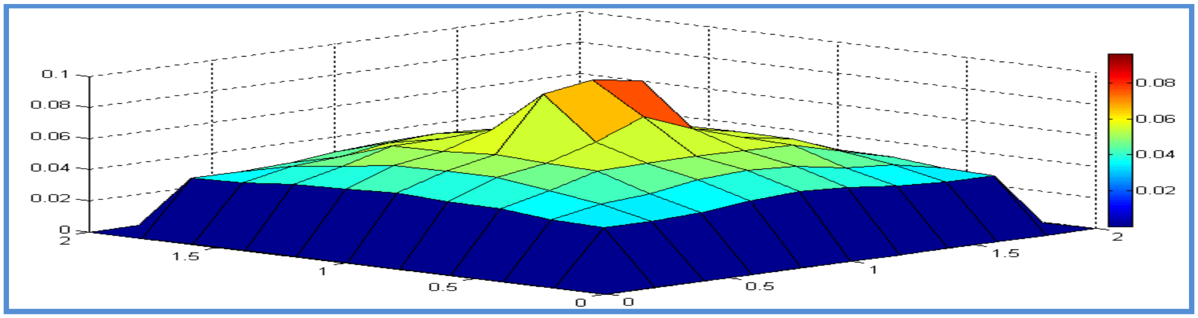


Fig. 10.29 First Mode Deformation Pattern (ω_{11})

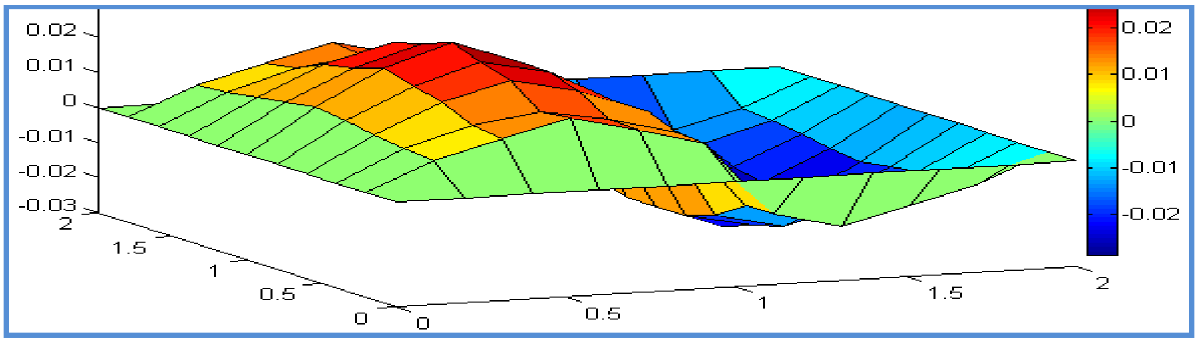


Fig. 10.30 Second & Third Mode Deformation Pattern (ω_{12} or ω_{21})

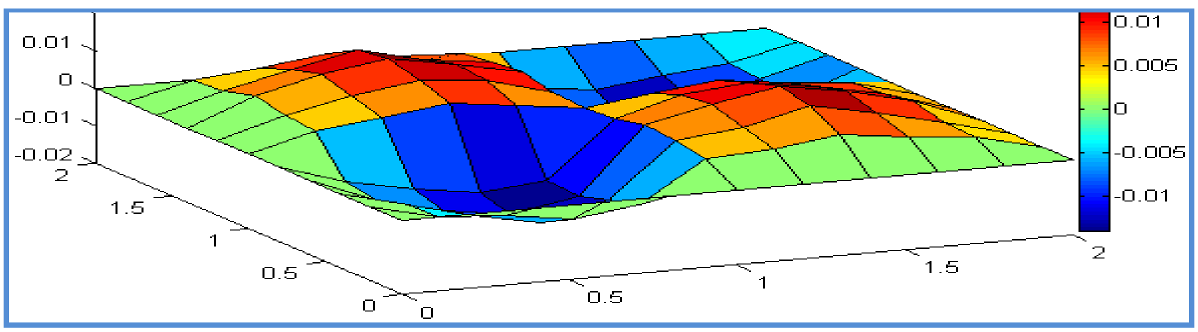


Fig. 10.31 Fourth Mode Deformation Pattern (ω_{22})

10.4 S-C-S-C SQUARE PLATE EXAMPLE

A square plate with two opposite edges (AB and CD) simply supported and the other two (BC and AD) clamped having same dimensions and material properties is considered now by discretizing bottom-left quadrant into 5 x 5 mesh under lumped and consistent mass criteria. The frequency results are summarized **Table 10.8** whereas results for normalized moments and nodal displacements given in **Table 10.9. Figs. 10.32** to **10.35** show the deformation patterns for different modes.

Table 10.8 Comparison of Frequency Values

Mode Number	$\omega_{(L)}$ rad/sec		$\omega_{(C)}$ rad/sec	
	IFM	EXACT	IFM	EXACT
First	100.01	111.741	105.09	111.741
Second	191.20	211.872	197.463	211.872
Third	249.98	267.532	257.08	267.532
Fourth	421.394	429.521	424.36	429.521

Table 10.9 Normalized Nodal Moments and Deflections

Points	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.00
E	0.744	0.665	0.722	0.836
D	0.533	0.511	0.622	0.633
C	0.288	0.218	0.651	0.311
B	0.155	0.093	0.677	0.112
A	-0.091	0.000	0.721	0.000

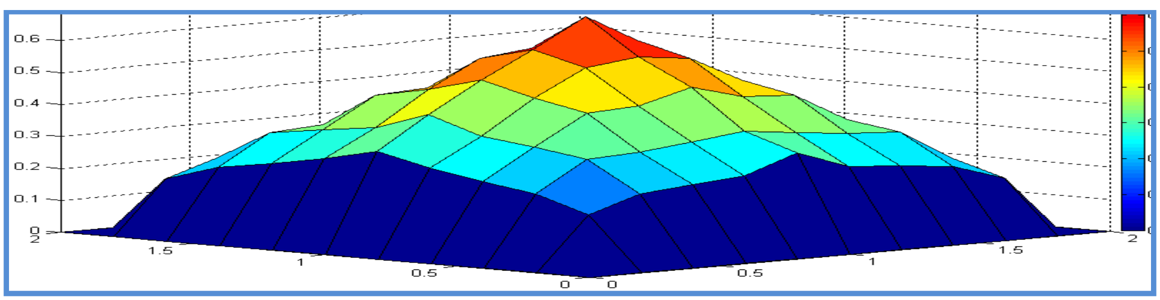


Fig. 10.32 First Mode Deformation Pattern (ω_{11})

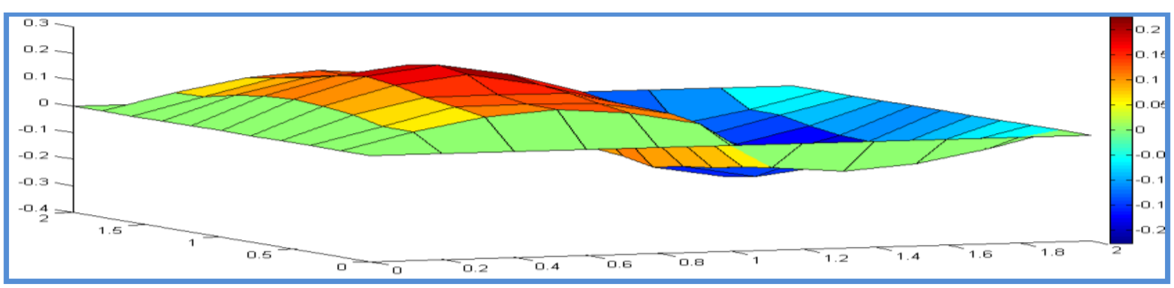


Fig. 10.33 Second Mode Deformation Pattern (ω_{12})

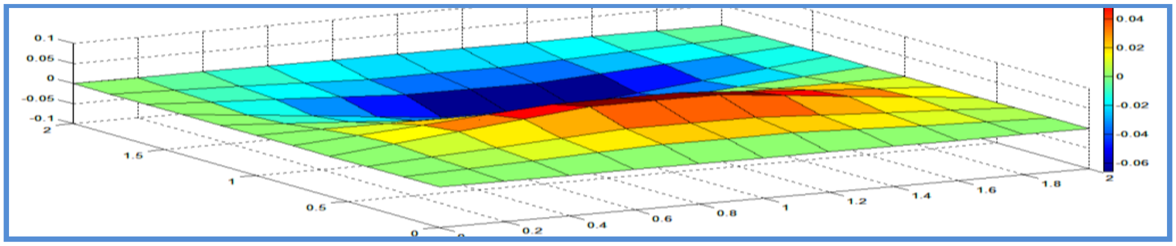


Fig. 10.34 Third Mode Deformation Pattern (ω_{21})

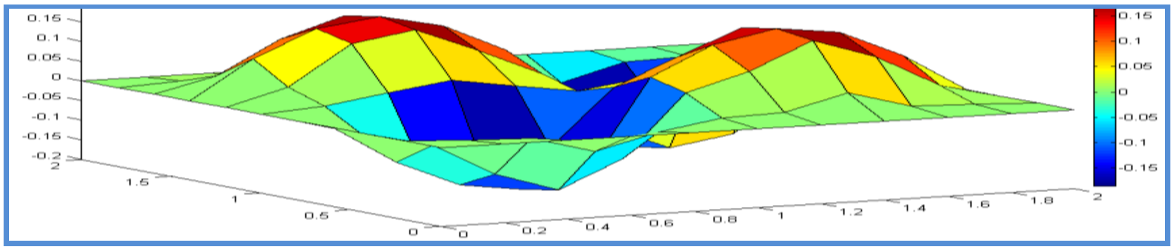


Fig. 10.35 Fourth Mode Deformation Pattern (ω_{22})

10.5 S-F-S-F SQUARE PLATE EXAMPLE

A square plate with two opposite edges (AB and CD) simply supported and other two (BC and AD) free having same dimensions and material properties is analysed now considering 5 x 5 mesh for quarter plate. The frequency results are summarized in **Table 10.10** whereas results for normalized moments and nodal displacements are reported in **Table 10.11.** Deformation patterns for first four modes are depicted in **Figs. 10.36** to **10.39.**

Table 10.10 Comparison of Frequency Values

Mode Number	$\omega_{(L)}$ rad/sec		$\omega_{(C)}$ rad/sec	
	IFM	EXACT	IFM	EXACT
First	27.6417	38.1581	32.4725	38.1581
Second	50.6077	62.3478	58.1081	62.3478
Third	128.897	152.632	146.679	152.632
Fourth	160.968	180.640	178.472	180.640

Table 10.11 Normalized Nodal Moments and Deflections

Points	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.00
E	0.655	0.881	0.922	0.944
D	0.411	0.744	0.729	0.833
C	0.193	0.411	0.898	0.518
B	0.091	0.199	0.787	0.421
A	0.00	0.000	0.488	0.000

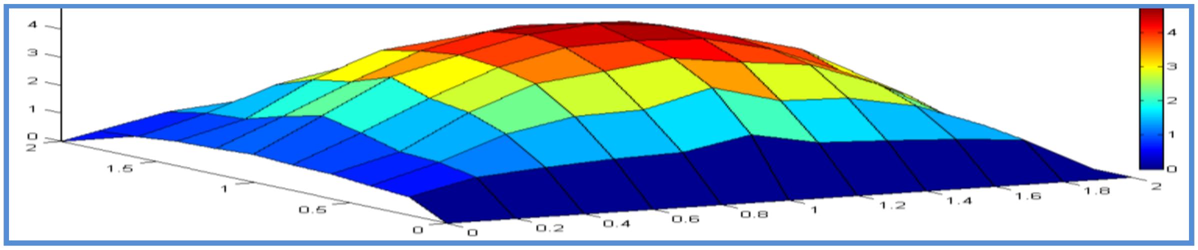


Fig. 10.36 First Mode Deformation Pattern (ω_{11})

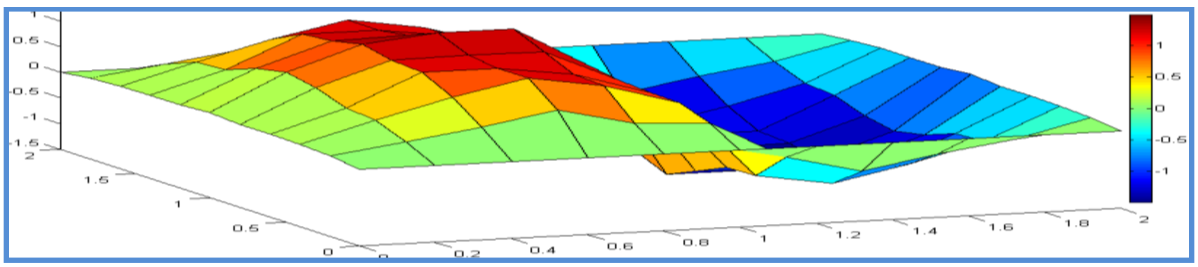


Fig. 10.37 Second Mode Deformation Pattern (ω_{12})

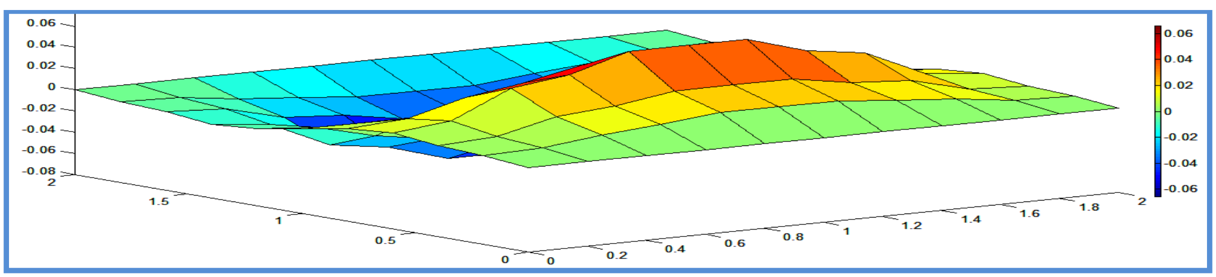


Fig.10.38 Second Mode Deformation Pattern (ω_{21})

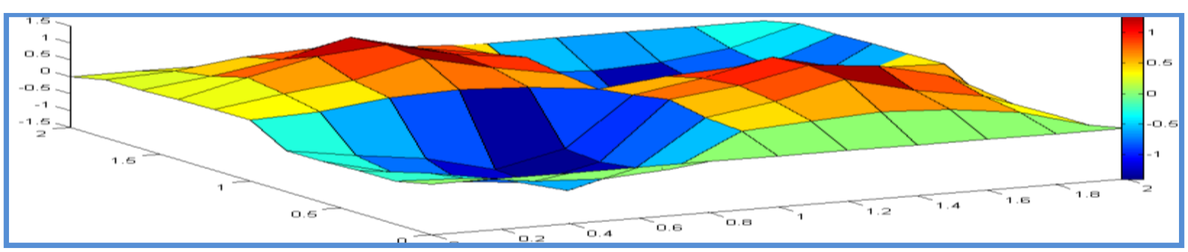


Fig. 10.39 Fourth Mode Deformation Pattern (ω_{22})

10.6 RECTANGULAR PLATE EXAMPLES

A rectangular plate having dimensions as 4m x 2m x 0.01m is considered now for frequency analysis. Due to symmetry, only quarter of the plate is discretized in 5 x 5 grid. Following the same procedure with same material properties internal moments and nodal displacements are calculated. Results are obtained for simply supported and clamped plate, frequency values are compared to the exact values are reported in **Table 10.13** whereas moments and deflections for two cases are reported here in **Table 10.14.**

Table 10.13 Comparison of Frequency Values

Mode Number	Simply Supported Plate		Clamped Plate	
	$\omega_{IFM\ (L)}/\omega_{EXACT}$	$\omega_{IFM\ (C)}/\omega_{EXACT}$	$\omega_{IFM\ (L)}/\omega_{EXACT}$	$\omega_{IFM\ (C)}/\omega_{EXACT}$
First	0.9122	0.832	0.8812	0.941
Second	0.9517	0.891	0.8991	0.932
Third	0.9517	0.899	0.9012	0.961
Fourth	0.9828	0.903	0.9312	0.988

Table 10.14 Normalized Nodal Moments and Deflections

Point	M_x		M_y		M_{xy}		Vertical Deflection	
	SS	CL	SS	CL	SS	CL	SS	CL
F	1.00	1.00	1.00	1.00	1.00	1.00	1.00	1.00
E	0.897	0.612	0.998	0.885	1.017	0.831	0.921	0.836
D	0.722	0.422	0.822	0.781	1.051	0.522	0.844	0.533
C	0.553	0.199	0.612	0.410	1.219	0.298	0.421	0.429
B	0.164	-0.102	0.221	0.320	1.322	0.102	0.198	0.103
A	0.00	-0.104	0.00	-0.095	1.412	0.001	0.00	0.000

Again deformation patterns for different modes can be drawn using matlab facility. Here these are included for the simply supported rectangular plate only in **Figs. 10.40** to **10.43**.

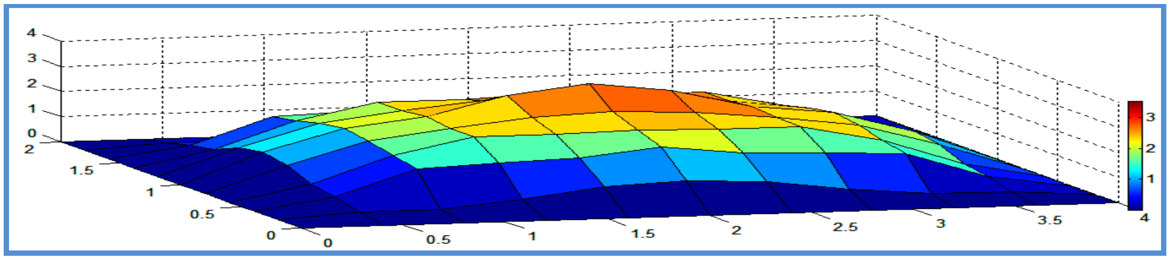


Fig. 10.40 First Mode Deformation Pattern (ω_{11})

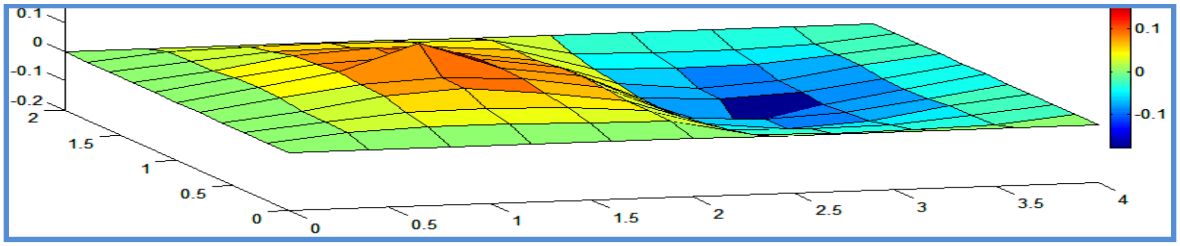


Fig. 10.41 Second Mode Deformation Pattern (ω_{12})

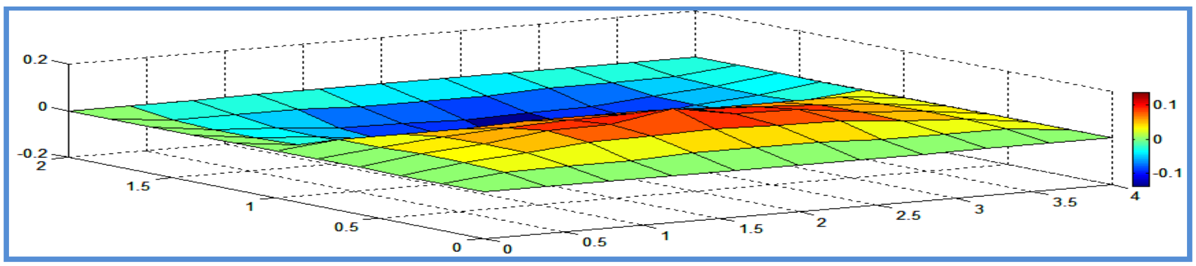


Fig.10.42 Third Mode Deformation Pattern (ω_{21})

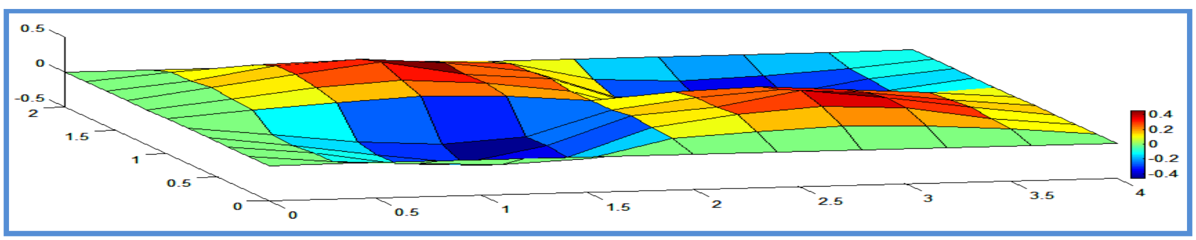


Fig. 10.43 Fourth Mode Deformation Pattern (ω_{22})

10.7 DISCUSSION OF RESULTS

- ❖ Convergence study of the first natural frequency under lumped mass criteria is carried out for simply supported plate by discretizing the quarter plate in 2 x 2, 3 x 3, 4 x 4 and 5 x 5 mesh. Continuous improvement is indicated. 5 x 5 discretization is found to give quite good results and hence the same mesh is used for solving the rest of the problems.

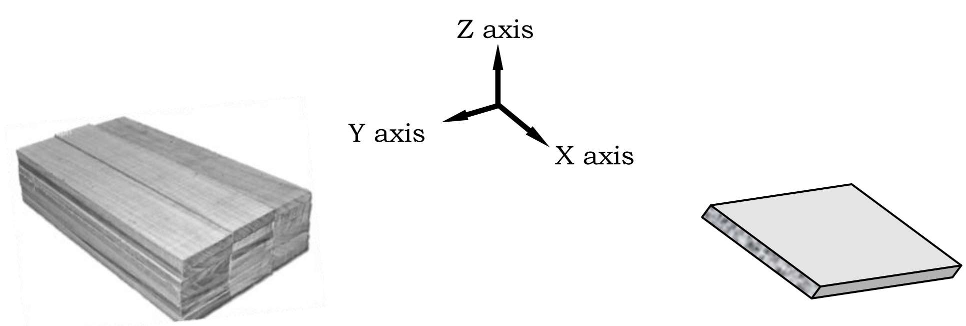
- With 2 x 2 discretization for simply supported plate the Frequency Ratio for the first frequency is found lower bound while for the other three it is upper bound. But with 5 x 5 discretization FR for the first four frequencies is found lower bound. For all other plate problems also with 5 x 5 discretization FR is found lower bound.
- In general, frequency calculated based on consistent mass criterion is found more nearer to the exact solution compared to that based on lumped mass criterion..
- The deformation pattern for the plate for first four frequencies are drawn using surface plotting module of Matlab named as "surf(x,y,z)". Square plate problems, with the symmetrical boundary condition, show fully up or down deformation with respective values of half waves (m and n) along the x and y axis. The first modal deformation pattern shows maximum normalized values at the centre of plate, second and third modal deformed patterns depict maximum value at the centre of left half and centre of right half with zero value along centre line in any one direction, while for fourth mode, having repetition of first mode values are depicted with opposite sign into respective quadrant.

CHAPTER 11

STATIC ANALYSIS OF ORTHOTROPIC PLATE PROBLEMS

11.1 GENERAL REMARKS

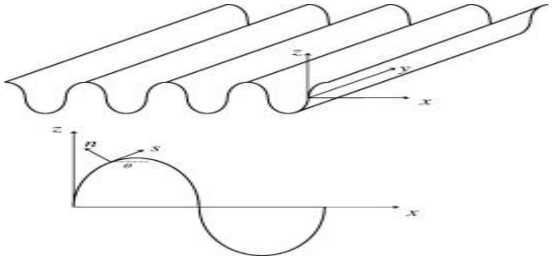
A special case of anisotropy in which the material properties are different in two orthogonal directions is called orthogonal anisotropy or orthotropy. From the plate bending analysis point of view there are two types of orthotropy namely material orthotropy **(Fig.11.1)** and structural orthotropy **(Fig.11.2)**. Material orthotropy is due to the physical structure of the material itself, while structural orthotropy is developed by special technique by which different members are fabricated as per the structural stiffness requirement. Practical examples of material orthotropy are wood, and certain crystals and fiber reinforced plastics. Stiffened plates and plates with ribs with varying rigidities in a direction parallel and perpendicular to the stiffeners or ribs are the examples of structural orthotropy. These plates are analytically modeled as equivalent orthotropic plates with elastic properties equal to the average properties of various components evenly distributed across the plate, which provides good approximation of measure of overall stiffness.



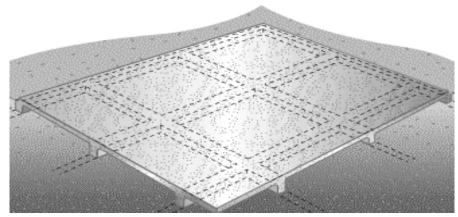
(a) Wooden Planks

(b) Glass Fiber Reinforced Plastic Plate

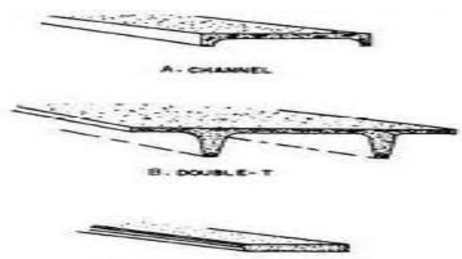
Fig 11.1 Examples of Material Orthotropy



(a) Corrugated Sheet



(b) Stiffened Raft Slab



(C) Precast Floor Slab

Fig 11.2 Examples of Structural Orthotropy

The analysis procedure for both depends upon actual calculation of rigidities in respective directions which totally depends upon values of modulus of elasticity (E), shear modulus (G) and Poisson's ratio (ν). For material orthrotropic cases it is easy to work out these material properties while for the structural orthotropic cases it is bit difficult and one may have to perform experiment to evaluate rigidities in two major directions.

As per the Kirchhoff's small deflection theory of isotropic plates, the numbers of independent elastic constants required are two i.e. E, ν. If the principal orthogonal direction of orthotropy coincides with the X and Y coordinate axis, it is evident that elastic constants E_x, E_y, ν_x, and ν_y are required for representing the material matrix $[D_{ortho}]$ of size 3 x 3. Strain stress relationship for materially orthotropic plate can be written as

$$\epsilon_x = \frac{\sigma_x}{E_x} - \nu_y \frac{\sigma_y}{E_y}, \qquad \epsilon_y = \frac{\sigma_y}{E_y} - \nu_x \frac{\sigma_x}{E_x} \quad \text{and} \quad \gamma_{xy} = \frac{\tau_{xy}}{G_{xy}} \qquad \dots (11.1)$$

In a matrix form, one can write the same as

$$\begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{1}{E_x} & \frac{-\nu_y}{E_y} & 0 \\ \frac{-\nu_x}{E_x} & \frac{1}{E_y} & 0 \\ 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix} \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = [D_{ortho}][\sigma] \qquad \dots (11.2)$$

As far as solution of orthotropic plate problems is concerned, IFM requires the same steps as used for isotropic plates. The only change is in flexibility matrix which is now calculated based on $[D_{ortho}]$ matrix discussed above.

A variety of examples of orthotropic plates are solved in the subsequent sections under different loading and support conditions using IFM based methodology. Where possible results are compared with the exact solution **[99,103]** available in the literature.

11.2 SIMPLY SUPPORTED ORTHOTROPIC PLATE EXAMPLE

A Glass Reinforced Plastic Plate having simply supported edges all over with dimensions as 2m x 2m x 3mm is shown in **Fig. 11.3.** Static analysis is carried out by considering dual symmetry and discretizing quadrant number 1 into 5 x 5 mesh with total 25 elements for: (i) Central Point Load (CPL) of 10kN and (ii) Uniform Lateral Pressure (ULP) of 10kN/m^2. The orthotropic plate has following elastic properties: E_x = 40kN/mm^2, E_y = 8kN/mm^2, G_{xy} = 4kN/mm^2 and ν_x = 0.25 **[102]**.

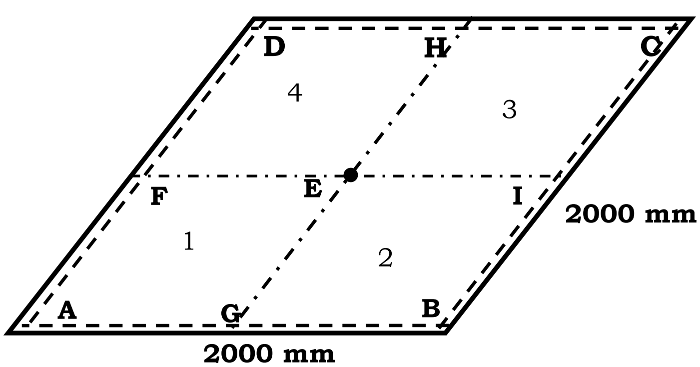
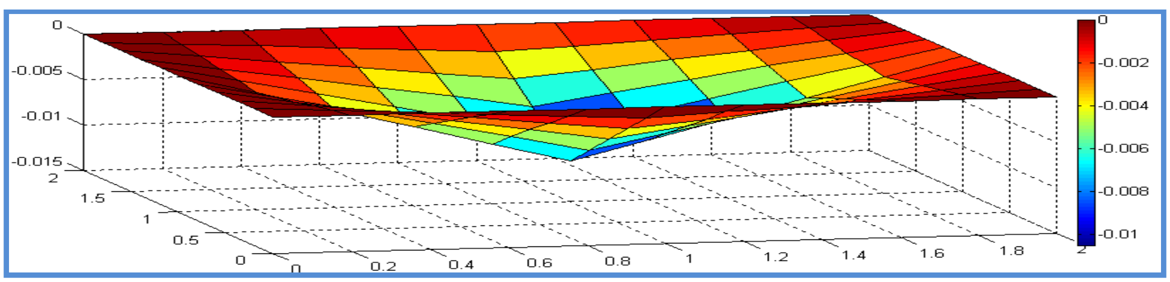


Fig 11.3 A Square Orthotropic Plate Bending Example

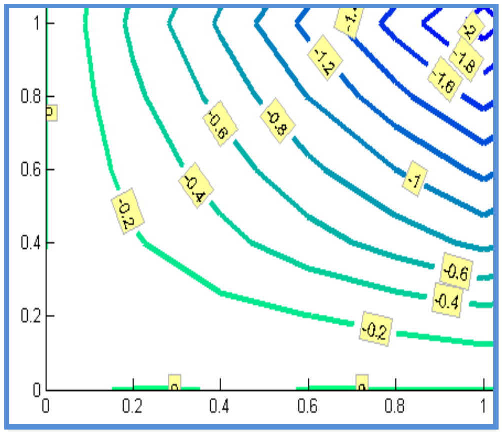
The basic steps and all the matrices are same as used for static analysis of isotropic plate bending problems. The development of global flexibility matrix [G] which is of size 225 x 225 is generated now using [D_{ortho}] for orthotropic material given in **Eq. (11.2)** instead of [D] for isotropic material. Following are the concise notations used for easy recognizing of the different cases included here. If edges AB, BC, CD and DA are simply supported then S_S_S_S with central point load is considered as S_S_S_S_CPL while for ULP the CPL is replaced by ULP. Similarly, C is used for clamped and F for free edge conditions respectively. Results obtained for moment and central displacement at the centre of plate i.e. at point E are compared with the exact solution in terms of Central Moment Ratio (CMR) and Central Deflection Ratio (CDR) and are reported here in **Table 11.1**. Various plots for 3D deformed shape and 2D moment contours which are developed in Matlab using proper modules are depicted in **Figs. 11.4 and 11.5.**

Table 11.1 Deflection and Moment Ratios

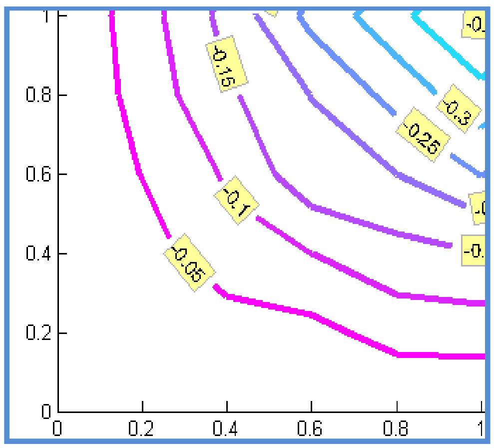
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio(CMR) $M_{E\,(IFM)}/M_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.872	0.889	0.841	0.883
ULP	0.902	0.932	0.872	0.904



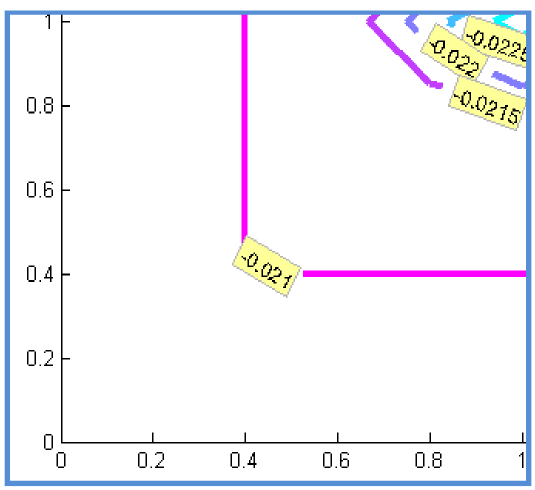
(i) Deformed Shape of Simply Supported Plate



(ii) Mxx Contours

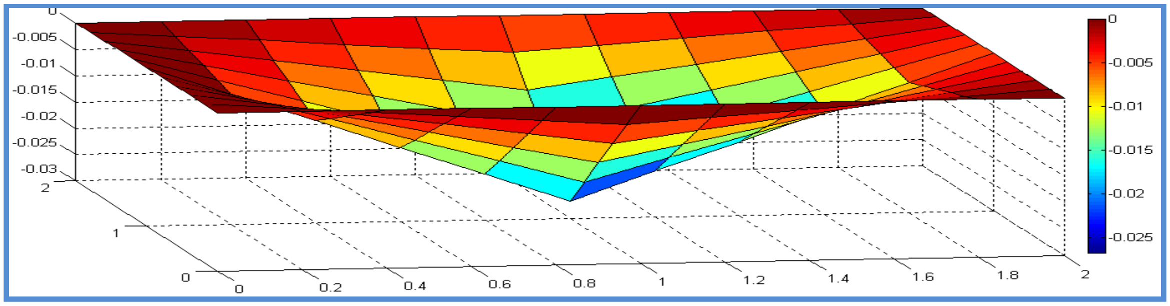


(iii) Myy Contours

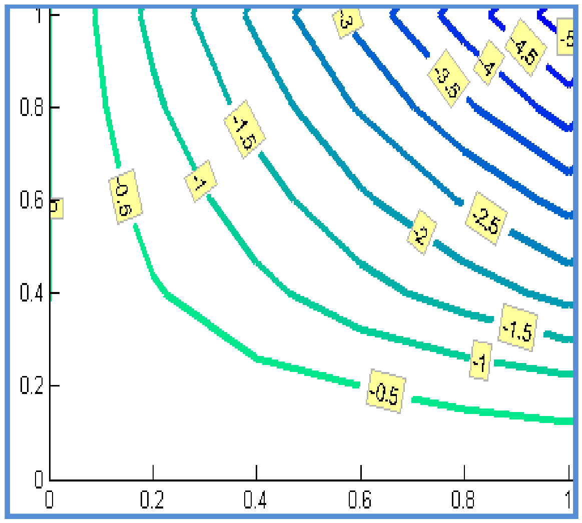


(iv) Mxy Contours

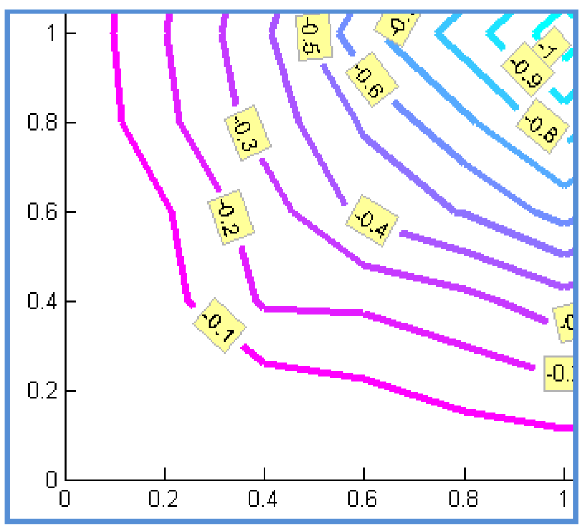
Fig.11.4 Various Plots of SS Square Plate under CPL



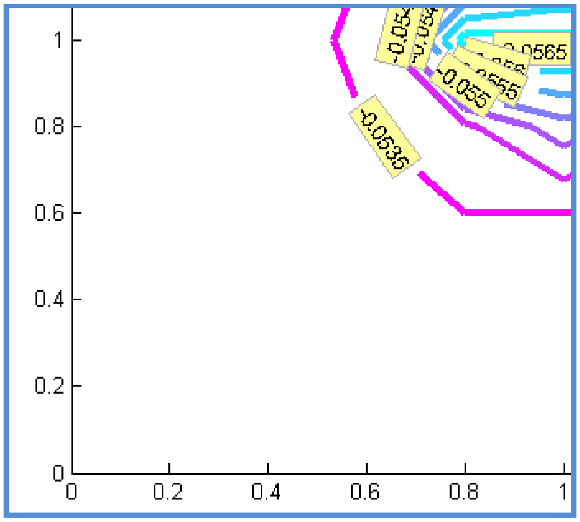
(i) Deformed Shape of Simply Supported Plate



(ii) Mxx Contours



(iii) Myy Contours



(iv) Mxy Contours

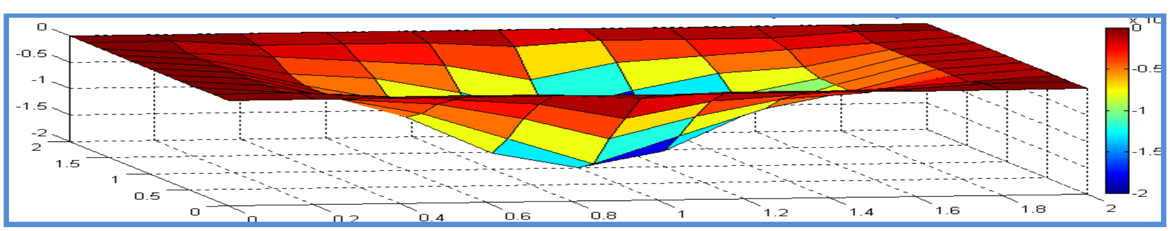
Fig. 11.5 Various Plots of SS Square Plate under ULP

11.3 CLAMPED ORTHOTROPIC PLATE EXAMPLES

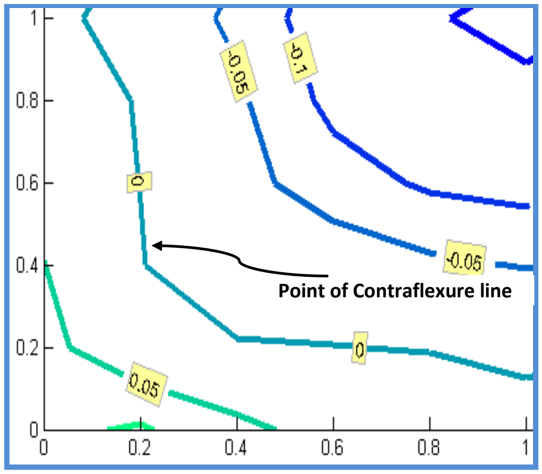
Considering all the edges of square plate as clamped with all other parameters are same, results are obtained for the moments and central displacement at the centre of plate and are compared with the exact solution in terms of Central Moment Ratio (CMR) and Central Deflection Ratio (CDR) in **Table 11.2**. Plots for 3D deformed shape and 2D moment contours are depicted in **Figs. 11.6** and **11.7.**

Table 11.2 Deflection and Moment Ratios

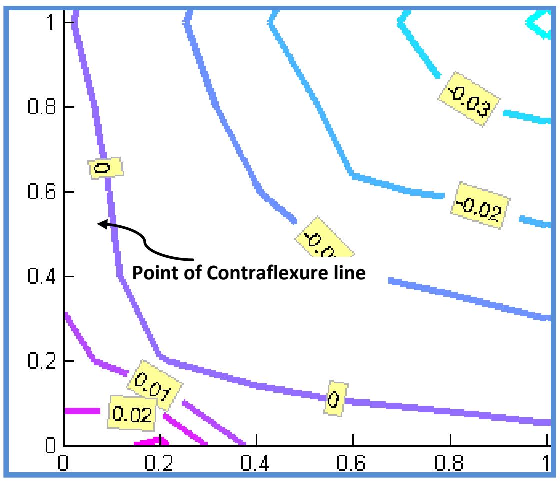
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio(CMR) $M_{E\,(IFM)}/M_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.893	0.885	0.851	0.891
ULP	0.922	0.904	0.877	0.892



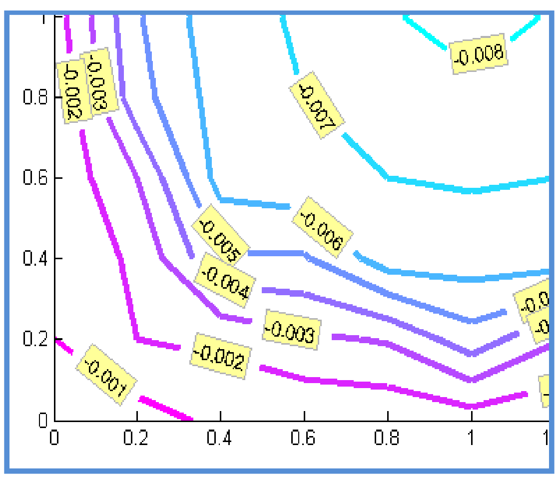
(i) Deformed Shape of Clamped Plate



(ii) Mxx Contours

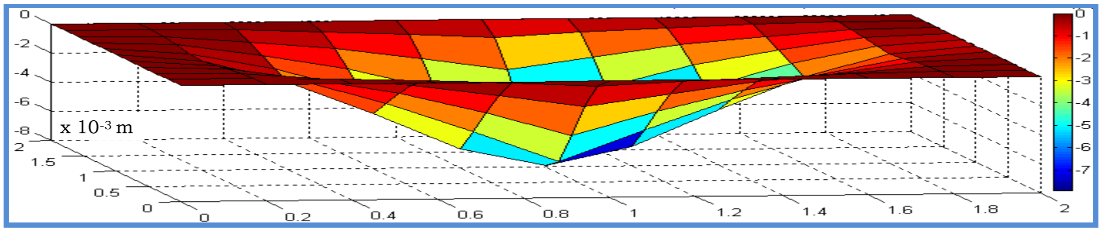


(iii) Myy Contours

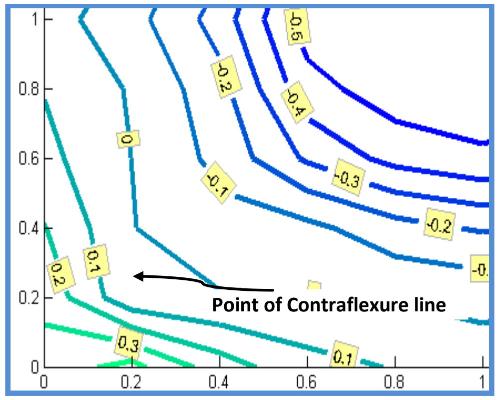


(iv) Mxy Contours

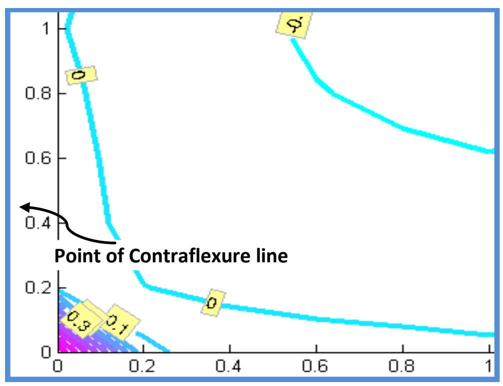
Fig. 11.6 Various Plots of Clamped Square Plate under CPL



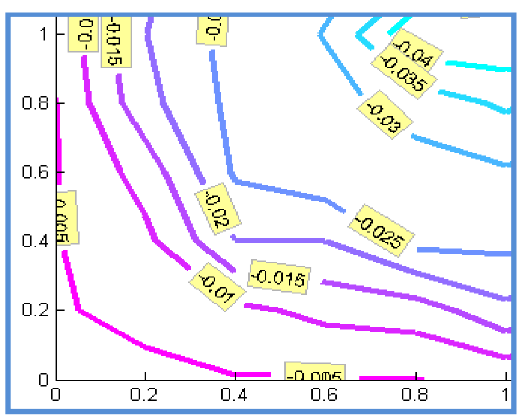
(i) Deformed Shape of Clamped Plate



(ii) Mxx Contours



(iii) Myy Contours



(iv) Mxy Contours

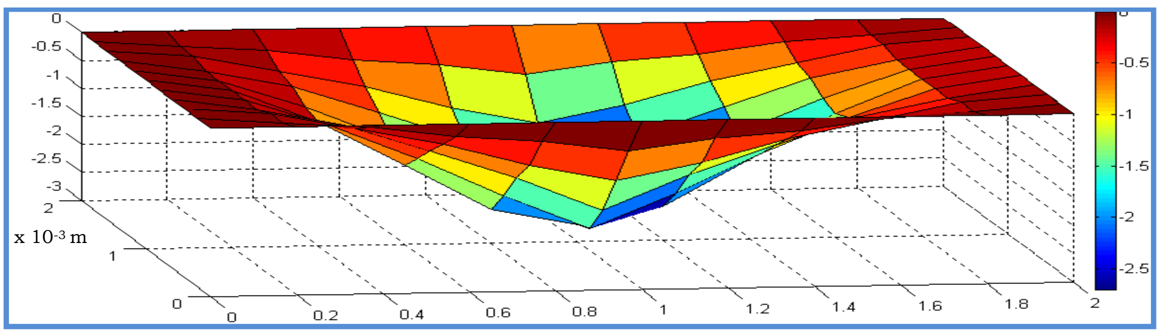
Fig. 11.7 Various Plots of Clamped Square Plate under ULP

11.4 C_S_C_S ORTHOTROPIC SQURE PLATE EXAMPLE

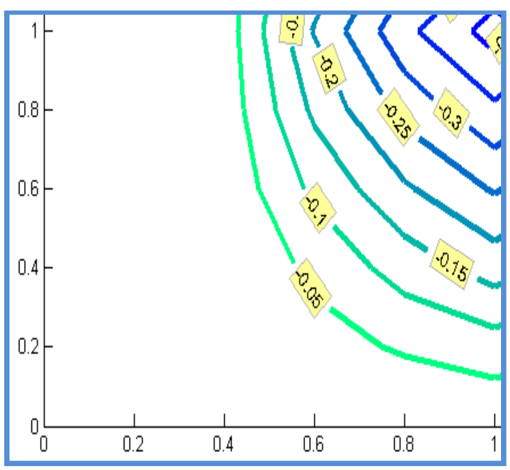
Considering AB and CD edges of square plate shown in **Fig. 11.3** as clamped and remaining two as simply supported results are obtained by discretizing quarter plate in 5 x 5 mesh. Various plots for 3D deformed shape and 2D moment contours developed using Matlab software and depicted in **Figs. 11.8** and **11.9.** Moment and central deflection obtained using IFM are compared with the exact solution in **Table 11.3.**

Table 11.3 Non-Dimensional Parameters at Centre of Plate

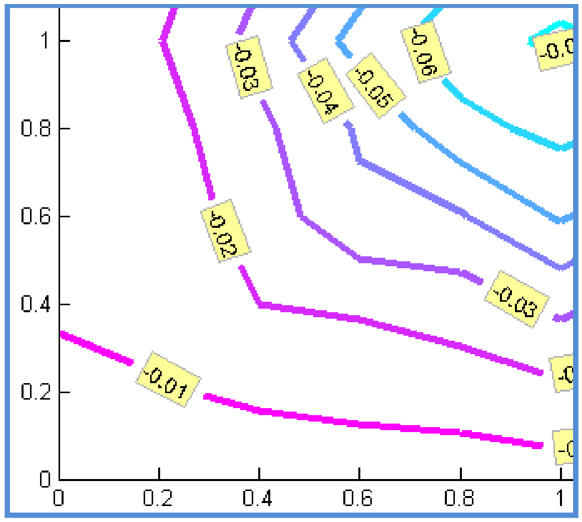
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio(CMR) $M_{E\,(IFM)}/M_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.853	0.902	0.892	0.806
ULP	0.916	0.941	0.909	0.894



(i) Deformed Shape for C_S_C_S Plate

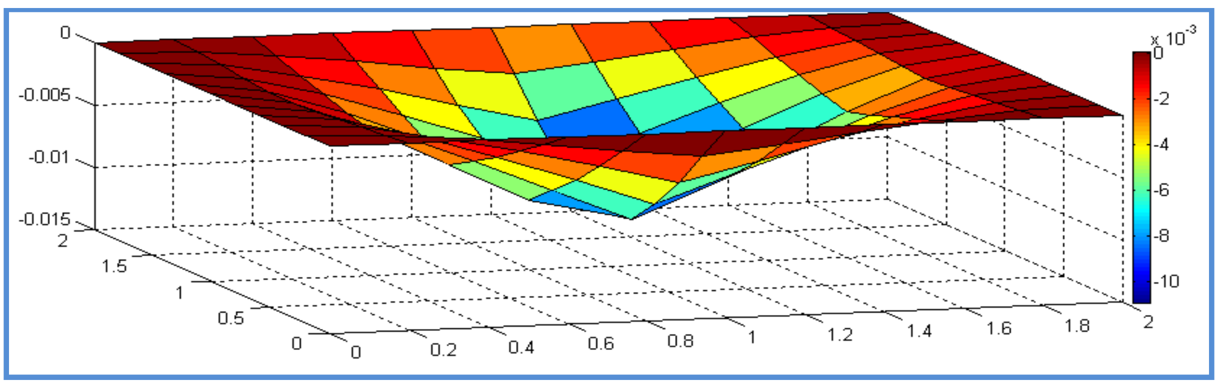


(ii) Mxx Contours

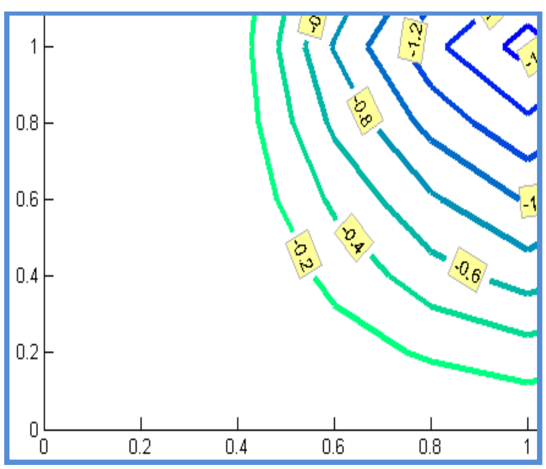


(iii) Myy Contours

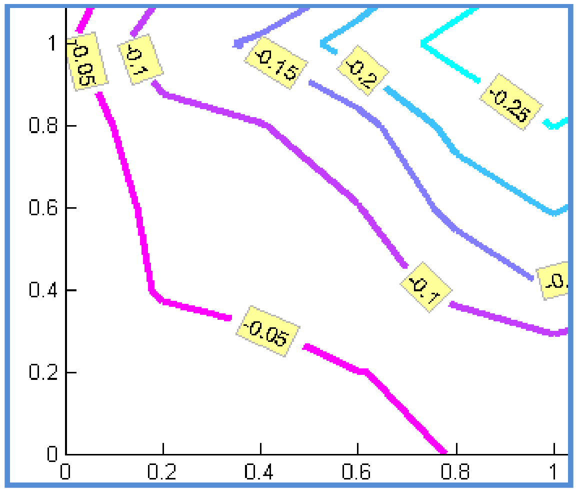
Fig. 11.8 Various Plots for C_S_C_S Square Plate under CPL



(i) Deformed Shape for C_S_C_S Plate



(ii) Mxx Contours



(iii) Myy Contours

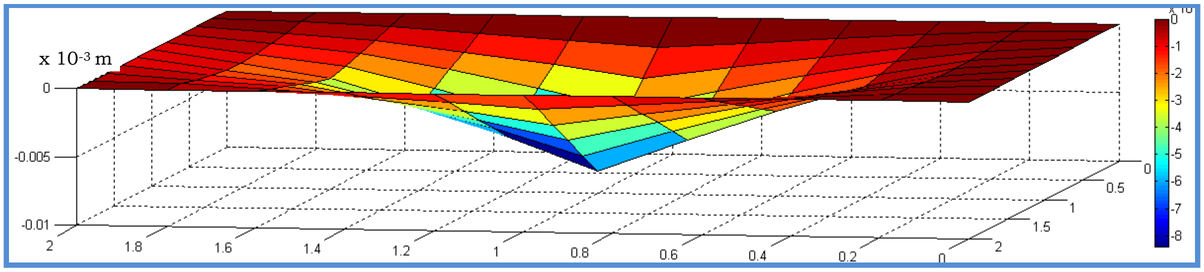
Fig 11.9 Various Plots for C_S_C_S Square Plate Under ULP

11.5 S_C_S_S ORTHOTROPIC PLATE EXAMPLES

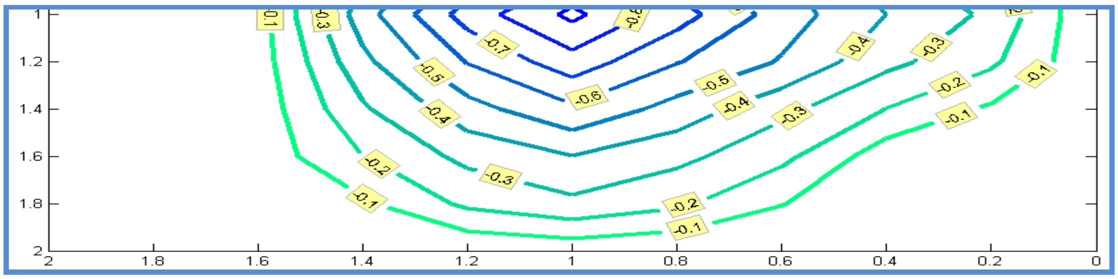
With AB, AD and DC edges as simply supported and BC as clamped (**Fig. 11.3**) due to one way symmetry only half of the plate is analysed considering 10 x 5 discretization. There are total 50 elements with 450 internal unknown moments. The global equilibrium matrix is developed of size 136 x 450 in which 214 compatibility conditions are required for complete solution. The global flexibility matrix is of size 450 x 450. Various plots for 3D deformed shape and 2D moment contours for CPL and ULP loading conditions are developed in Matlab using proper modules as depicted in **Figs. 11.10** and **11.11.** Moment and central displacement at centre of plate are compared with the in **Table 11.4**.

Table 11.4 Non-Dimensional Parameters at Centre of Plate

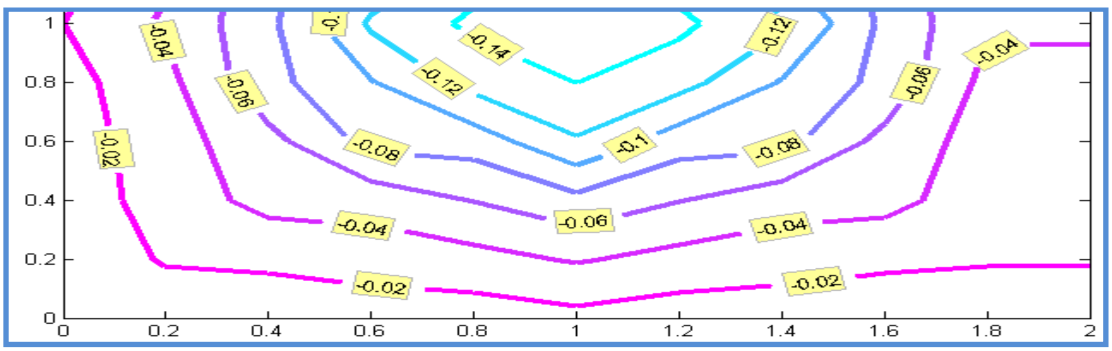
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio (CMR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.829	0.927	0.858	0.804
ULP	0.953	0.911	0.903	0.872



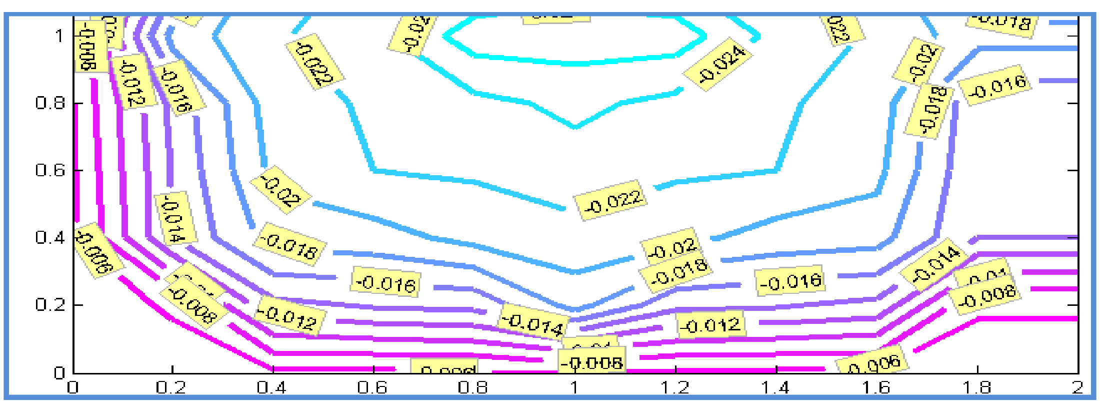
(i) Deformed Shape for S_C_S_S Plate



(ii) Mxx Contours

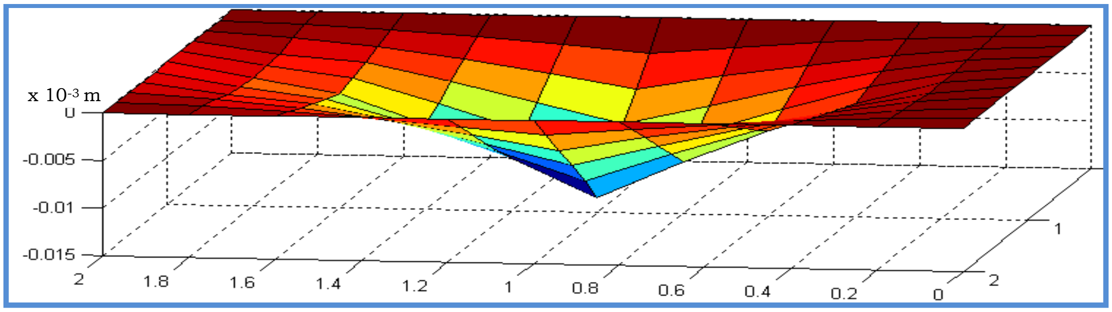


(iii) Myy Contours

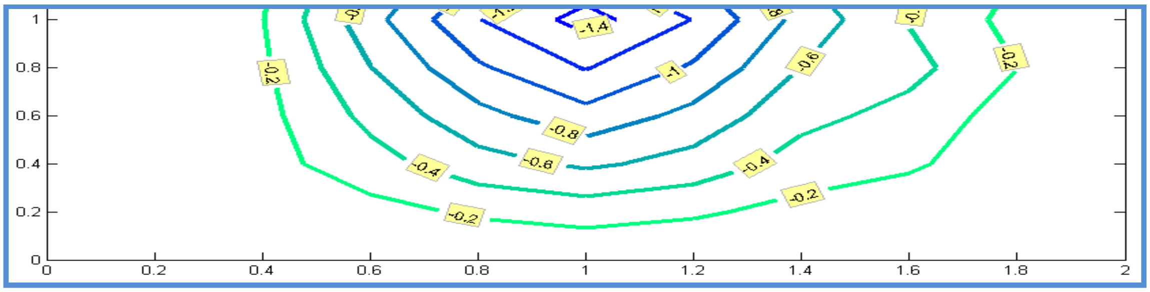


(iv) Mxy Contours

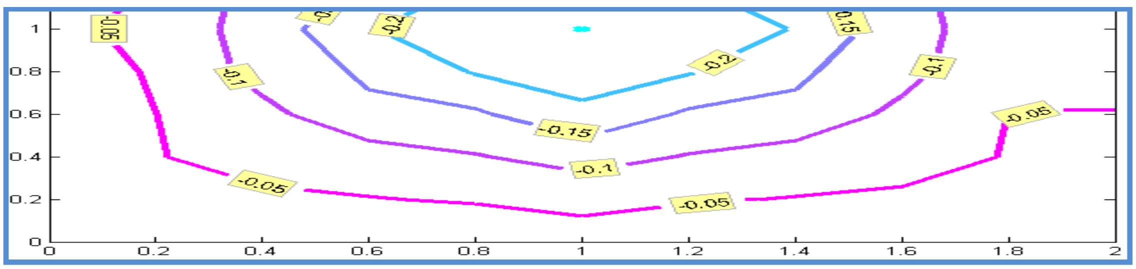
Fig. 11.10 Various Plots for S_C_S_S Square Plate Under CPL



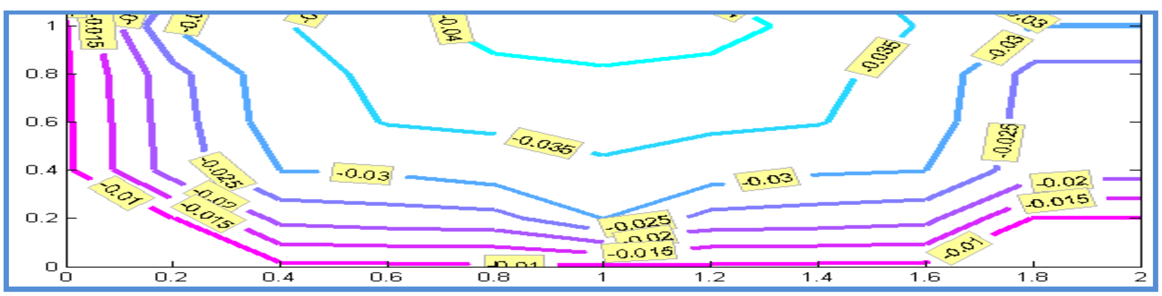
(i) Deformed Shape for S_C_S_S Plate



(ii) Mxx Contours



(iii) Myy Contours



(iv) Mxy Contours

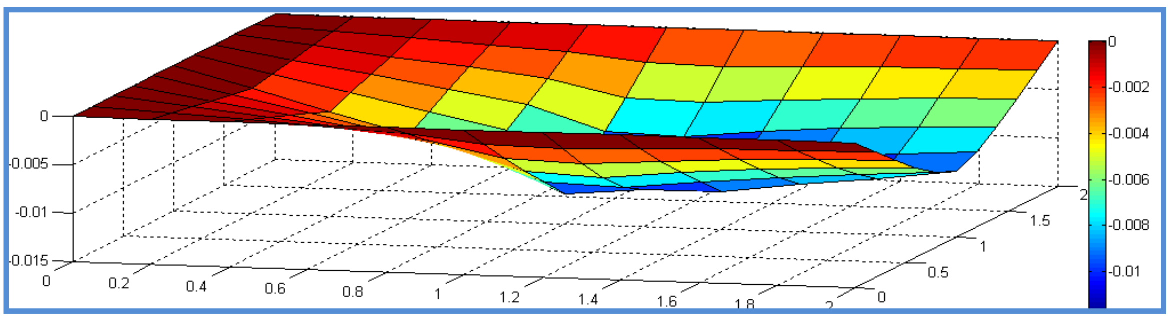
Fig. 11.11 Various Plots for S_C_S_S Square Plate Under CPL

11.6 S_F_S_S ORTHOTROPIC PLATE EXAMPLE

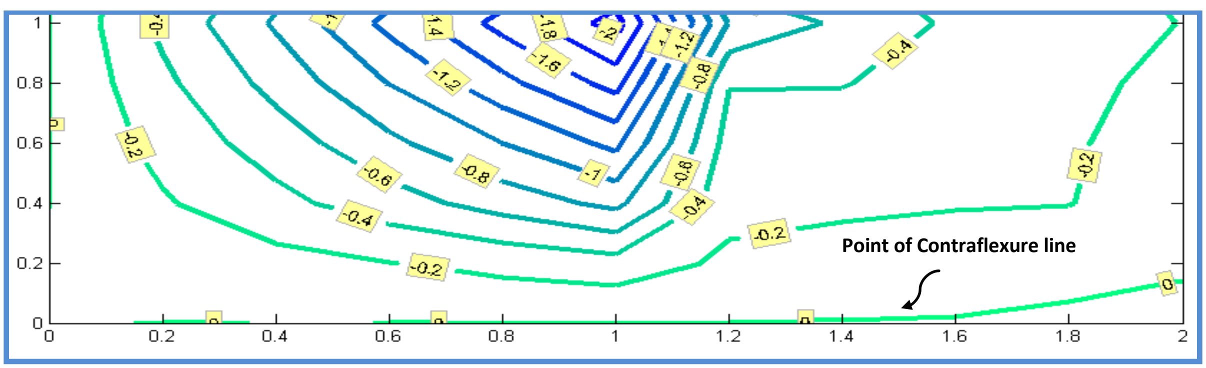
Considering AB, AD and DC edges as simply supported and BC as free **(Fig. 11.3)** half of the plate is discretized into 10 x 5 grid leading to 50 elements and 450 internal unknown moments. The global equilibrium matrix is of size 151 x 450 which requires 249 compatibility conditions for complete solution. The global flexibility matrix will be of size 450 x 450. Based on the results various plots for 3D deformed shape and 2D moment contours are developed in Matlab which are depicted in **Figs. 11.12** and **11.13.** Moments and central displacement at the centre of plate are compared with the exact solution in **Table 11.5**.

Table 11.5 Deflection and Moment Ratios at Centre of Plate

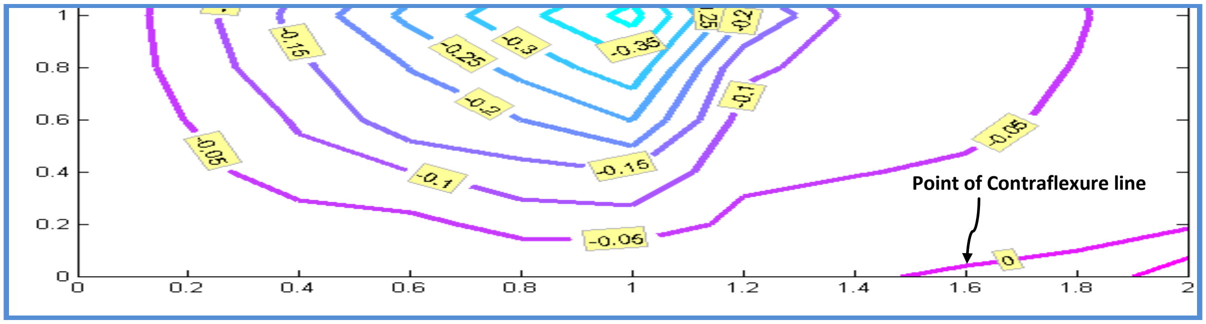
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio (CMR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.869	0.884	0.833	0.871
ULP	0.974	0.903	0.911	0.891



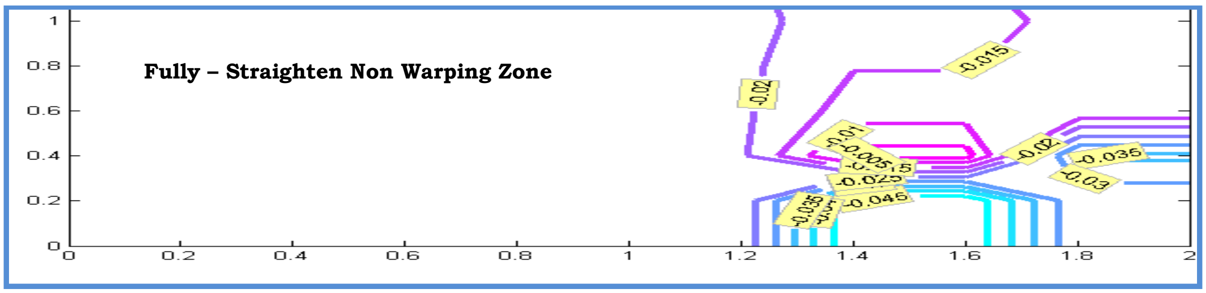
(i) Deformed Shape for S_F_S_S Plate



(ii) Mxx Contours

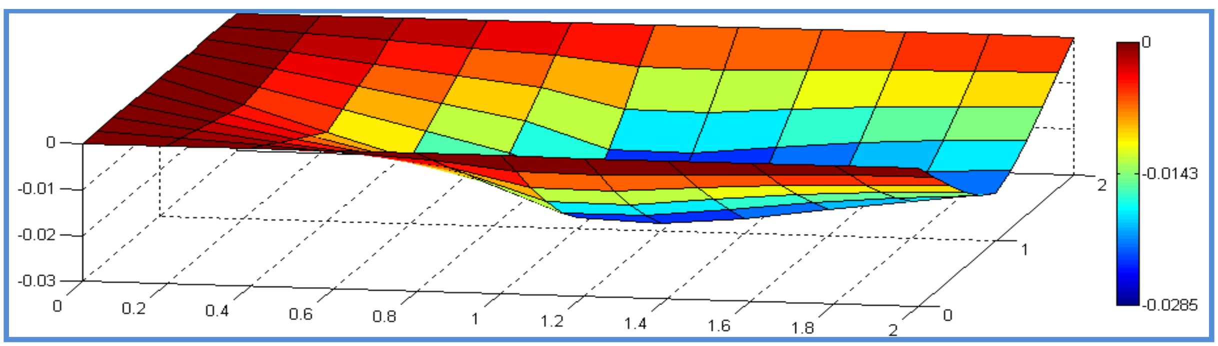


(iii) Myy Contours

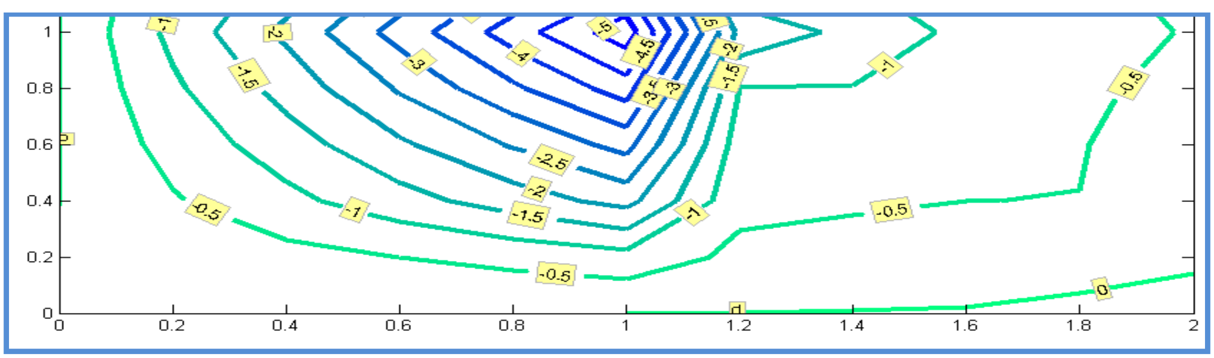


(iv) Mxy Contours

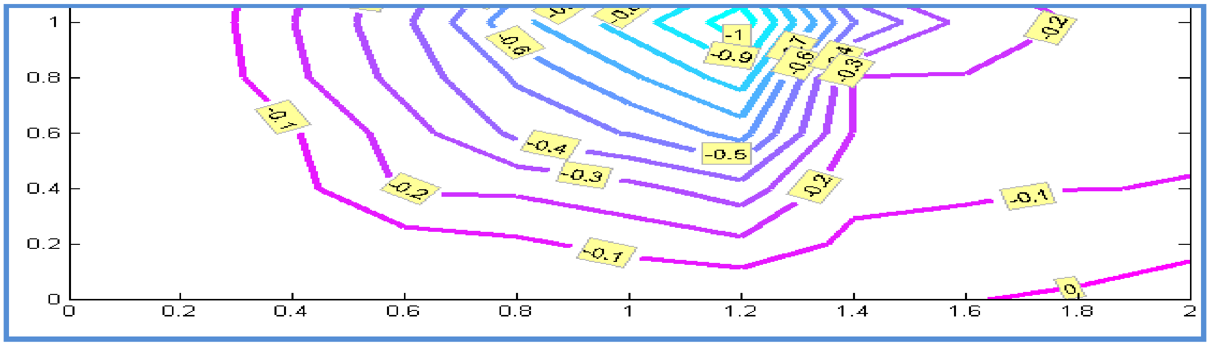
Fig. 11.12 Various Plots for S_F_S_S Square Plate Under CPL



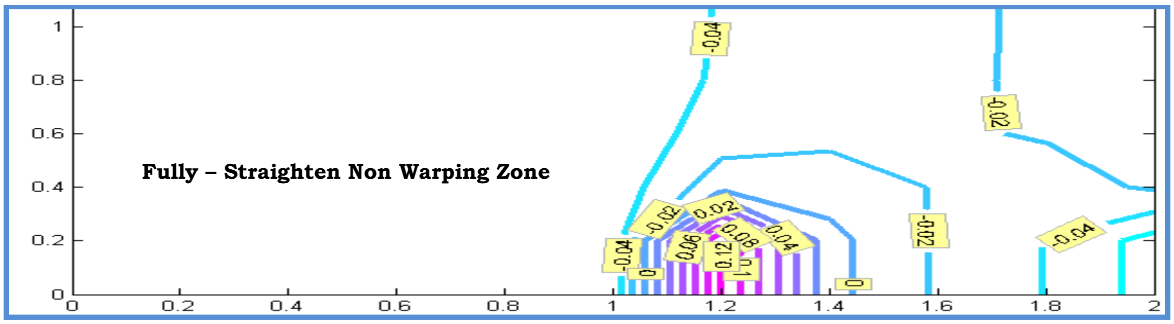
(i) Deformed Shape for S_F_S_S Plate



(ii) Mxx Contours



(iii) Myy Contours



(iv) Mxy Contours

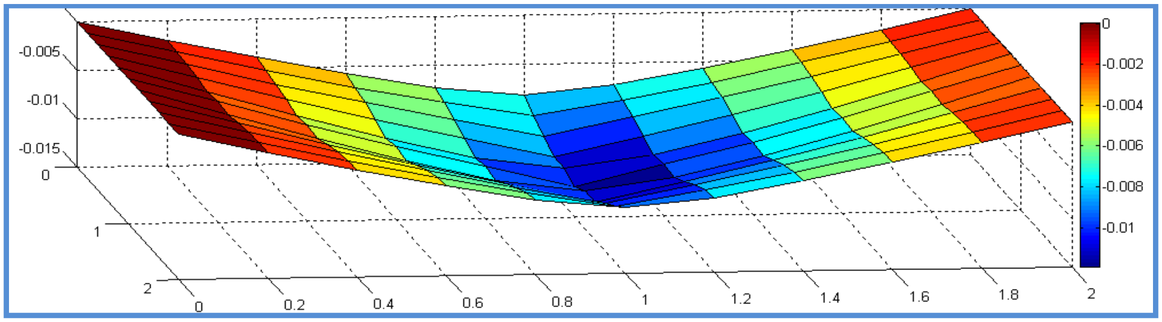
Fig 11.13 Various Plots for S_F_S_S Square Plate Under ULP

11.7 F_S_F_S ORTHOTROPIC PLATE EXAMPLE

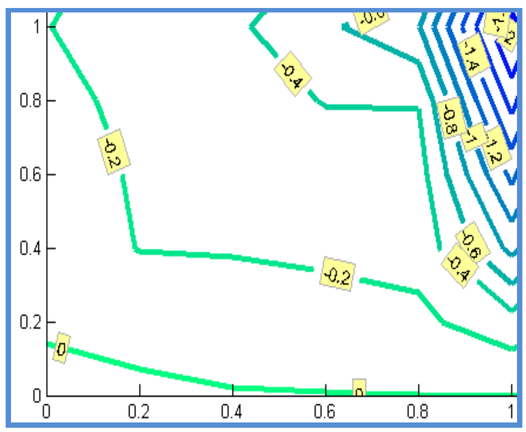
If AB and CD edges are free while other two are simply supported (**Fig. 11.3)**, the problem can be analysed considering dual symmetry. A (5 x 5) discretization scheme will have total 36 nodes in which 20 nodes are with 3 ddofs, 9 nodes are with 2 ddofs and 7 nodes are with one ddof. Thus, there are total 85 displacement degrees of freedom with 225 force degrees of freedom. Hence it requires 140 numbers of compatibility conditions for making the global equilibrium matrix square. Results are reported here in the form of plots for 3D deformed shape and 2D moment contours as depicted in **Figs. 11.14** and **11.15.** Moments and displacement at the centre of plate are compared with exact solution and are reported here in the form of ratios in **Table 11.6**.

Table 11.6 Displacement and Moment Ratios at Centre of Plate

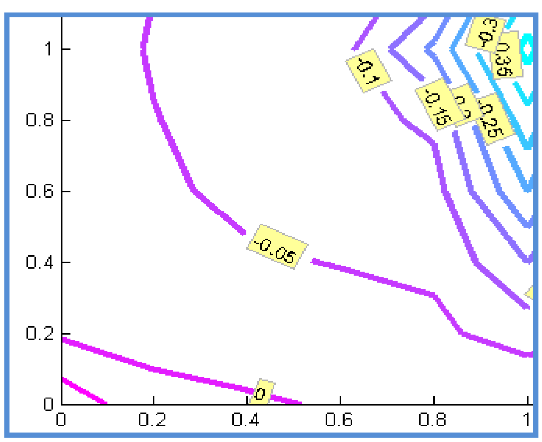
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\ (IFM)}/\delta_{E\ (EXACT)}$	Central Moment Ratio (CMR) $\delta_{E\ (IFM)}/\delta_{E\ (EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.887	0.86	0.851	0.885
ULP	0.892	0.876	0.889	0.842



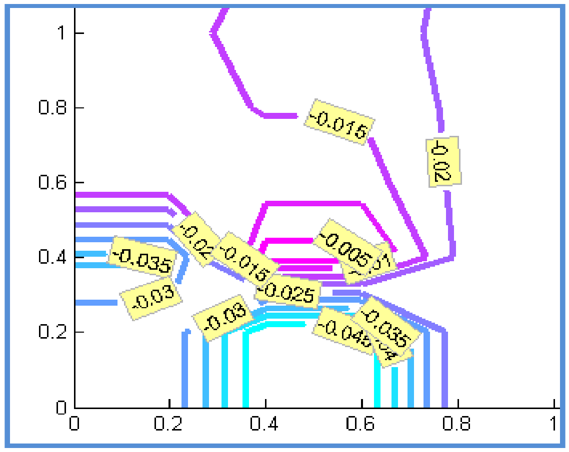
(i) Deformed Shape for F_S_F_S Plate



(ii) Mxx Contours

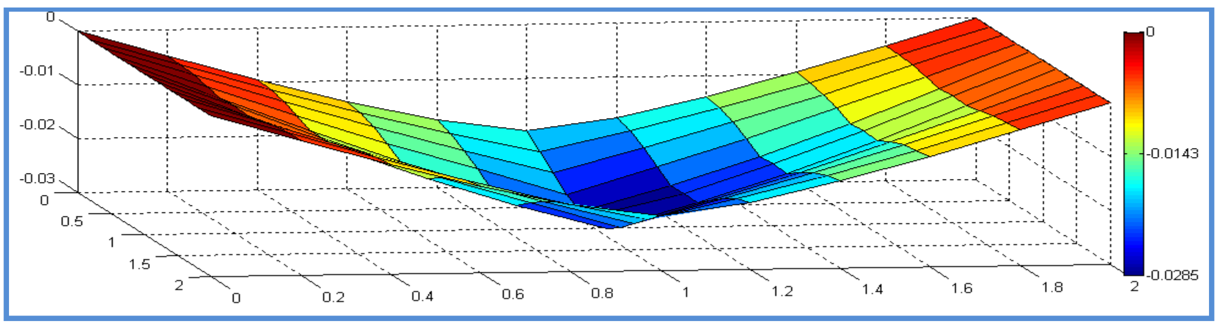


(iii) Myy Contours

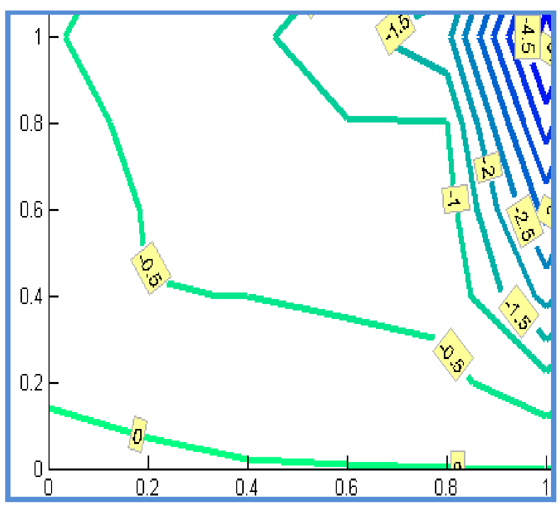


(iv) Mxy Contours

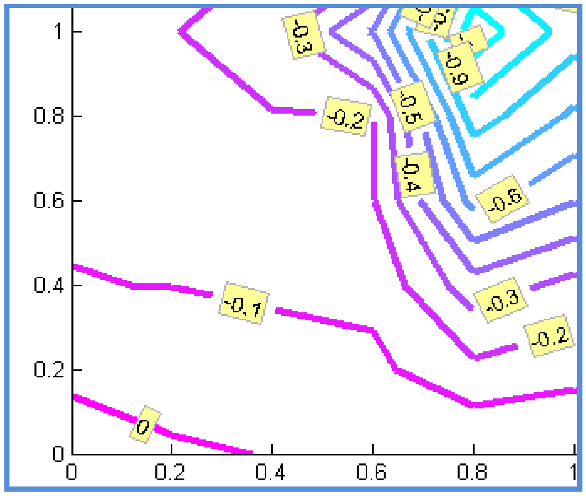
Fig 11.14 Various Plots for F_S_F_S Square Plate Under CPL



(i) Deformed Shape for F_S_F_S Plate



(ii) Mxx Contours



(ii) Myy Contours

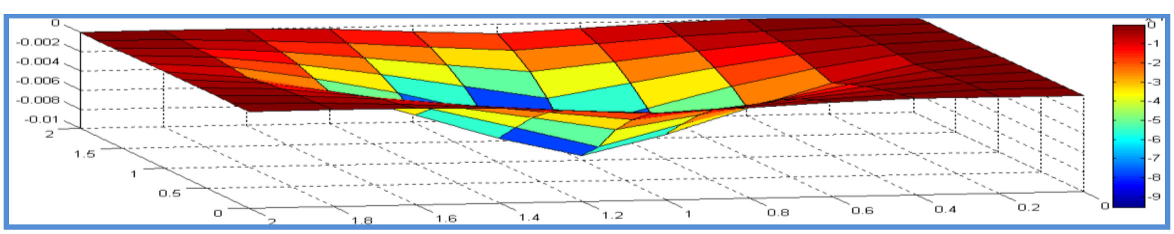
Fig 11.15 Various Plots for F_S_F_S Square Plate Under ULP

11.8 F_C_F_C ORTHOTROPIC PLATE EXAMPLES

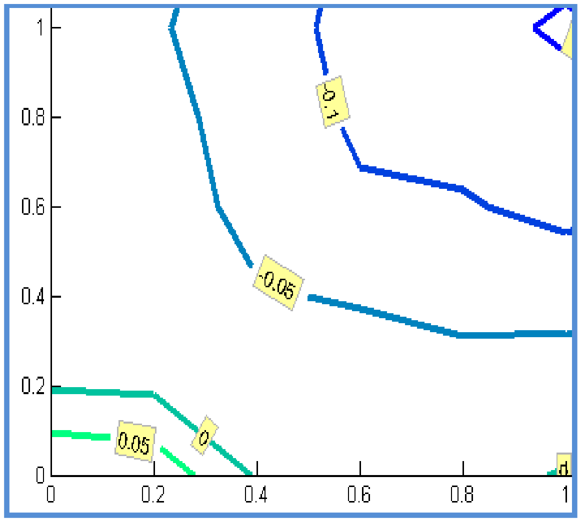
Considering AB and CD edges as free with other two as clamped, the plate is analysed considering double symmetry with 5 x 5 discretization scheme. There are total 36 nodes in which 20 nodes are with 3 ddofs, 9 nodes are with 2 ddofs. Thus, there are total 78 displacement degrees of freedom with 225 force degrees of freedom. Plots for 3D deformed shape and 2D moment contours are included here in **Figs 11.16 and 11.17** whereas moments and central displacement at the centre of plate are included in **Table 11.7**

Table 11.7 Frequency and Moment Ratios at Centre of Plate

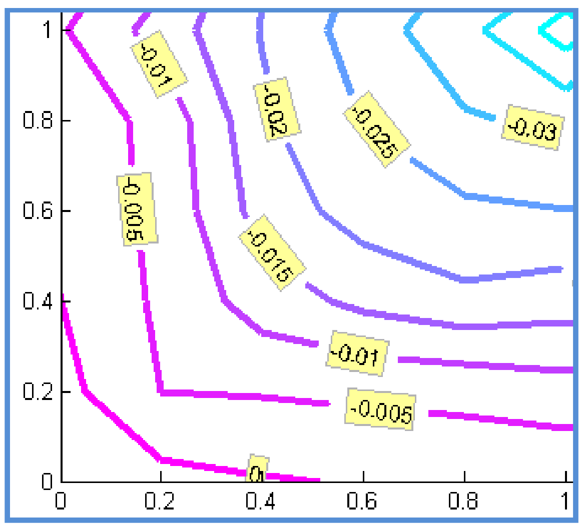
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio (CMR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.833	0.89	0.878	0.8
ULP	0.902	0.903	0.87	0.889



(i) Deformed Shape for F_C_F_C Plate

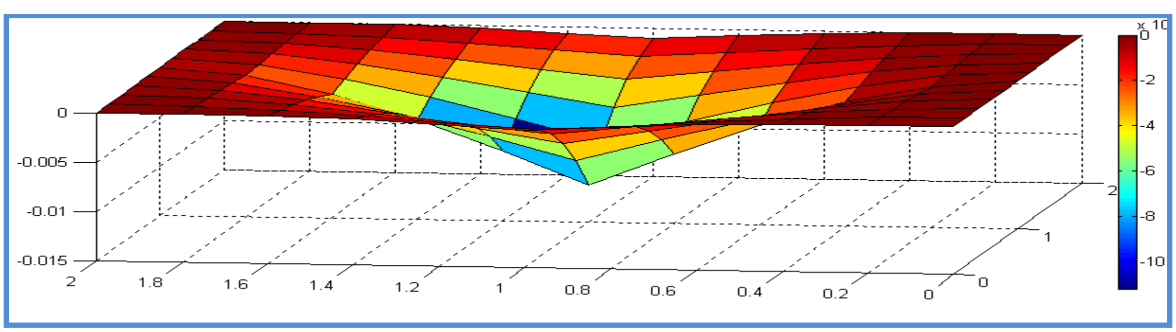


(ii) Mxx Contours

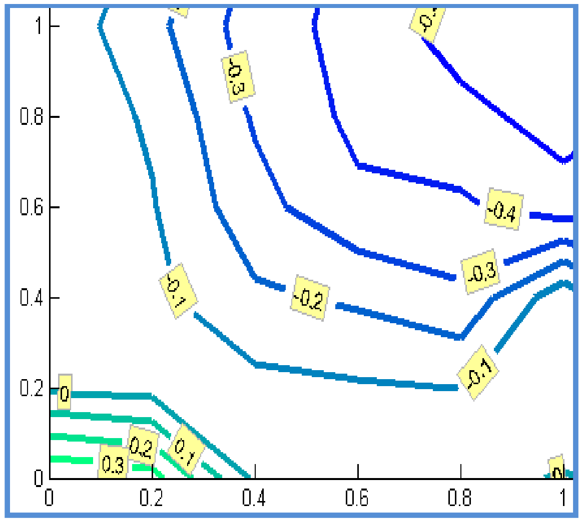


(iii) Myy Contours

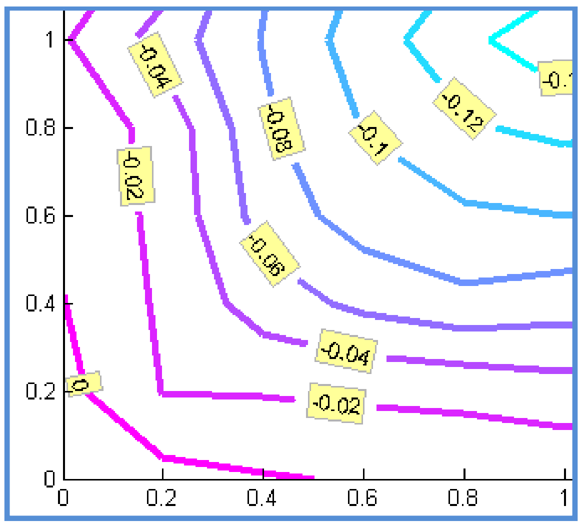
Fig 11.16 Various Plots for F_C_F_C Square Plate Under CPL



(i) Deformed Shape for F_C_F_C Plate



(ii) Mxx Contours



(iii) Myy Contours

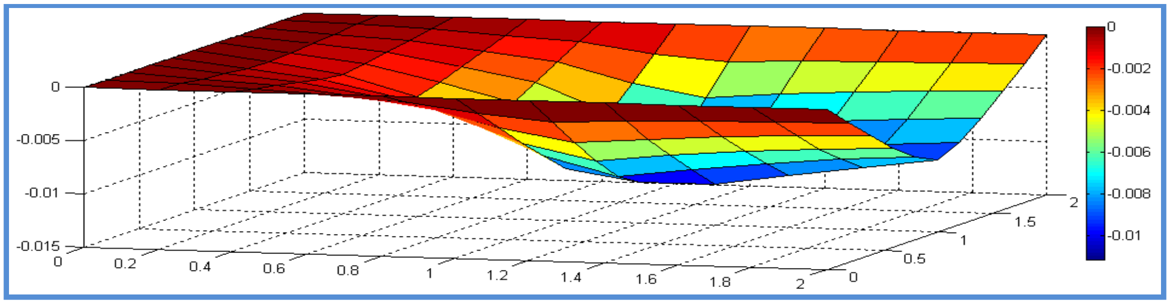
Fig 11.17 Various Plots for F_C_F_C Square Plate Under ULP

11.9 S_F_S_C ORTHOTROPIC SQUARE PLATE EXAMPLE

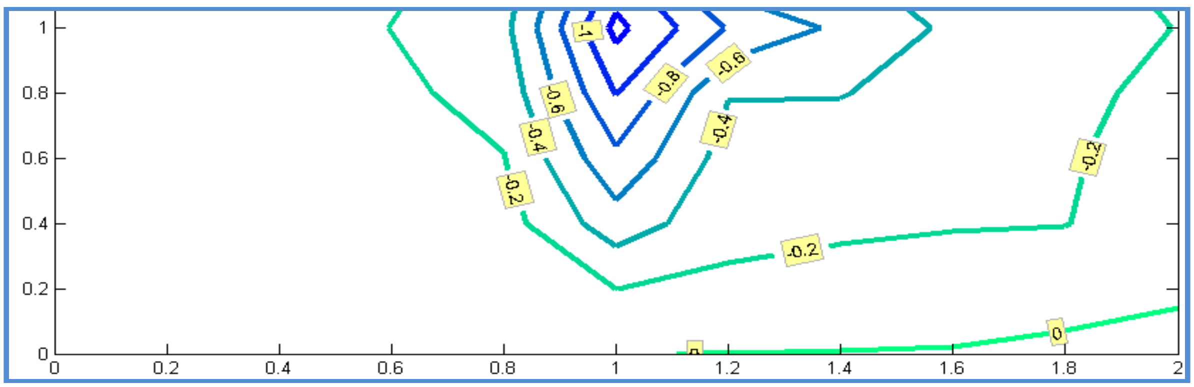
With AB and CD edges as simply supported, BC as free and AD as clamped, considering one way symmetry, half of the plate (1m x 2m) is divided into 50 elements with 5 x 10 discretization scheme. Based on the IFM based results 3D deformed shape and 2D moment contour plots are included here in **Figs. 11.18** and **11.19,** whereas moment and displacement values obtained at the centre of plate are reported in **Table 11.8**.

Table 11.8 Comparison of Results

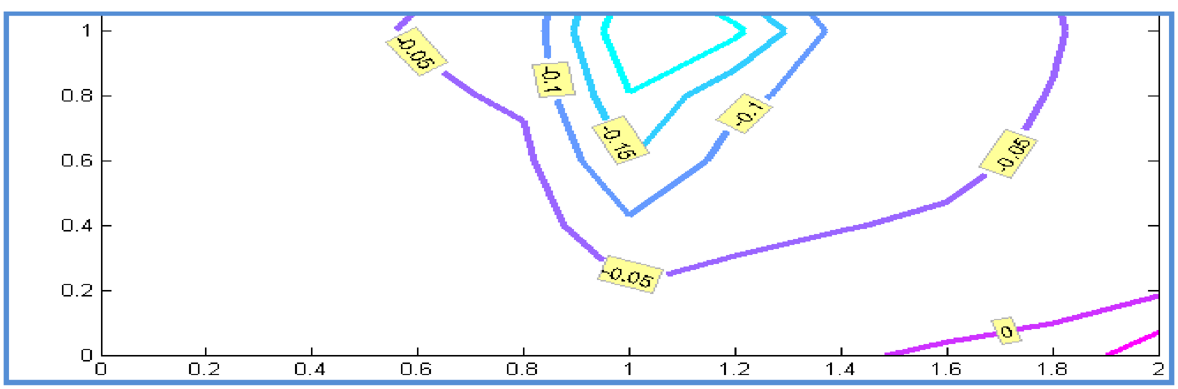
Type of Loading	Central Deflection Ratio (CDR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$	Central Moment Ratio (CMR) $\delta_{E\,(IFM)}/\delta_{E\,(EXACT)}$		
		M_x	M_y	M_{xy}
CPL	0.896	0.904	0.855	0.831
ULP	0.907	0.955	0.911	0.885



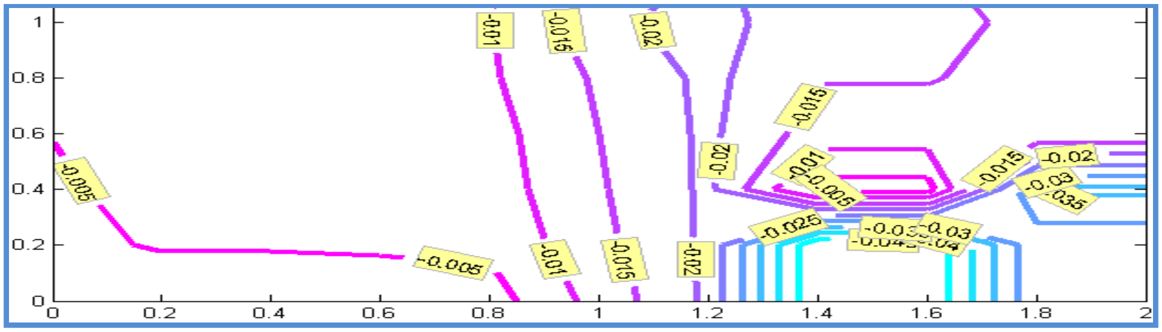
(i) Deformed Shape for S_F_S_C Plate



(ii) Mxx Contours

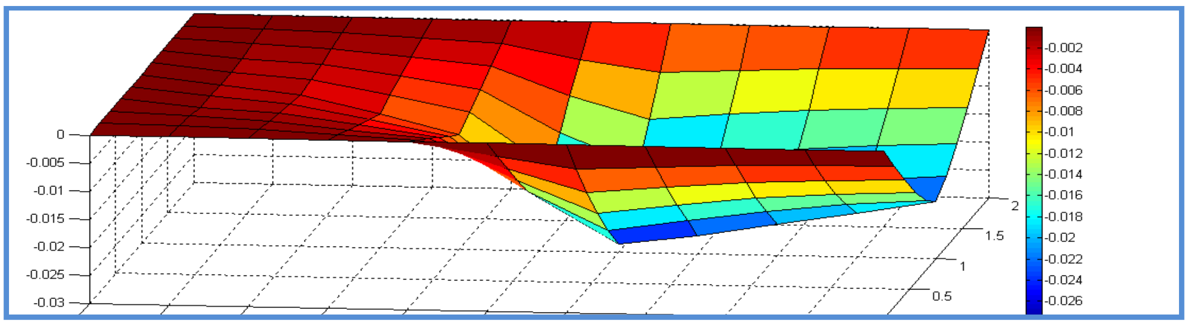


(iii) Myy Contours

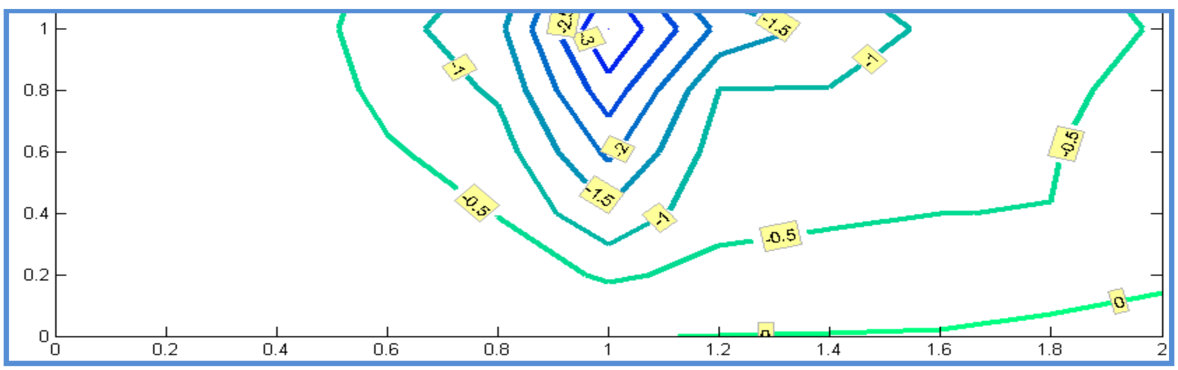


(iv) Mxy Plot

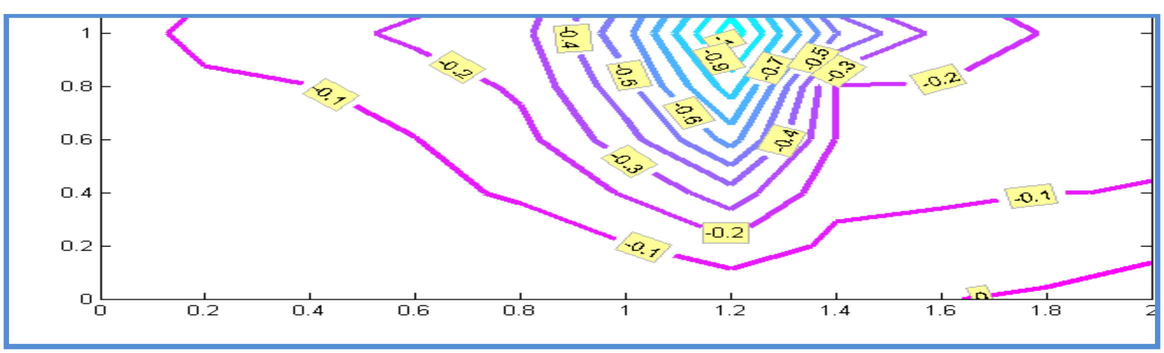
Fig 11.17 Various Plots for S_F_S_C Square Plate Under CPL



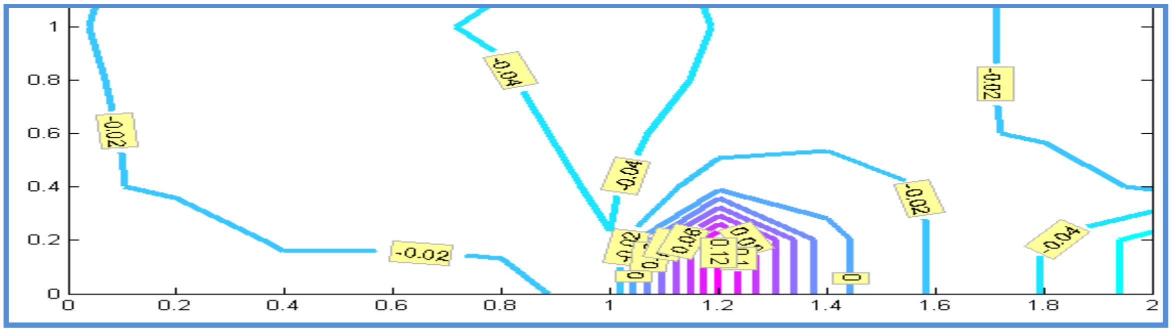
(i) Deformed Shape for S_F_S_C Plate



(ii) Mxx Contours



(iii) Myy Contours



(iv) Mxy Contours

Fig 11.18 Various Plots for S_F_S_C Square Plate Under ULP

11.10 RECTANGULAR ORTHOTROPIC PLATE EXAMPLES

Linear static analysis is also carried of GFRP rectangular plate bending problems by considering aspect ratio (AR) as 1.5 and 2 with plan dimensions as 2m x 3m and 2m x 4m respectively. Analysis is to be carried out under central point loading (CPL) as well as under uniform intensity of lateral pressure (ULP) using RECT_9F_12D IFM based element. Results obtained for central deflection and moments at the centre of the plate for simply supported and clamped plate orthotropic plates are reported here in Tables **11.9** and **11.10** respectively.

Table 11.9 Results for Simply Supported Rectangular Orthotropic Plate

LOAD	Aspect Ratio (AR)	Central Deflection Ratio (CDR)	Central Moment Ratio (CMR)		
			M_x	M_y	M_{xy}
CPL	1.5	0.903	0.854	0.853	0.884
	2.0	0.787	0.827	0.843	0.855
ULP	1.5	0.878	0.933	0.996	0.931
	2.0	0.889	0.917	0.993	0.876

Table 11.10 Results for Clamped Rectangular Orthotropic Plate

LOAD	Aspect Ratio (AR)	Central Deflection Ratio (CDR)	Central Moment Ratio (CMR)		
			M_x	M_y	M_{xy}
CPL	1.5	0.932	0.861	0.897	0.957
	2.0	0.872	0.752	0.904	0.875
ULP	1.5	0.931	0.917	0.95	0.811
	2.0	0.893	0.884	0.82	0.808

11.11 AN EXAMPLE OF STRUCTURAL ORTHOTROPY

A RCC slab with equi-distance rib along one direction is shown in **Fig. 11.9.** It is analysed here as simply supported orthotropic plate problem of 4000 mm x 4000 mm size with thickness of rib is 500 mm. Static analysis is carried out by considering dual symmetry and discretizing the bottom left quadrant into 2 x 2 mesh under Central point load (CPL) of 10 kN. The slab is considered with properties: $E_{conc} = 2.23 \times 10^4$ N/mm^2, and $\nu = 0.24$.

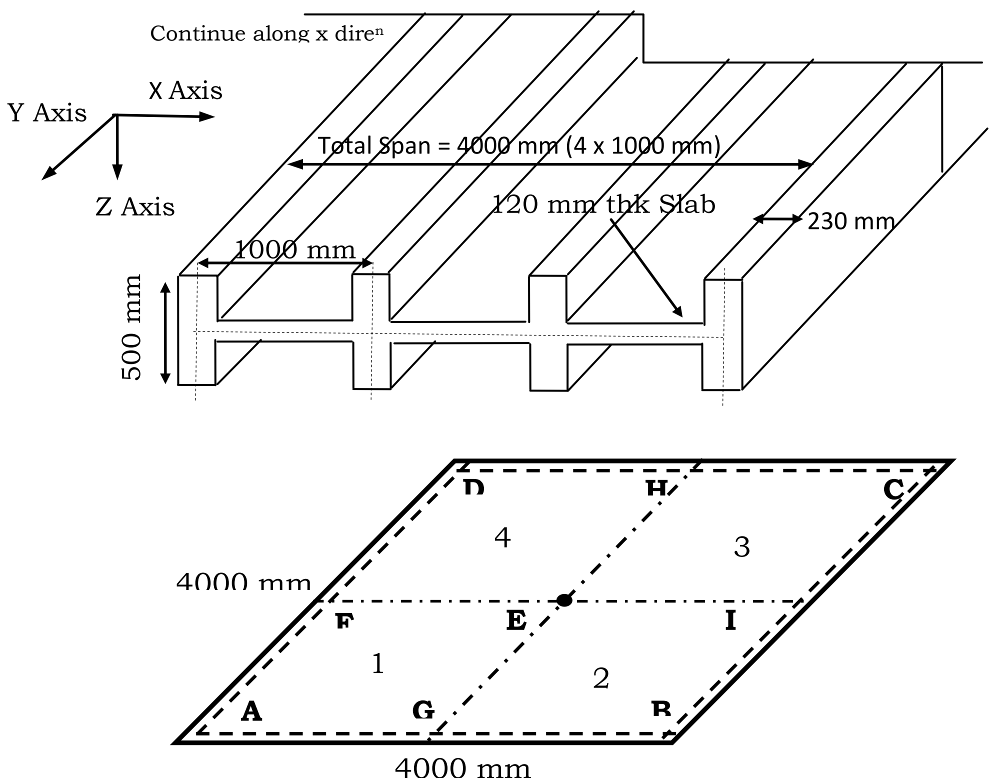


Fig. 11.19 RCC Slab Bending Example

All the necessary matrices are same, except the global flexibility matrix [G] which is of size 36 x 36 where [D] matrix of size 3 x 3 is replaced by [D_{ortho}] for structural orthotropic material. For the given slab problem material matrix [D_{ortho}] is calculated as follows:

1. The effective width (b_f) is calculated based on RCC T-beam theory as

$$b_f = \frac{L_o}{12} + b_w + 3d = 925\text{mm (approximately)}$$

where, L_o = 4000mm which is span of slab, bw = 230 mm which is width of RCC rib and d = 120 mm which is depth of slab.

2. The flexural rigidity (D_x) along x-x direction of slab (neglecting rib contribution) is calculated as per plate formula based on the elastic and geometrical properties (E, ν and d) of RCC slab and is equal to 3.4074 x 10^{9} N-mm^2.
3. The flexural rigidity (D_y) along y-y direction including rib and reinforcement contribution of approximate area (750mm^2) on both side of slab is calculated as follows

$$D_y = D_x + \frac{EI_{yy}}{h} = 5.6411 \times 10^{13} \text{ N-mm}^2.$$

4. The average torsional rigidity (D_{xy}) is worked out as 9.06 x 10^{11} N -mm^2.

Using the above rigidities, one can develop moment curvature relations from which by inverting finally [D_{ortho}] is worked out which is as follows.

$$[D_{ortho}] = \begin{bmatrix} 0.3114\,x\ 10^{-09} & -0.0747\,x\ 10^{-09} & 0 \\ -4.517\,x\ 10^{-15} & 1.88\,x\,10^{-14} & 0 \\ 0 & 0 & 0.006\,x10^{-09} \end{bmatrix} \quad \dots (11.3)$$

Now the element flexibility matrix [G_e] is worked out by the formula as

$$[G_e] = \int_{-a}^{+a}\int_{-b}^{+b}[Y]^T[D_{ortho}][Y]dxdy \quad \dots (11.4)$$

where [Y] is the stress interpolation matrix for rectangular plane stress element of size (3 x 9)

Step 0 - Solution strategy: A problem of square plate with simply supported boundary condition is divided into four quadrants 1, 2, 3 and 4. The quadrant 1 is descretized such as 11, 12, 13 and 14 in anticlockwise direction as depicted in **Fig. 11.20.** With 4 element discretization of quarter plate, it has total 12 displacement degrees of freedom ($\theta_1, \theta_2 \ldots\ldots \delta_5 \ldots\ldots \theta_{12}$). IFM based RECT_9F_12D FE element is used.

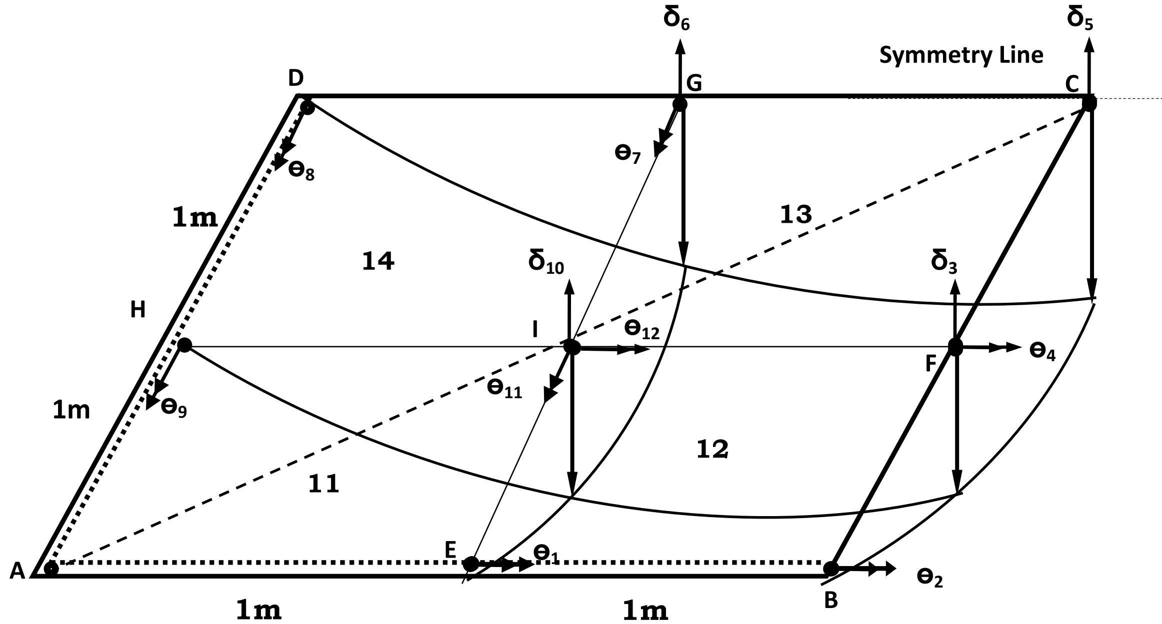


Fig.11.20 2 x 2 Discretization of SS Orthotropic Plate

Step 1-Development of elemental matrices: The element equilibrium matrix [Be] and elemental flexibility matrix [Ge] are calculated by substituting values of a and b as 0.5m. Thus,

$$[Be] = \begin{bmatrix} 0 & 0.5 & 0 & -0.083 & 0 & 0 & 0.5 & -0.0833 & 2 \\ -0.5 & 0.25 & 0.0833 & -0.04167 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -0.5 & 0.0833 & 0.25 & -0.04167 & 0 \\ 0 & -0.5 & 0 & 0.0083 & 0 & 0 & 0.5 & 0.0833 & -2 \\ 0.5 & 0.25 & -0.083 & -0.0416 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -0.5 & -0.083 & 0.25 & 0.04167 & 0 \\ 0 & -0.5 & 0 & -0.0833 & 0 & 0 & -0.5 & -0.0833 & 2 \\ 0.5 & 0.25 & 0.083 & -0.0416 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5 & 0.0833 & 0.25 & -0.4167 & 0 \\ 0 & 0.5 & 0 & 0.0833 & 0 & 0 & -0.5 & 0.0833 & -2 \\ -0.5 & 0.25 & -0.083 & 0.04167 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0.5 & -0.083 & 0.25 & -0.04167 & 0 \end{bmatrix}$$

The nonzero components for [Ge_1] matrix are as follows

$G_{e(1,1)} = 3.114 \times 10^{-10}$, $G_{e(2,2)} = 0.26 \times 10^{-10}$, $G_{e(4,4)} = G_{e(3,3)} = G_{e(2,2)}$

$G_{e(5,5)} = 1.8 \times 10^{-14}$, $G_{e(6,6)} = 1.5 \times 10^{-15}$, $G_{e(7,7)} = G_{e(6,6)}$,

$G_{e(8,8)} = 1.3 \times 10^{-16}$, $G_{e(9,9)} = 6.0 \times 10^{-13}$, $G_{e(1,5)} = -0.18 \times 10^{-10}$

$G_{e(2,6)} = -0.062 \times 10^{-10}$, $G_{e(3,7)} = -0.062 \times 10^{-10}$,$G_{e(4,8)} = -5.18 \times 10^{-13}$

$G_{e(5,1)} = -4.514 \times 10^{-15}$, $G_{e(6,2)} = -3.77 \times 10^{-16}$, $G_{e(7,3)} = G_{e(6,2)}$ & $G_{e(8,4)} = G_{e(6,2)}$

Step 2- Development of Global Matrices: The compatibility matrix for the four elements is obtained from the displacement deformation relations (DDR) i.e. $\beta = [B]^T\{\delta\}$. In the DDR, 36 deformations which correspond to 36 force variables are expressed in terms of 12 displacements. The problem requires 24 compatibility conditions [C] that are obtained by eliminating the 12 displacements from the 36 DDRs. These are generated by using auto-generated matlab based computer program by giving input as upper part of the global equilibrium matrix [B]. The global flexibility matrix for the problem is obtained by diagonal concatenation of the four elemental flexibility matrices as

$$[G] = \begin{bmatrix} Ge_1 & & & \\ & Ge_2 & & \\ & & Ge_3 & \\ & & & Ge_4 \end{bmatrix}$$

Where, all the above sub matrices are calculated as described in Step 1.

Step 3 Calculations of Forces {F}: The internal forces in element 13 are obtained by using Mat Lab's inverting procedure, which are found as follows

Table 9.9 Internal Forces in Element 13

F_{19}	F_{20}	F_{21}	F_{22}	F_{23}	F_{24}	F_{25}	F_{26}	F_{27}
1012	2871	2596	4364	13021	17673	10218	21736	-640

After substituting the values of internal unknowns in moment equations, one can calculate moments at the corner I by substituting values of coordinates considering origin of coordinate at the centre of the element as shown in **Fig. 11.21**.

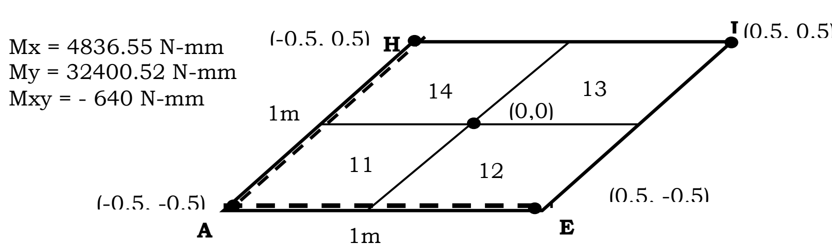


Fig. 11.21 Sub-Elements of Element Number 1

Finally, the nodal displacements for the quarter plate are found as follows:

Table 11.12 Nodal Displacements x 10^{-07} mm or radians

θ_1	θ_2	δ_3	θ_4	δ_5	δ_6	θ_7	θ_8	θ_9	δ_{10}	θ_{11}	θ_{12}
-1.2	-50	-47	-4	-94.1	-65	40	50	20	-52.2	-3.2	-1.1

11.12 DISCUSSION OF RESULTS

- In all the plate examples, the deflection at the centre of plate is found less in case of central point loading compared to that of uniform lateral pressure. It is because of CPL is considered as 10 kN whereas ULP is considered as 10 kN/m^2. However, the effect of concentrated load can be clearly seen in the deflection profile by the rate of change of deflection near the centre of the plate irrespective of the boundary conditions.
- Moment contours included for Mx, My and Mxy also indicates higher values for plate subjected to uniform lateral pressure compared to concentrated point load acting at the centre of the plate.
- In all the orthotropic plate examples, the values of Mxx is found greater than Myy which clearly indicates the effect of higher modulus in the x direction (Ex = 40 kN/m^2) compared to the modulus of elasticity in y direction (Ey = 8 kN/m^2).
- A variety of boundary conditions could be easily handled by integrated force based methodology. The effect of change of boundary conditions could be easily visualized through the deflection profiles provide for fully simply supported and fully clamped plates. The impact of making one of the edges of the as free also could be easily visualized.
- Moment contours drawn based on the values of moments obtained by using IFM also clearly indicated the effect of the boundary conditions on the analysis results. It may be noted here that in IFM bilinear variation of moments is assumed.
- From the results provide for central deflection and moments at the centre of all the plates it is clear that the result obtained for the case of

ULP are more nearer to the exact solution compared to the results obtained in case of CPL.

- The warping of the plate and changing from sagging moment to hogging moments could also be easily visualized from the moment contours included for the square plate examples.
- Fully simply supported and fully clamped boundary condition plate problems analysed by changing the plate aspect ratio from 1.0 to 1.5 and 2.0 also clearly indicated the effect of change of aspect ratio on the central deflection and on the values of moments at the centre of the orthotropic plate.

CHAPTER 12

DYNAMIC ANALYSIS OF ORTHOTROPIC PLATE PROBLEMS

12.1 STRATEGY ADOPTED

Frequency analysis is carried out in the present chapter of GFRP (Glass Fiber Reinforced Plastic) orthotropic plate. Total five problems are considered here to demonstrate the applicability of IFM based formulation under different boundary conditions. The formulation of different matrices, development of software, strategy adopted for finding solutions using IFM, use of Matlab software etc are already discussed in detail in the previous chapters. Either 5 x 5 or 10 x 5 discretization is used depending upto the type of symmetry exhibited by the problem. Only first four modes are considered for the development of deformation pattern for each problem using IFM based eigen value analysis. Using first modal frequency value, the internal unknowns (F_1, F_2.....F_n) are taken for further calculations based on the number of elements. Using these values internal moments are calculated using stress polynomials by substituting the corresponding the coordinate values for each element. Normalized moments are worked out and are reported in respective tables. Nodal displacements are also worked out for first mode of vibration. After normalizing the value with respect to value at the centre of plate, other values are worked out and are reported in tabular form for all the examples included in this chapter. Steps required for finding the solution are explained here with reference to an example given in next section.

12.2 DYNAMIC ANALYSIS OF ORTHOTROPIC PLATE

A simply supported GFRP square plate having dimensions as 2m x 2m x 0.005m is shown in **Fig. 12.1.** Frequency analysis is carried out by considering dual symmetry and discretizing the bottom left most quadrant into 5 x 5 discretization pattern (**Fig. 12.2**) by considering lumped mass criteria. Considering E_x, E_y and G_{xy} as 40kN/mm^2, 8 kN/mm^2 and 4 kN/mm^2

respectively ν_x as 0.25 and material density for the plate as 1850 kg/m^3, results are obtained and summarized in terms of Frequency value and are comparing the values with the exact solution available in literature **[99]**. Normalized moments, and nodal displacements are also worked out with respect to the centre of plate.

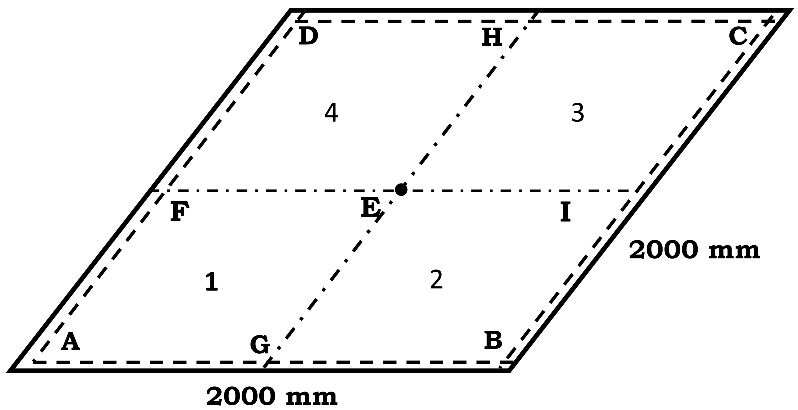


Fig. 12.1 A Square Plate Bending Example

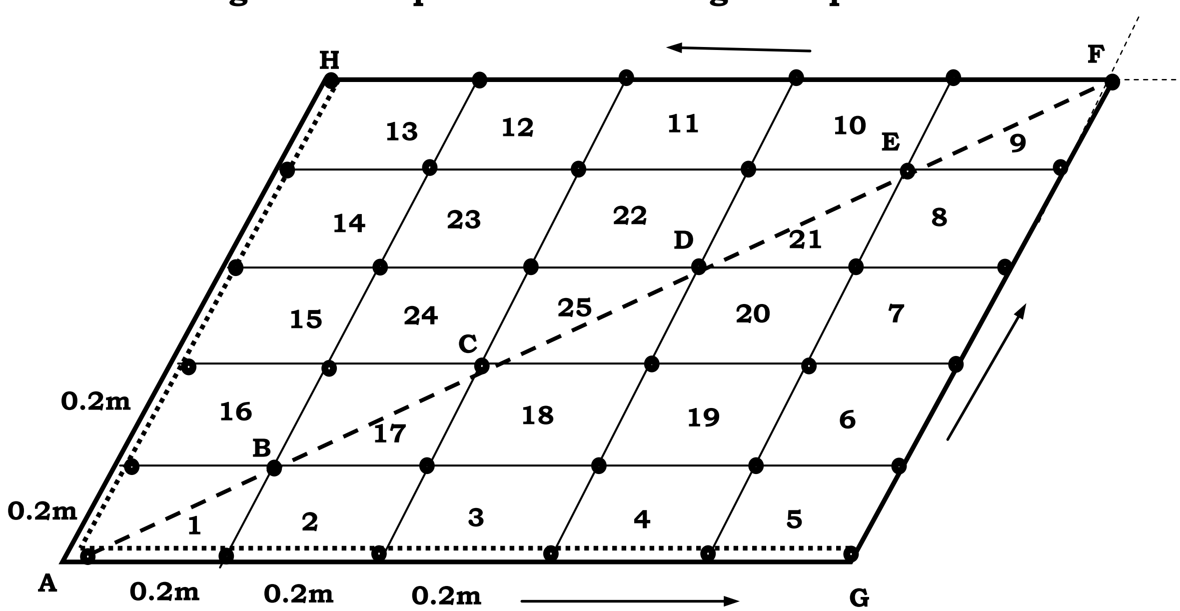


Fig. 12.2 Discretization Scheme (5 x 5)

Step 0 – Solution strategy: Due to two way-symmetry, the quarter of the simply supported plate is discretized into 5 x 5 meshing using RECT_9F_12D elements. After discretization the complete problem has has total 225 force unknowns and 75 displacement degrees of freedom.

Step 1- Development of Elemental Matrices: Different matrices such as elemental equilibrium matrix [Be] of size 12 x 9, elemental flexibility matrix [Ge] of size 9 x 9, Lumped mass matrix [M_L] of size 12 x 12 are developed by using the procedure given in earlier chapters by substituting values of a and b as 0.1m considering mass per unit area $\bar{m}$ equals to 9.25 kg/m^2. Depending upon tributary area at each node, the mass is calculated corresponding to the lateral and rotational degree of freedom using finite element based grid analogy technique.

Step 2 - Development of Global Matrices: The assembled equilibrium matrix [B] will be of size 75 x 225. The global flexibility matrix [G] is diagonally concatenated and is of size 225 x 225. Total 150 compatibility conditions are developed by using the Matlab based file named as "mtechexamplemod (B)". The multiplication of compatibility matrix and global flexibility matrix [G] gives the global compatibility matrix named as [CCmatrix]. Thus, concatenation of [CCmatrix] after normalizing with respect to maximum number of [B] matrix gives a global equilibrium matrix [S], which is basically a square matrix. The global lumped mass matrix is developed by concatenating necessary rows and columns of zeros from right and bottom side so as to have a square matrix of 225 x 225 size.

Step 3 – Calculation of Natural Frequencies: The frequency vector is worked out by carrying out eigen value analysis using two major matrices i.e. [S] which is the global equilibrium matrix and MJG which is a product of Mass matrix [M], [Jmatrix] and [G] matrix which is the global flexibility matrix. Using Matlab command window by typing `[Fmatrix, Freq] = eig(S, MJG)` further work is carried out.

Step 4 – Calculation of normalized moments and nodal displacements: The internal unknowns of [Fmatrix] are auto-calculated by substituting each frequency value in IFM based eigen equation. By taking each element and its coordinates all the internal moments are calculated. Then along a diagonal line of **(Fig. 12.2)** all the moment values are normalized with respect to unit value at point F. The nodal displacement matrix [Disp] of size 75 x 225 is worked out and is normalized with respect to unit value at the centre of the plate. **Table 12.1** shows the IFM based natural frequency and exact solution **[99]** available for the first four mode.

Table 12.1 Comparison of Frequency

Frequency Number	$\omega_{IFM(L)}$ (rad/sec)	ω_{EXACT}(rad/sec)
First	0.0190	0.0197
Second	0.0334	0.0372
Third	0.0648	0.0686
Fourth	0.0742	0.0788

Table 12.2 shows the normalized nodal moments and deflections corresponding to unit value at the centre of plate for first frequency value.

Table 12.2 Normalized Nodal Moments and Deflections

Point	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.000
E	0.655	0.422	1.015	0.855
D	0.591	0.398	1.255	0.732
C	0.428	0.211	2.055	0.4239
B	0.202	0.102	2.633	0.287
A	0.000	0.00	2.8321	0.000

From the results obtained for 1st quadrant, results are obtained for the three remaining quadrants using concept of symmetry. Finally, deformed shapes drawn for the first four frequencies using the Matlab software are depicted in **Figs. 12.3-12.6.**

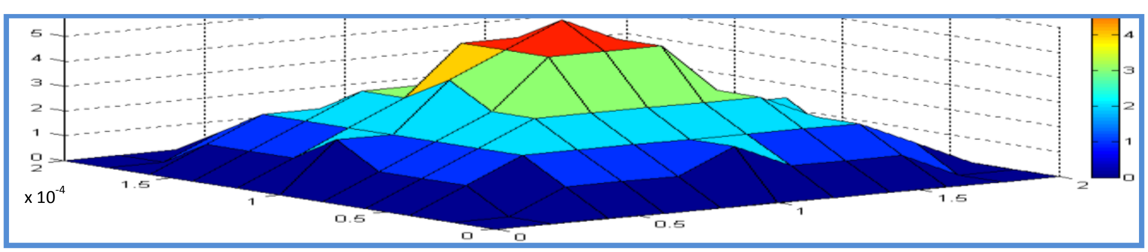


Fig. 12.3 First Mode Deformation Pattern (ω_{11}) for SS Plate

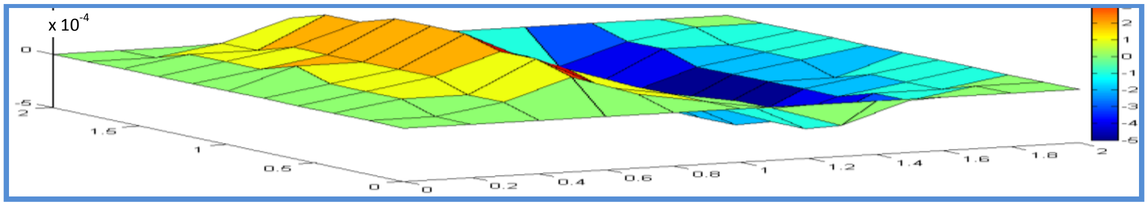


Fig. 12.4 Second Mode Deformation Pattern (ω_{12}) SS Plate

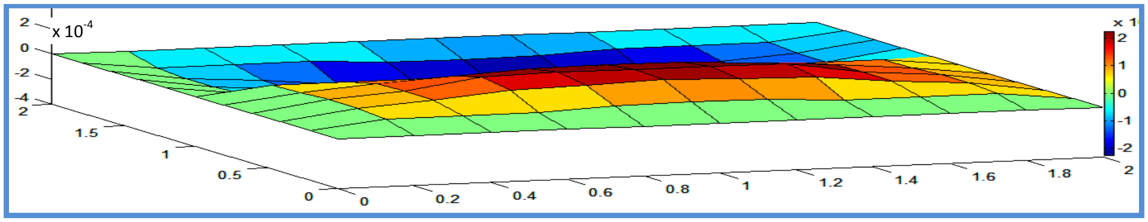


Fig. 12.5 Third Mode Deformation Pattern (ω_{21}) SS Plate

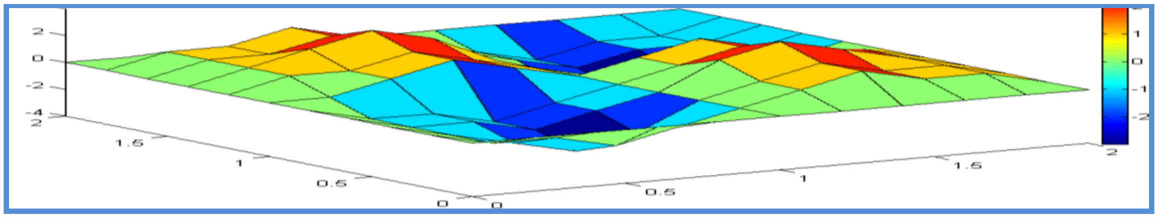


Fig 12.6 Fourth Mode Deformation Pattern (ω_{22}) SS Plate

12.3 DYNAMIC ANALYSIS OF S_C_S_C ORTHOTROPIC PLATE

For a square plate with two opposite edges AB and CD simply supported and other two BC and AD as clamped, frequency analysis is carried out by considering dual symmetry and discretizing the left most quadrant into 5 x 5 mesh by considering lumped and consistent mass criteria. Following the same steps, total five displacement degrees of freedom for the clamped edges are restrained while five displacement degrees of freedom are made free along simply supported edges. As per these changes all the major and minor matrices are modified. **Table 12.3** shows the IFM based natural frequency and exact solution available for the first four modes.

Table 12.3 Comparison of Frequency

Mode Number	$\omega_{\text{IFM(L)}}$ (rad/sec)	ω_{EXACT}(rad/sec)
First	0.342	0.0393
Second	0.045	0.0508
Third	0.095	0.1056
Fourth	0.1074	0.1129

Table 12.4 shows the normalized nodal moments and deflections corresponding to unit value at the centre of the plate under the first mode of vibration. Also, deformation pattern are drawn for the first four modes as depicted in **Figs. 12.7 – 12.10**.

Table 12.4 Normalized Moments and Deflections

Point	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.00
E	0.833	0.553	0.722	0.932
D	0.621	0.421	0.832	0.721
C	0.321	0.198	0.733	0.419
B	0.103	0.003	0.421	0.311
A	-0.083	-0.002	0.188	0.000

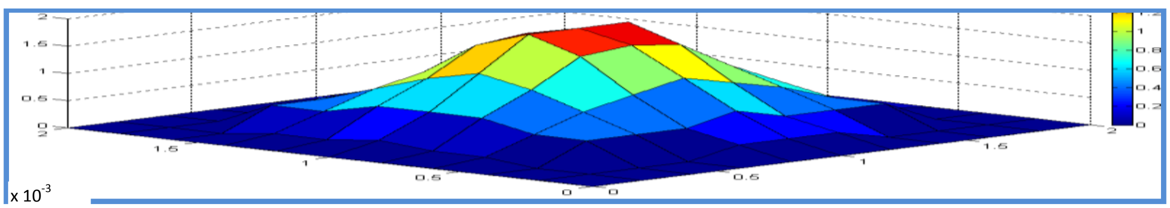


Fig. 12.7 First Mode Deformation Pattern (ω_{11}) for S_C_S_C Plate

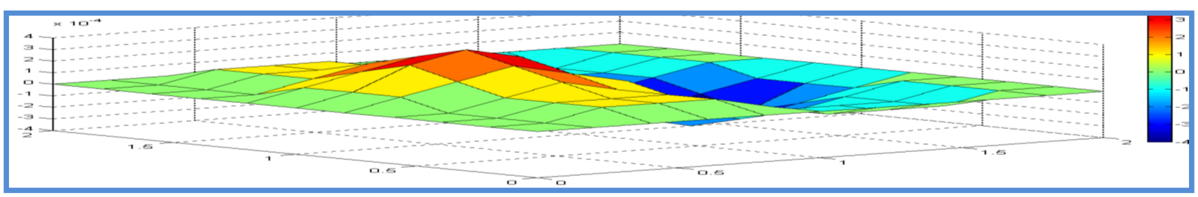


Fig. 12.8 Second Mode Deformation Pattern (ω_{12}) for S_C_S_C Plate

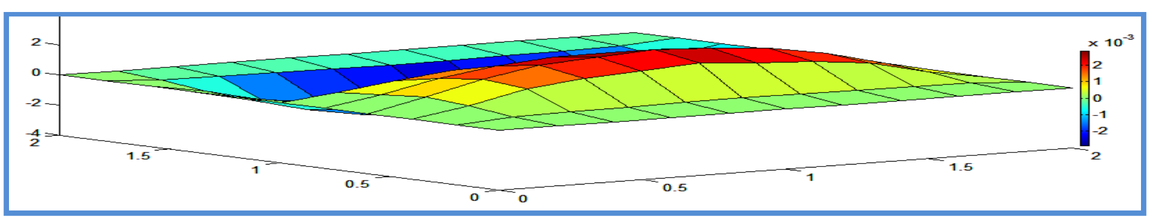


Fig. 12.9 Third Mode Deformation Pattern (ω_{21}) for S_C_S_C Plate

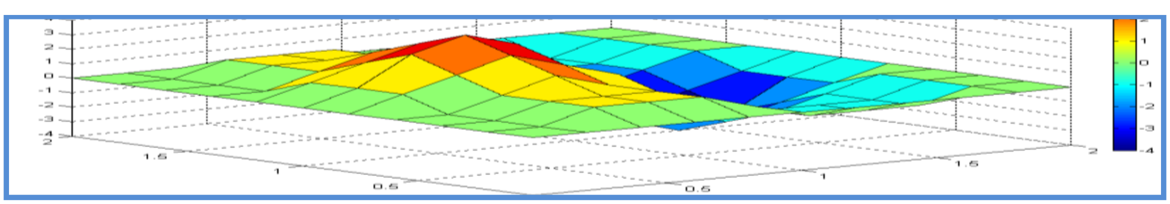


Fig 12.10 Fourth Mode Deformation Pattern (ω_{22}) for S_C_S_C Plate

12.4 DYNAMIC ANALYSIS OF S_S_S_C ORTHOTROPIC PLATE

A square plate with three edges AB, BC and CD simply supported and AD as clamped is now considered for the frequency analysis by considering one way symmetry and discretizing the bottom half portion in 10 x 5 mesh by considering a lumped mass criteria. **Table 12.5** shows the Frequency Ratio first four modes under lumped mass criterion, where as **Table 12.6** shows the normalized nodal moments and deflections for the first mode of vibration. Deformed shapes are drawn for the first four natural frequencies, using Matlab facility, which are depicted in **Figs. 12.11-12.14.**

Table 12.5 Frequency Ratio for S_S_S_C Plate

Mode Number	$\frac{\omega_{IFM (L)}}{\omega_{EXACT}}$
First	0.8213
Second	0.8677
Third	0.889
Fourth	0.9015

Table 12.6 Normalized Moments and Deflections

Points	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.00
E	0.855	0.622	1.012	0.855
D	0.642	0.532	0.944	0.743
C	0.422	0.322	0.924	0.488
B	0.174	0.109	0.988	0.215
A	-0.012	0.000	0.943	0.000

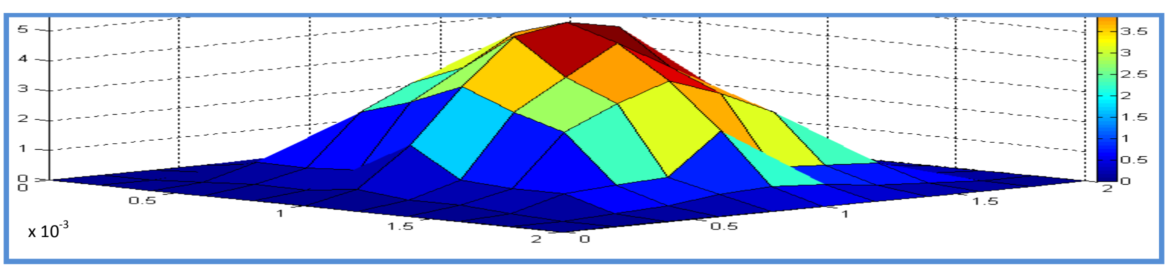


Fig. 12.11 First Mode Deformation Pattern (ω_{11}) for S_S_S_C Plate

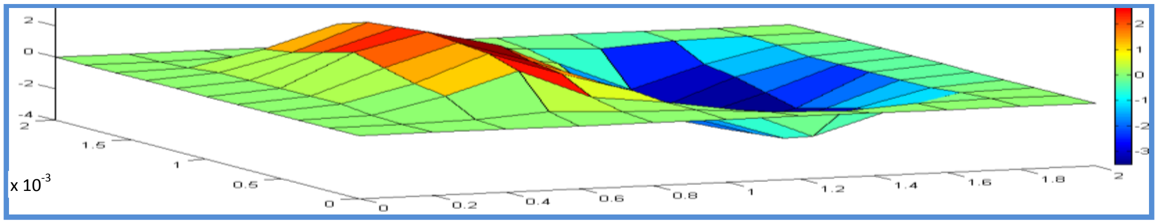


Fig. 12.12 Second Mode Deformation Pattern (ω_{12}) for S_S_S_C Plate

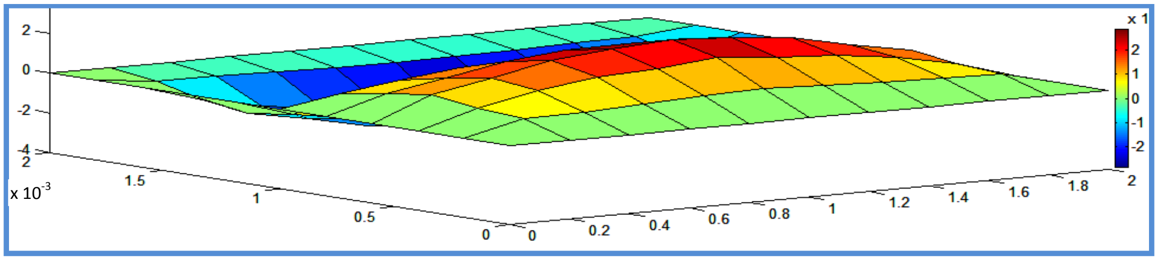


Fig. 12.13 Third Mode Deformation Pattern (ω_{21}) for S_S_S_C Plate

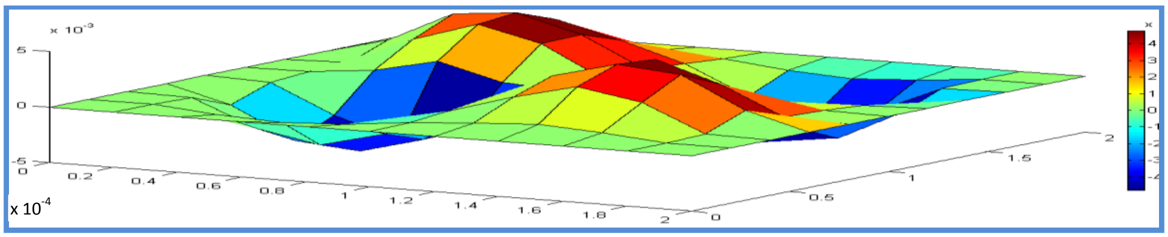


Fig. 12.14 Fourth Mode Deformation Pattern (ω_{22}) for S_S_S_C Plate

12.5 DYNAMIC ANALYSIS OF C_C_C_S ORTHOTROPIC PLATE

Frequency analysis of square plate with three edges AB, BC and CD clamped and AD simply supported is now carried out by considering one way symmetry and dividing bottom half portion into 10 x 5 mesh by considering a lumped mass criteria. The results are summarized in **Tables 12.7** and **12.8** and deformation plots are included in **Figs. 12.15 -12.18.**

Table 12.7 Frequency Ratio for C_C_C_S Plate

Mode Number	$\frac{\omega_{IFM (L)}}{\omega_{EXACT}}$
First	0.8891
Second	0.895
Third	0.9092
Fourth	0.9177

Table 12.8 Normalized Moments and Deflections

Point	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.00
E	0.844	0.488	1.017	0.811
D	0.605	0.311	0.885	0.736
C	0.388	0.288	0.633	0.461
B	0.211	0.102	0.322	0.204
A	-0.04	-0.01	0.022	0.000

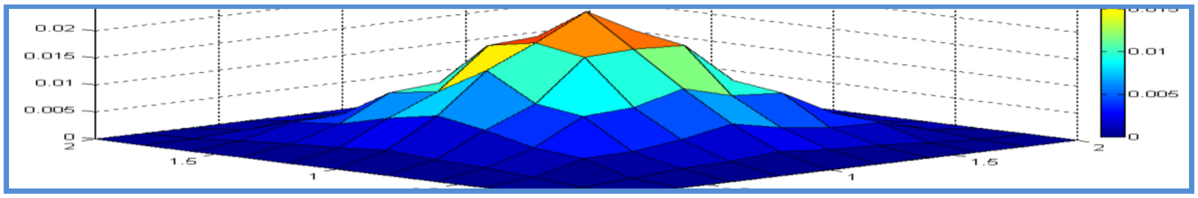


Fig. 12.15 First Mode Deformation Pattern (ω_{11}) for C_C_C_S Plate

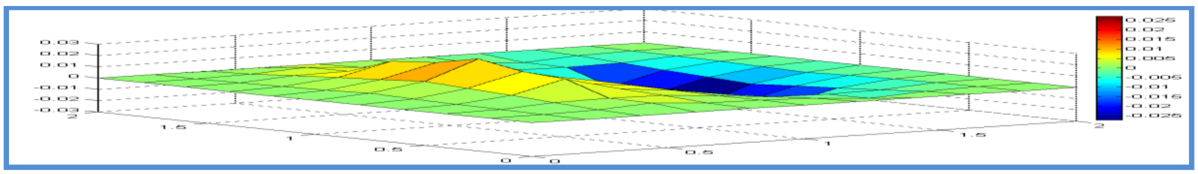


Fig. 12.16 Second Mode Deformation Pattern (ω_{12}) for C_C_C_S Plate

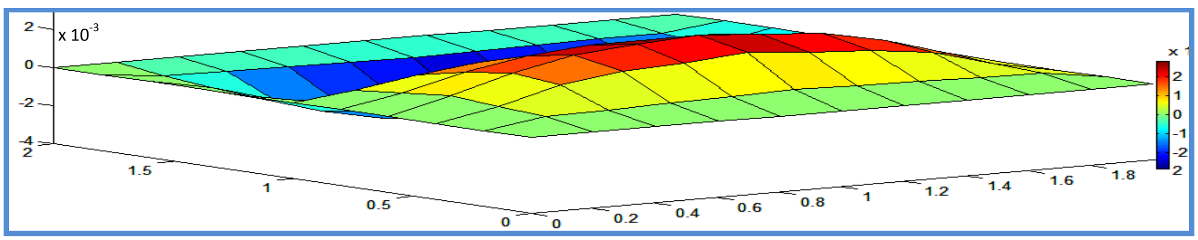


Fig. 12.17 Third Mode Deformation Pattern (ω_{21}) for C_C_C_S Plate

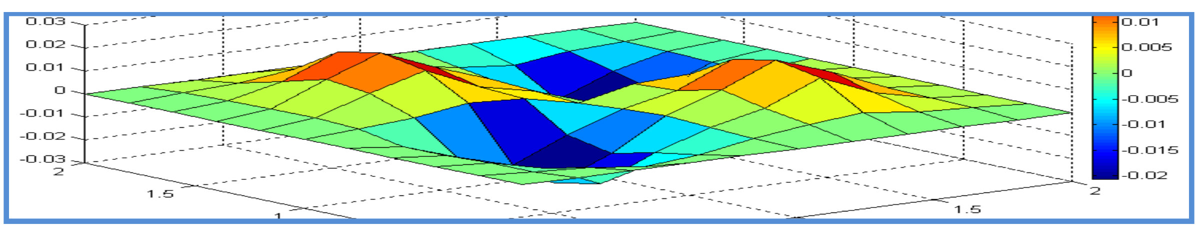


Fig. 12.18 Fourth Mode Deformation Pattern (ω_{22}) for C_C_C_S Plate

12.6 DYNAMIC ANALYSIS OF ORTHOTROPIC PLATE

The last example considered here is that of fully clamped square plate. Frequency analysis is carried out by considering dual symmetry and discretizing the bottom left quadrant into 5 x 5 mesh by considering lumped mass criteria. The results are summarized for Frequency Ratio in **Table 12.9** and for normalized moments, and nodal displacements in **Table 12.10.** Deformation patterns for first four modes are depicted in **Figs. 12.19-12.22**.

Table 12.9 Frequency Ratio for Clamped Plate

Mode Number	$\frac{\omega_{IFM\ (L)}}{\omega_{EXACT}}$
First	0.9022
Third	0.9133
Fourth	0.9033
Fourth	0.9155

Table 12.10 Normalized Nodal Moments and Deflections

Point	M_x	M_y	M_{xy}	Vertical Deflection
F	1.00	1.00	1.00	1.00
E	0.533	0.422	0.421	0.722
D	0.329	0.244	0.231	0.514
C	0.166	0.105	0.188	0.311
B	0.098	0.084	0.091	0.094
A	-0.114	-0.094	0.011	0.000

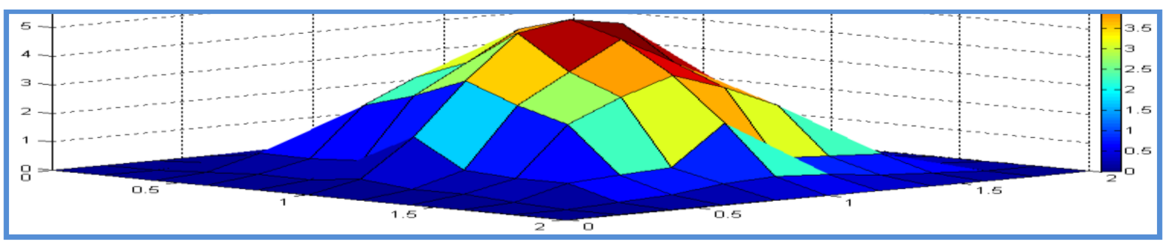


Fig. 12.19 First Mode Deformation Pattern (ω_{11}) for Clamped Plate

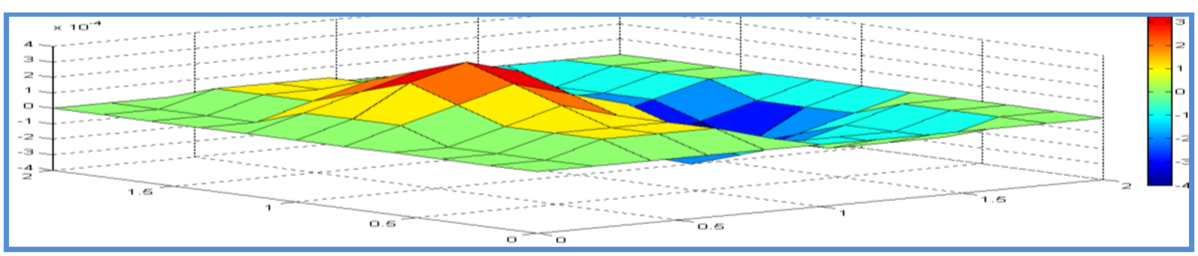


Fig. 12.20 Third Mode Deformation Pattern (ω_{12}) for Clamped Plate

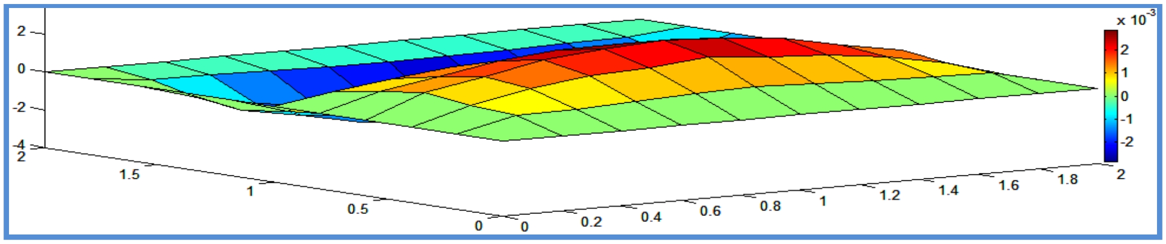


Fig. 12.21 Third Mode Deformation Pattern (ω_{21}) for Clamped Plate

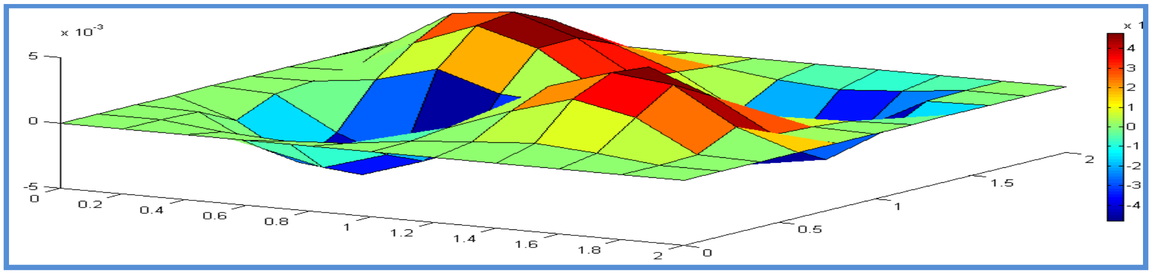


Fig. 12.22 Fourth Mode Deformation Pattern (ω_{22}) for Clamped Plate

12.7 DISCUSSION OF RESULTS

- In case of simply supported orthotropic plate, out of total 75 displacement degrees of freedom (one for each frequency) only first four frequencies are considered for comparison purpose. Frequency values are found to differ 4% to 10% from the exact solution. The nodal displacements are normalized with respect to the value at the centre of plate for the first vibration. The maximum value is found at the centre and zero value at the edges with variation as per deflection curve. The moment values along x-x direction (Mxx) are found higher than the y-y direction (Myy). The warping moment (Mxy) is found to have higher values at edges due to torsional curvature. The mode shapes drawn using Matlab module show that the first mode has maximum convex or concave deflected pattern. Because of the variation in elastic properties in x and y directions, nodal displacements are uneven which reflects orthotropic behavior. The next frequency having higher values leads to two half waves along x direction and one along y direction. It shows nearly half the displacement values compared to the first one with lesser warping behavior, which is found to repeat for third mode of vibration. The fourth mode having two half waves in both direction shows still lesser values of deflections compared to the first about one i.e. one fourth of first mode central deflection.

- In case of S_C_S_C support condition, the frequency values are found in good agreement with the exact values. Due to mix boundary condition at the corners of the plate a very small value of hogging moment along x-x and y-y direction is indicated. The first modal central displacement is found to reduce by approximately 50% compared to the previous example due to the fixity of edges. The remaining second and third modes of vibration also show warping behavior with undulated surface due to orthotropic behavior of plate.
- In case of S_S_S_C square orthotropic plate due to one way symmetry, total 50 elements were considered with 225 internal unknown moments. Using Eigen value analysis first four frequencies values are compared with the exact solution and are found to differ by about 10% to 18%. As usual values of Mx are found higher compared to My due to orthotropic behavior while at the corner some hogging moment is also developed along x-x direction and zero values along y-y due to simply supported ends. The torsional moment shows dual behavior after normalizing all the values with respect to centre. The mode shapes show approximately the same behavior as S_S_S_S plate problem with slightly reduced nodal values. The problem also depicts non uniform undulation due to orthotropic behavior in all the modal behavior.
- In case of C_C_C_S orthotropic plate the frequency values are found to differ by about 8% to 12% from the exact solution. The first modal deformation pattern shows maximum normalized value at the centre of the plate, second and third modal deformation patterns depict maximum value at centre of left half and centre of right half with zero

along centre line. While for the fourth mode repetition of first mode with opposite sign into four quadrants is observed.

- For the last problem of clamped boundary condition all around, first four frequencies are found within 10% to those of exact values. Due to clamp condition at A, the values of moments M_x and M_y are found of hogging nature. The mode shapes clearly indicated the effect of fixity of all edges on the behavior of the plate.

CHAPTER 13

CIRCULAR AND ANNULAR PLATE BENDING PROBLEMS

13.1 GENERAL REMARKS

Geometrically, circular plate is a 2D continuum problem having thickness much smaller than least plan dimension. If one is interested in the response of plate for which deformations are not axially symmetric, for example, the problem may be of a sector of a plate or the axial symmetry of geometry may be lost by presence of unsymmetrical loading, one is forced to treat it as a 2D problem. However, if plate is symmetrically loaded, its axisymmetric character allows it to model it as a one dimensional problem.

Circular plated structures subjected to arbitrary loading $q_o(r, \theta)$ can be analysed by applying the governing basic differential equation to the bending form of deformed circular plate surface which is given by

$$\nabla^4 w = \frac{q_o(r,\theta)}{D} \qquad \ldots (13.1)$$

Where $\nabla^2 = \left(\frac{\partial^2}{\partial r^2} + \frac{1}{r}\frac{\partial}{\partial r} + \frac{1}{r^2}\frac{\partial^2}{\partial \theta^2}\right)$ is a biharmonic operator.

In the present chapter, to deal with axisymmetric plate problem, a two noded line element is developed with two force and four displacement degrees of freedom, which is named as CIRC_2F_4D. Different matrices are derived by discretizing the expressions for potential and complimentary strain energies. Steps required for finding the solution based on IFM are described. Simply supported and clamped plate problems are analysed under central point load and uniform lateral pressure. Results obtained for moments and deflections are compared with the available solutions.

13.2 DEVELOPMENT OF IFM BASED FORMULATION

13.2.1 Governing Equations

The IFM equations are obtained by coupling the 'm` number of equilibrium equations and r = n – m compatibility conditions. The m equilibrium equations (EE) are written as

$$[B]\ \{F\} = \{P\} \qquad \text{… (13.2)}$$

and the' r' compatibility conditions are written as

$$[C]\ [G]\ \{F\} = \{\delta R\} \qquad \text{… (13.3)}$$

These conditions are combined to obtain the IFM governing equations as

$$\begin{bmatrix} [B] \\ \hline [C][G] \end{bmatrix} \{F\} = \begin{bmatrix} \{P\} \\ \hline \{\delta \mathbf{R}\} \end{bmatrix} \text{ Or } [S]\ \{F\} = \{P\} \qquad \text{… (13.4)}$$

After knowing the forces {F}, displacements {δ} are calculated using the following equation

$$\{\delta\} = [J]\ \{[G]\ \{F\} + \{\beta^0\} \qquad \text{… (13.5)}$$

where [J] = m rows of $[[S]^{-1}]^T$, [B] is equilibrium matrix of m x n size which is sparse and unsymmetrical, [G] is a symmetrical flexibility matrix; it is a block-diagonal matrix where each block represents the element flexibility matrix for an element, [C] is the compatibility matrix of size r x n, {δR} = -[C] $\{\beta\}^0$ is the effective deformation vector with $\{\beta\}^0$ being the initial deformation vector of dimension 'n', [S] is the unsymmetrical matrix of size n x n and [J] is the m x n size deformation coefficient matrix which is back-calculated from [S] matrix.

In IFM, the equilibrium matrix [B] links internal forces to external loads, compatibility matrix [C] governs the deformations and flexibility matrix [G] relates deformations to forces. Both the equilibrium and compatibility matrices of the IFM are unsymmetrical whereas the material constitutive matrix and the flexibility matrix are symmetrical.

13.2.2 Element Equilibrium Matrix [Be]

The elemental equilibrium matrix written in terms of forces at grid points represents the vectorial summation of 'n' internal forces {F} and 'm' external loads {P}. The nodal EE in matrix notation can be stored as rectangular matrix $[B_e]$ of size m x n. The variational functional is evaluated as a portion of IFM functional which yields the basic elemental equilibrium matrix $[B_e]$ in explicit form as follows:

$$U_p = \int_D \left\{ M_r \frac{\partial^2 w}{\partial r^2} + M_\theta \frac{1}{r}\frac{\partial w}{\partial r} \right\} 2\pi\, r\, dr \qquad \text{... (13.6)}$$

$$= \int_D \{M\}^T \{\varepsilon\}\, da \qquad \text{... (13.7)}$$

where $\{M\}^T = (M_r,\ M_\theta,)$ are the internal moments and $\{\epsilon\}^T = \left(\frac{\partial^2 w}{\partial r^2} \quad \frac{1}{r}\frac{\partial w}{\partial r} \right)$ represents the curvatures corresponding to each internal moment.

Consider a two-noded, 4 ddof (δ_1 to δ_4) line element of thickness t with length as 'a' along the radial direction as shown in **Figs. 13.1 and 13.2**.

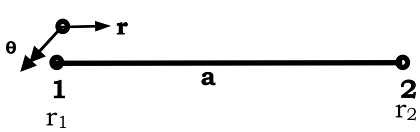


Fig. 13.1 Line Element

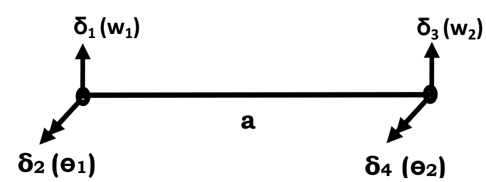


Fig. 13.2 Degrees of Freedom

The force field is chosen in terms of two independent forces F_1 and F_2. Relations between internal moments and independent forces are written as

$$M_r = F_1 + F_2 r \quad \text{and} \quad M_\theta = F_1$$

Arranging in matrix form,

$$\begin{Bmatrix} M_r \\ M_\theta \end{Bmatrix} = \begin{bmatrix} 1 & r \\ 1 & 0 \end{bmatrix} \{F_e\}$$

Or $\{M\} = [Y]\{F_e\}$ ... (13.8)

Displacement function by considering r_1 and r_2 as radius of extreme points of the element can be written as

$$w(r,\theta) = N_1\delta_1 + N_2\delta_2 + N_3\delta_3 + N_4\delta_4 = [N]\{\delta\} \quad \text{... (13.9)}$$

where $N_1 = \left(\frac{r - r_2}{r_1 - r_2}\right)^2 \left[1 + 2\left(\frac{r_1 - r}{r_1 - r_2}\right)\right], N_2 = \left(\frac{r - r_2}{r_1 - r_2}\right)^2 (r - r_1),$

$$N_3 = \left(\frac{r - r_1}{r_2 - r_1}\right)^2 \left[1 + 2\left(\frac{r_2 - r}{r_2 - r_1}\right)\right], N_4 = \left(\frac{r - r_1}{r_2 - r_1}\right)^2 (r - r_2)$$

where [N] is a shape function matrix and {δ} is a displacement vector. By arranging all force and displacement functions properly, one can discretize the Eq. (13.6) to obtain the element equilibrium matrix as follows.

$$U^e = \{X\}^T[B^e]\{F\} \quad \text{... (13.10)}$$

Where, $[B^e] = 2\pi\int_s [Z]^T[Y]rdr$ ... (13.11)

where again [Z] = [L][N]; [L] is the x differential operator matrix with respect to r, [N] is the displacement interpolation function matrix and [Y] is the force interpolation function matrix. After calculating [Z] matrix and [Y] matrix and integrating, the equilibrium matrix $[B^e]$ is obtained using Eq. (13.10). For element 1, for example, with $r_1 = 0$ and $r_2 = a$, the $[B^e]$ matrix will be as follows.

$$[B^e] = \begin{bmatrix} 0 & -a \\ 0 & -\frac{a^2}{6} \\ 0 & a \\ -a & -\frac{5a^2}{6} \end{bmatrix} \quad \text{... (13.12)}$$

13.2.3 Elemental Flexibility Matrix $[G^e]$

The basic elemental flexibility matrix is obtained by discretizing the complementary strain energy which gives

$$[G^e] = 2\pi\int_s [Y]^T[D][Y] r dr \quad \ldots \; (13.13)$$

where [Y] is force interpolation function matrix and [D] is material property matrix of size (2 x 2). Substituting values in Eq.(13.13) and integrating radially, it yields the symmetrical flexibility matrix $[G^e]$ which for the first element is as follows.

$$[G^e] = \frac{24\pi}{Et^3}\begin{bmatrix} \frac{7}{10}a^2 & \frac{7}{30}a^3 \\ \frac{7}{30}a^3 & \frac{1}{4}a^4 \end{bmatrix} \quad \ldots \; (13.14)$$

13.2.4 Global Compatibility Matrix [C]

The compatibility matrix is obtained from the deformation displacement relation ($\{\beta\} = [B]^T\{\delta\}$), using directly [B] matrix as input for LIUT based matlab module. In DDR all the deformations are expressed in terms of all possible nodal displacements and the 'n - m' compatibility conditions are developed in terms of internal forces.The concatenating or global compatibility matrix z.cMatrix, which is compatibility matrix [C] is evaluated by multiplying the coefficients of the compatibility conditions and the global flexibility matrix [G].

13.3 CIRCULAR PLATE BENDING EXAMPLES

Total four examples of mild steel circular plate are considered here to demonstrate the applicability of the proposed formulation. Centrally loaded and uniformly loaded plates with simply supported and clamped circular edges are considered. Each plate is subjected to a central point load (CPL) of 10 kN and Uniformly Lateral Pressure (ULP) of 10 kN/m^2. The diameter of the plate is taken as 6000 mm with thickness of plate as 100 mm. The modulas of elasticity of steel material is considered as 2.01 x 10^{11} N/m^2. whereas Poisson's ratio is considered as 0.3. **Fig. 13.3** shows a clamped circular plate under two different loading conditions. With reference to a clamped plate, the steps for finding the solution are as follows:

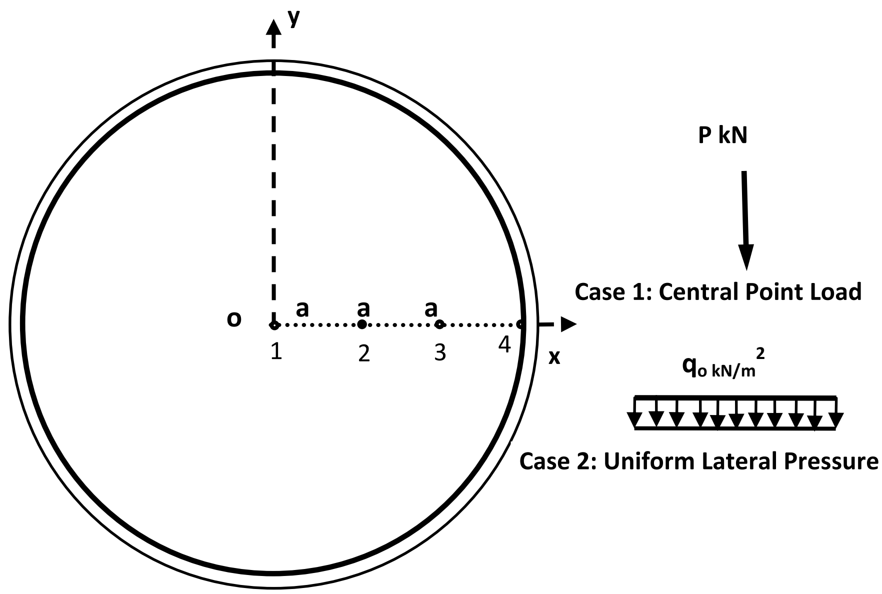


Fig. 13.3 Clamped Plate showing Discretization including Loading

Step 1: A two-noded line element having length 'a' with 2 fdof and 4 ddof is used for discretizing the problem into three elements. The [B^e] matrix is obtained by substituting a = 1m in Eq. (13.12). After assembly, the size of global equilibrium matrix will be 5 x 6.

Step 2: The compatibility matrix for the three elements is obtained from the displacement deformation relations (DDR) i.e. $\beta = [B]^T\{\delta\}$. In the DDR, 6 deformations which correspond to 6 force variables are expressed in terms of 5 displacements. The problem requires only one compatibility condition [C]; which is obtained by eliminating the 5 displacements from the 6 DDRs. It is auto-generated using Matlab based computer program by giving input as upper part of the global equilibrium matrix.

Step 3: The flexibility matrix for the problem is obtained by diagonal concatenation of the three elemental flexibility matrices of size 2 x 2 calculated as per Eq. (13.14).

Step 4: By multiplying compatibility matrix [C] and global flexibility matrix [G], bottom most part of the global equilibrium matrix is obtained. Assembling both gives the complete [S] matrix of size 6 x 6, which comprises of EE and CC. The forces {F} are obtained by using Matlab's inverting routine.

Step 5: The displacements are obtained by using the relation ({δ} = [J][G]{F}), where [J] = m rows of matrix $[[S]^{-1}]^T$.

Following the above steps, analysis is carried of the circular plate problems by changing loading and boundary conditions. **Figs. 13.4** and **13.5** respectively show variation of lateral deflection and rotation along a radial direction of the plate for different loading and the boundary conditions. **Fig. 13.6** shows the deformed shape of a simply supported circular plate under uniform lateral pressure whereas **Fig. 13.7** shows the same for clamped plate under central point load.

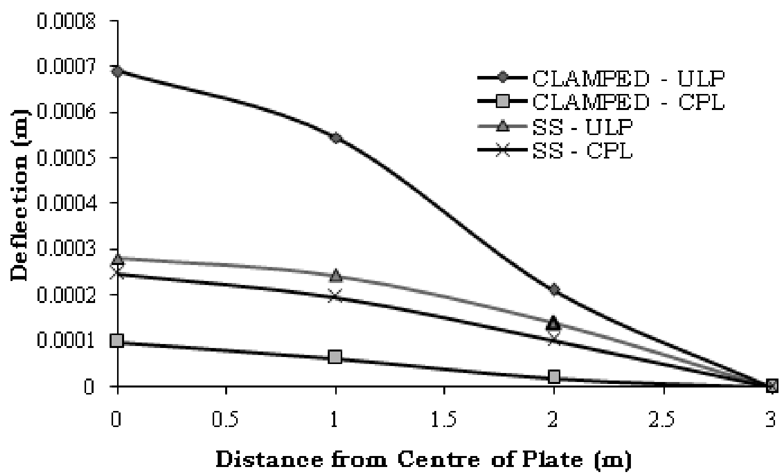


Fig.13. 4 Plot of Deflection along a Radial Line

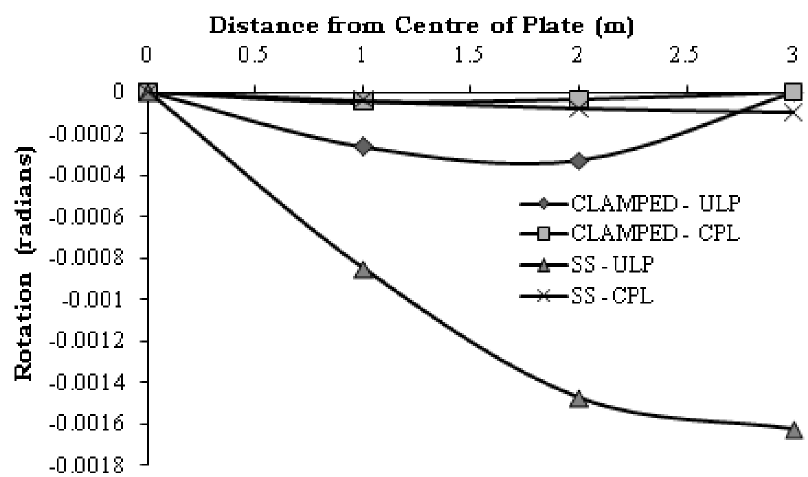


Fig. 13. 4 Plot of Deflection along a Radial Line

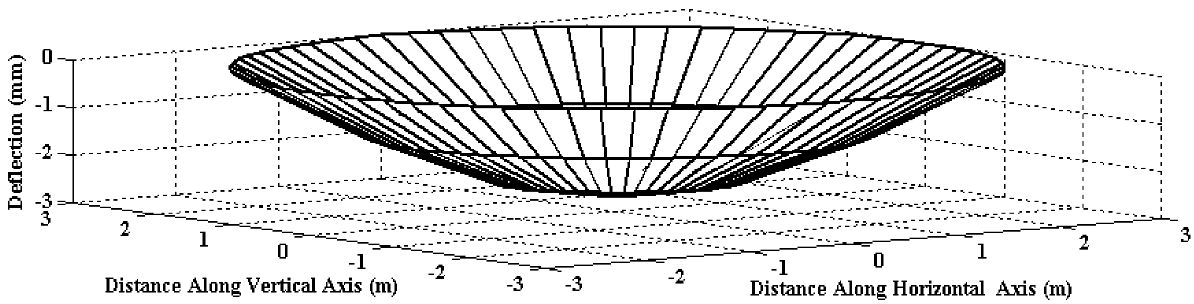


Fig. 13.6 Deformed Shape for SS Circular Plate under ULP

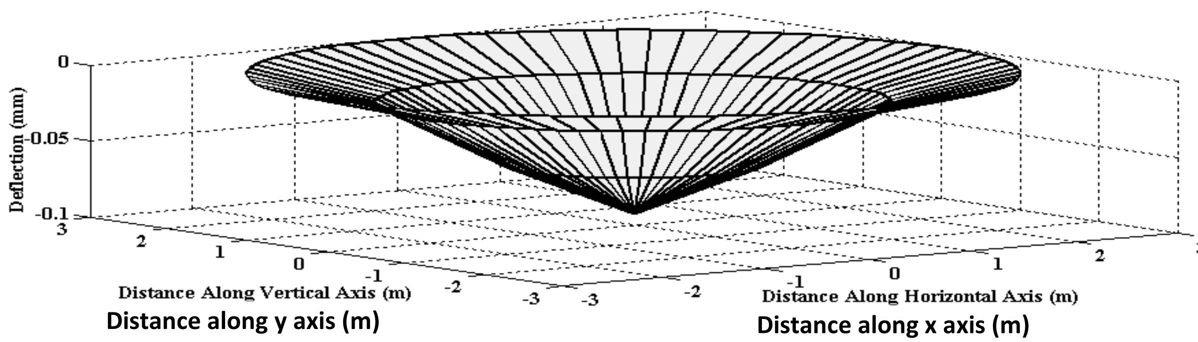


Fig. 13.7 Deformed Shape for Clamped Circular Plate under CPL

Table 13.1 shows the comparison of results for nodal deflection whereas Tables 13.2 and 13.3 show comparison of results obtained for moments M_r and M_θ. For simply supported plates, IFM results are compared with those given by Timoshenko **[103]** whereas for clamped plates the results are compared with those reported by Chandrasekhar **[99,101]**. **Figs 13.8** and **13.9** show respectively the contours for Mr and M_θ for a simply supported circular plate under uniform lateral pressure. These contours are depicted for a top right quarter of the plate and are drawn using the Matlab software.

Table 13.1 Comparison of Lateral Deflection

Dist from centre (m)	Lateral Deflection x 10^{-5} (m)							
	SS - ULP		SS - CPL		Clamped – ULP		Clamped - CPL	
	IFM	Exact	IFM	Exact	IFM	Exact	IFM	Exact
0.0	270.06	281.73	23.25	24.83	66.17	69.10	9.23	9.78
1.0	233.49	243.58	18.99	19.68	53.28	54.60	5.96	6.30
2.0	133.59	140.36	9.62	10.27	20.42	21.32	1.90	1.91
3.0	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Table 13.2 Comparison of Moments for Simply Supported Plate

Dist from centre (m)	SS - ULP				SS - CPL			
	M_r (kN-m)		M_θ (kN-m)		M_r (kN-m)		M_θ (kN-m)	
	IFM	Exact	IFM	Exact	IFM	Exact	IFM	Exact
0.0	18.00	18.56	18.00	18.56	-8×10^6	∞	-9×10^7	∞
1.0	15.82	16.50	16.66	17.38	-1.11	-1.13	-1.82	-1.86
2.0	9.90	10.31	13.24	13.81	-0.41	-0.42	-1.12	-1.14
3.0	0	0	7.55	7.87	0	0	-0.71	-0.72

Table 13.3 Comparison of Moments for Clamped Plate

Dist from centre (m)	Clamped - ULP				Clamped - CPL			
	M_r (kN-m)		M_θ (kN-m)		M_r (kN-m)		M_θ (kN-m)	
	IFM	Exact	IFM	Exact	IFM	Exact	IFM	Exact
0.0	7.07	7.31	7.07	7.31	-0.75	-0.79	-0.23	-0.24
1.0	5.08	5.25	5.92	6.12	-0.32	-0.34	0.85	0.90
2.0	-0.91	-0.94	2.48	2.56	0.35	0.37	0.17	0.18
3.0	-1.09	-1.12	-3.26	-3.37	0.75	0.79	-0.23	-0.24

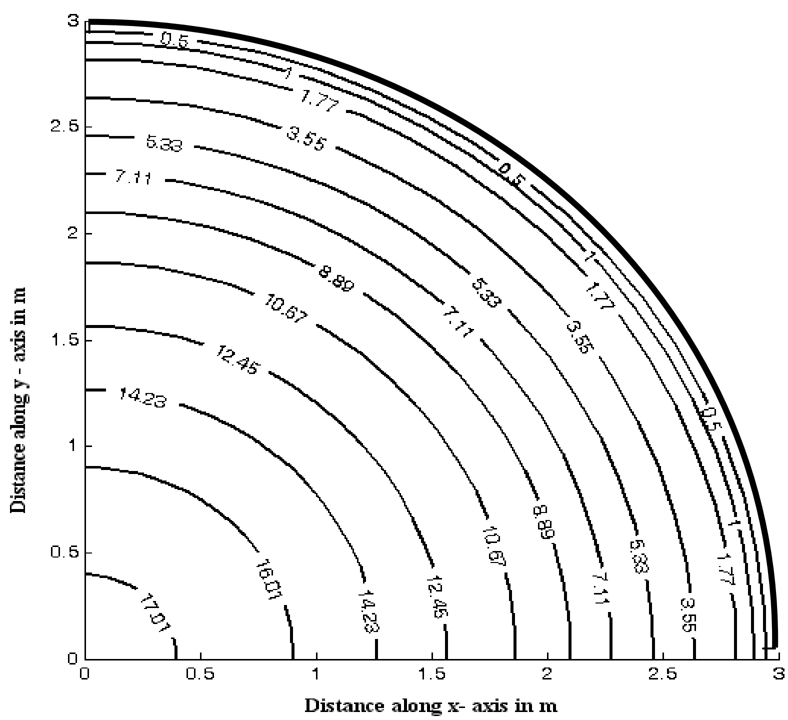


Fig. 13.8 Contours of M_r (kN-m) for SS Circular Plate under ULP

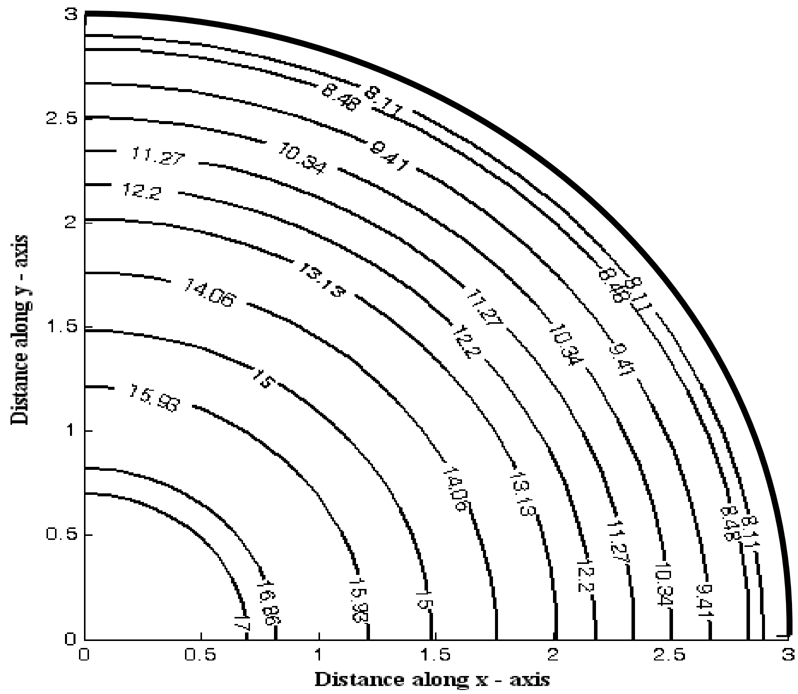


Fig. 13.9 Contours of M_θ (kN-m) for SS Circular Plate under ULP

13.4 ANNULAR PLATE BENDING EXAMPLES

Annular plate is a plane geometry with circular plan having a circular hole at the center with dimensions as per practical applications, i.e. pressure vessels, automotive and large machinery components etc. If the thickness of circular plate is not greater than one tenth of the diameter, one needs not to model it as a 3D continuum and simple 2D plate theory can be applied to calculate the deformation and stresses. Further, if the plate is axisymmetrically loaded instead of asymmetrically loaded, its axisymmetric behavior allows one to treat it as a 1D problem.

For the analysis of annular plates also CIRC_2F_4D having two force and four displacement degrees of freedom is used. After calculating [Z] matrix and stress interpolation matrix [Y] and integrating, the equilibrium matrix for annular geometry [B_e] is obtained for element 1 with inner radius r_1 = b and outer radius r_2 = (a + b). The [B_e] matrix will be as follows:

$$[B_e] = \begin{bmatrix} 0 & \frac{b}{2} - a \\ 0 - \frac{b^2}{3} & -\frac{a^2}{6} \\ 0 & a + \frac{b}{2} \\ -a & -\frac{5a^2}{6} \end{bmatrix} \quad \ldots (13.15)$$

The basic elemental flexibility matrix is obtained by discretizing the complementary strain energy which gives:

$$[G_e] = 2\pi \int_{r_1}^{r_2} [Y]^T [D][Y]\, r\, dr \quad \ldots (13.16)$$

where [Y] is moment interpolation function matrix and [D] is material property matrix of size 2 x 2. Substituting values in Eq. (13.16) and integrating, it yields the symmetrical flexibility matrix $[G_e]$ which for the first element is as follows:

$$[G_e] = \frac{24\pi}{Et^3} \begin{bmatrix} \frac{7}{10}a^2 & \frac{7}{30}a^3 + \frac{14}{15}a^2 b \\ \frac{7}{30}a^3 + \frac{14}{15}a^2 b & a^4 + \frac{a^2 b^2}{8} \end{bmatrix} \quad \ldots (13.17)$$

The data assumed for numerical study of axisymmetrically loaded annular plates is as follows. A Uniform Lateral Pressure (ULP) q of 10^6 N/m^2 (1 MPa) on total plate surface and Line Loading (LL) p of 1000 N/m at inner or outer edge of the annular plate are considered. The outer and inner diameters of the plate are taken as 200 mm and 80 mm respectively with thickness of plate as 10 mm. E is considered as 2.01 x 10^{11} N/m^2 and ν as 0.3.

Using the IFM based procedure, first, a convergence study is carried out to decide the suitable number of elements. **Fig 13.10** depicts an annular plate with necessary details and **Fig 13.11** shows the convergence graph for deflection at the outer edge for annular plate with clamped condition at the inner edge and free at the outer boundary subjected to uniformly lateral pressure of 1 MPa. As can be seen from the convergence graph, discretization with 5 elements gives results quite close to the exact solution and therefore for remaining examples also same scheme is used

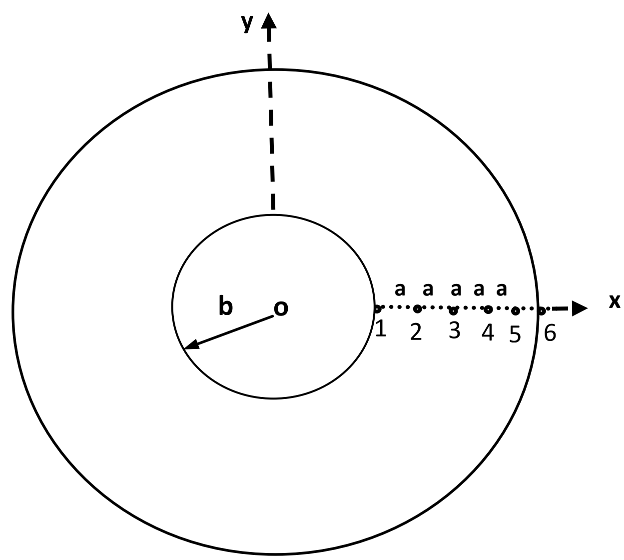


Fig. 13.10 Annular Plate with 5 Elements

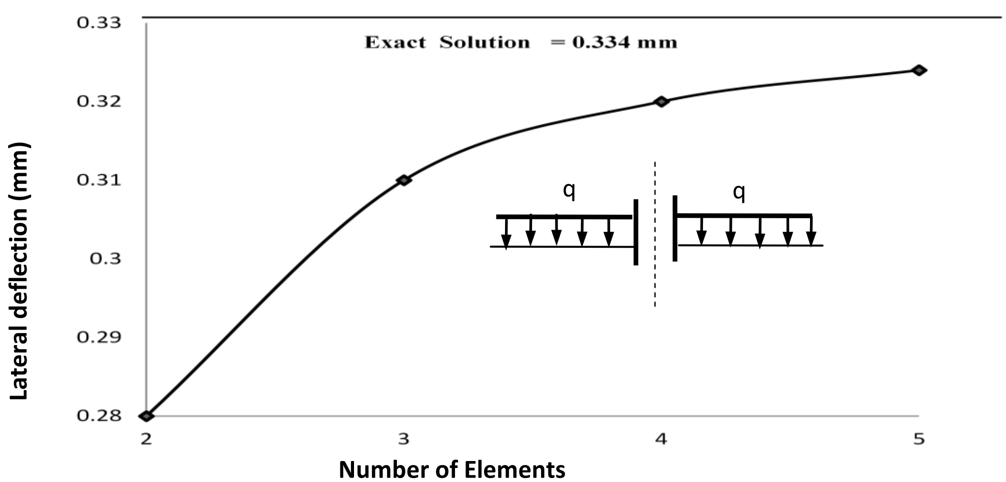


Fig. 13.11 Convergence Graph for Deflection at Outer Periphery

In **Table 13.4** and **Table 13.5** results obtained for deflection w and moments M_r and M_Θ along a radial line at different nodal points are compared with the available solution **[102,104]** for uniform lateral pressure q and line loading p for clamped boundary conditions

Table 1 Results Obtained for Clamped Annular Plate

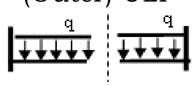
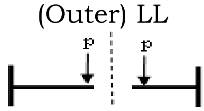
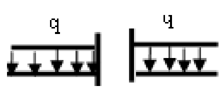
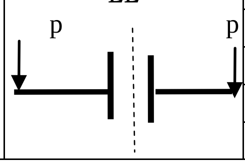
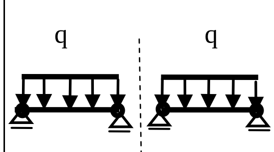
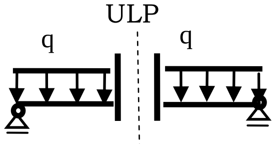
Ex. No	TYPE	NODE	δ (mm)		Mr (kN-mm)		Mθ (kN-mm)	
			IFM	Exact	IFM	Exact	IFM	Exact
1	Clamped (Outer) ULP 	1	0.0185	0.0199	0.00	0.000	899.14	901.6
		2	0.0081	0.0087	41.32	44.19	596.21	617.3
		3	0.0032	0.0004	89.21	100.5	365.1	394.1
		4	0.0020	0.0003	334.2	339.8	145.12	174.1
		5	0.0001	0.0002	670.4	675.9	59.21	63.11
		6	0.0000	0.0000	998.1	1088	321.14	330.1
2	Clamped (Outer) LL 	1	0.0012	0.002	0.00	0.000	20.14	24.60
		2	0.0014	0.0016	-5.14	-6.01	19.14	24.98
		3	0.0011	0.0013	-9.11	-9.84	19.00	23.66
		4	0.0007	0.0009	-12.0	-12.62	18.11	22.73
		5	0.0003	0.0004	-14.1	-14.81	18.04	21.60
		6	0.000	0.0000	-16.5	-16.58	18.00	20.14
3	Clamped (Inner) ULP 	1	0.000	0.000	-331.	-335.7	-92.14	-100
		2	0.032	0.040	-180.0	-183.3	-214.1	-238.
		3	0.101	0.140	345.2	375.0	-294.1	-295.
		4	0.135	0.166	365.1	375.9	-349.1	-357.
		5	0.211	0.243	228.2	241.8	-442.5	-449.
		6	0.322	0.332	0.00	0.00	-568.1	-579.
4	Clamped (Inner) LL 	1	0.000	0.00	-50.00	-57.45	-89.12	-92.4
		2	0.0011	0.0014	-30.44	-34.69	-60.75	-68.6
		3	0.0031	0.0033	-19.45	-21.27	-50.14	-56.2
		4	0.0051	0.0055	-11.47	-12.14	-43.14	-47.1
		5	0.0070	0.008	-5.00	-5.36	-38.54	-40.3
		6	0.091	0.108	0.00	0.00	-30.11	-35.1

Table 2 Results Obtained for Annular Plates Simply Supported at Outer Edge

Ex. No.	TYPE	NODE	δ (mm)		Mr (kN-mm)		Mθ (kN-mm)	
			IFM	Exact	IFM	Exact	IFM	Exact
1	SS (Outer & Inner) ULP 	1	0.00	0.00	0.00	0.00	566.2	597.1
		2	0.018	0.020	-124.3	-127.1	145.3	150.2
		3	0.031	0.032	-345.1	-350.1	-187.2	-195.
		4	0.030	0.034	-487.1	-500.8	-502.5	-515.
		5	0.021	0.024	-700.3	-724.1	-823.5	-837.
		6	0.00	0.00	0.00	0.00	-1021	-1133
2	SS (Outer) & Clamped (Inner) ULP 	1	0.00	0.00	1698	1761	498.2	528.1
		2	0.005	0.005	534.3	567.1	434.1	471.1
		3	0.011	0.014	-312.1	-339.1	302.3	334.5
		4	0.014	0.020	-1093	-1141	-267.2	-285.
		5	0.014	0.017	-1832	-1894	-734.1	-741.
		6	0.00	0.00	-2477	-2652	-1178	-1219

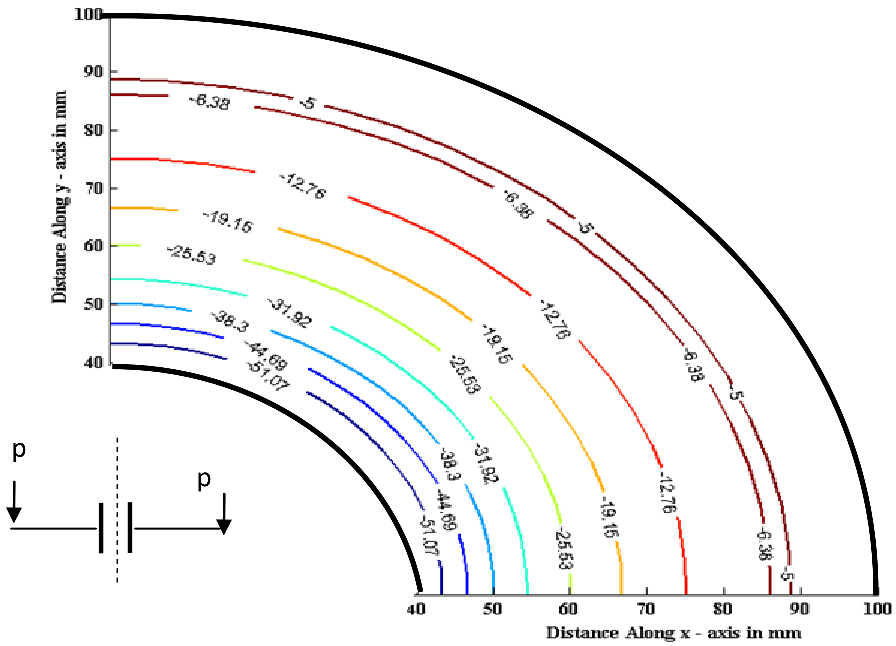


Fig. 13.12 Contours of M_r for Plate Clamped along Inner Edge and Outer Edge under Line Loading

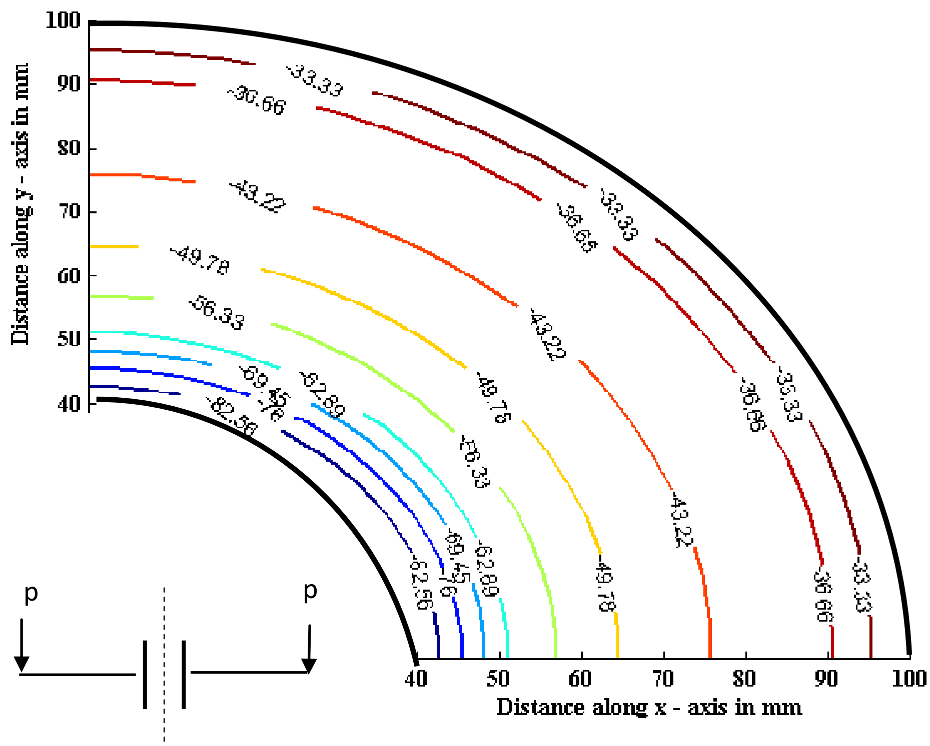


Fig. 13.13 Contours of M_θ for Plate Clamped along Inner Edge and Outer Edge under Line Loading

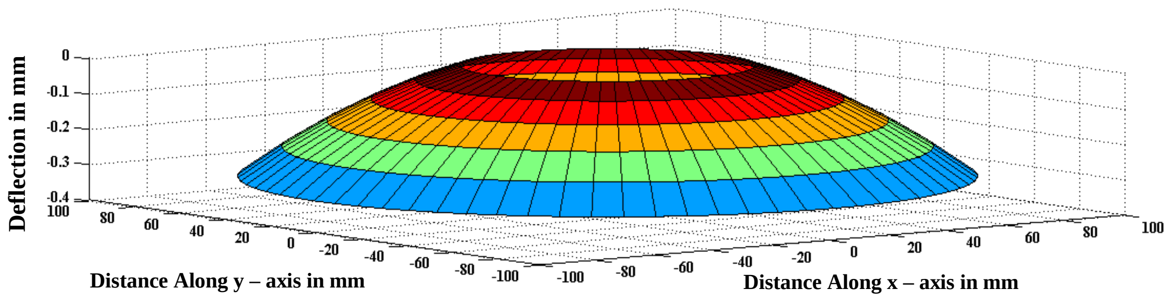


Fig. 13.14 Deformed shape of annular plate under uniform lateral pressure with clamped inner edge

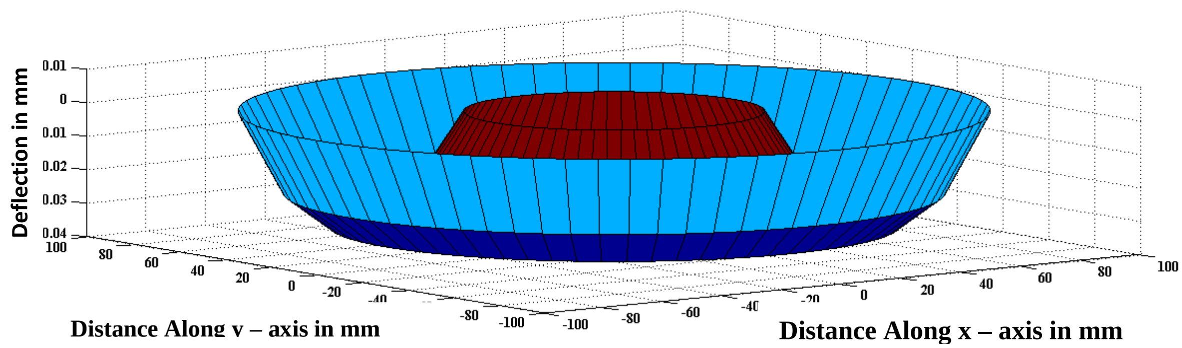


Fig. 13.15 Deformed Shape of Annular Plate under Uniform Lateral Pressure with Inner and Outer Edges as SS

13.5 DISCUSSION OF RESULTS

- Results obtained for simply supported and clamped circular plates under central point load and uniform lateral pressure by using just three elements in the radial direction are found quite encouraging. Results obtained for moments, deflections and rotations are found within 5 percent, in most of the cases, of those obtained by using classical method. Also, moment results are found more accurate than the deflection results. The comparison of results obtained for deflections and moments using IFM based CIRC_2F_4D element with 5 element discretization indicated a good agreement with those based on the classical methods for a variety of problems under different boundary conditions at inner and outer periphery when the annular plate is subjected to uniform lateral pressure or a liner load along inner or outer periphery.

CHAPTER 14

BUCKLING ANALYSIS OF 1D AND 2D STRUCTURES

14.1 GENERAL REMARKS

In structures where dominant loading is usually static, the most common cause of the collapse is a buckling failure. Buckling may occur locally in a manner that may or may not trigger collapse of the entire structure. Compression members or beams may buckle as individual members, or structure as a whole may buckle. Often the buckling of a system can be studied by using the stiffness obtained from appropriate load deflection characteristics defined by elementary theory.

In the stiffness based formulation, one has to consider geometric stiffness and mechanical stiffness in the buckling analysis of the structure. The geometric stiffness purely depends on in-plane loading and length of member whereas mechanical stiffness of a member depends upon the physical properties i.e. material and area based inertia. Geometric stiffness exists in all structural members; however, it only becomes important if it is large compared to the mechanical stiffness. This converts the complete problem into eigen value analysis and gives the balancing numerical eigen vector.

Evaluation of buckling strength of truss is an important limit state of collapse and design must ensure highest reliability. The buckling load of an overall system is conceptually intuitive and simple to evaluate, whenever buckling of weakest member in compression causes the instability of complete system. But, whenever there is a rigid jointed structure, the evaluation of buckling load is more complex as it often involves the instability of more than one member at a time. Thus computation of the buckling load requires tedious non-linear iterative analysis, which is mostly done by utilizing the Finite Element based software packages.

In the present chapter, IFM is extended to facilitate buckling analysis of various types of skeletal and continuum structures. Different closed form matrices are worked out. Eigen operators are used from Matlab software for calculating the buckling load, relative internal moments and nodal displacements. The critical load is calculated and validated by comparing the same with the analytical solution.

14.2 FORMULATION FOR SKELETAL STRUCTURES

14.2.1 IFM Based Stability Equations

In IFM, the element forces {F} and the external load vector {P} are related as

$$\begin{bmatrix} \frac{[B]}{[C][G]} \end{bmatrix} \{F\} = \{P\} \qquad \ldots (14.1)$$

Or $$[S]\{F\} = \{P\} \qquad \ldots (14.2)$$

where [B] is the basic equilibrium matrix of size m x n, [C] is the compatibility matrix of size (n - m) x n and [G] is the concatenated flexibility matrix of size n x n, with m being the force degrees of freedom and n being the displacement degrees of freedom.

The nodal displacement vector {δ} is related to the element force vector {F} as follows:

$$\{\delta\} = [S^{-1}]^T[G]\{F\} \qquad \ldots (14.3)$$

The eigen based stability analysis equation is obtained by usual perturbation theory.

$$[S]\{F\} = \lambda[K_g][J][G]\{F\} \qquad \ldots (14.4)$$

Or $$[[S] - \lambda[S_b]]\{F\} = \{0\}$$

where $[K_g]$ is the geometric stiffness matrix and λ is the stability parameter. The matrix $[S_b]$ is referred as the 'IFM stability matrix' and [J] consists of number of rows taken from $[S^{-1}]^T$ matrix. After calculating the eigen vector λ of size m, each is substituted in Eq. (14.4) for the calculation of {F}. Nodal displacements {δ} are worked out for each vector by substituting {F} in Eq. (14.3).

Two different methods are available for modeling the geometric stiffness matrix $[K_g]$. In the first method, the consistent geometric stiffness matrix $[K_{gc}]$ is derived by using the third order polynomial function by neglecting the contributions of rotational effect assigned to nodal coordinates whereas in the second method, a simplified geometric stiffness matrix $[K_{gs}]$ is derived by using only the first order polynomial function by neglecting the contribution due to the rotational effect assigned to the nodal coordinates. The second method for developing stiffness matrix with relatively few terms ensures full numerical stability. The procedure for deriving the different matrices is illustrated below with reference to a beam member.

14.2.2 Geometric Stiffness Matrix

Consider a beam member of length L and flexural rigidity EI, subjected to an axial force P along its length. It consists of total four nodal displacements, two at each node i and j respectively, as shown in **Fig. 14.1.**

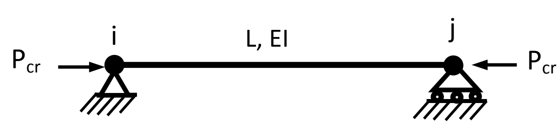


Fig. 14.1 Member under Buckling Load

Combining all the coefficients for consistent and simplified geometric stiffness matrices related to each degree of freedom for uniform beam member, one can write the matrices $[K_{gc}]$ and $[K_{gs}]$ of size 4 x 4 for each member as follows:

$$[K_{gc}] = \frac{P}{30L}\begin{bmatrix} 36 & 3L & -36 & 3L \\ 3L & 4L^2 & -3L & -L^2 \\ -36 & -3L & 36 & -3L \\ 3L & -L^2 & -3L & 4L^2 \end{bmatrix} \text{ and}$$

$$[K_{gs}] = \frac{P}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \qquad \ldots (14.5)$$

Transformation of the element geometric stiffness matrix $[K_{ge}]$ of pin jointed structure from the local coordinate to global coordinate system can be done by the following formula:

$$[K_g] = [T]^T[K_{ge}][T], \text{ where}$$

$$[T] = \begin{bmatrix} c & s & 0 & 0 \\ -s & c & 0 & 0 \\ 0 & 0 & c & s \\ 0 & 0 & -s & c \end{bmatrix} \text{ and } [K_{ge}] = \frac{P}{L}\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \qquad \ldots (14.6)$$

In the transformation matrix [T], c = cosθ, s = sinθ, and θ is the angle between the global and local axes.

14.2.3 Steps for Buckling Analysis

Steps required for the buckling analysis are explained here with the help of a beam example. A prismatic propped cantilever beam with total span 2L is discretized into two elements with length of each element as L as shown in **Fig. 14. 2.**

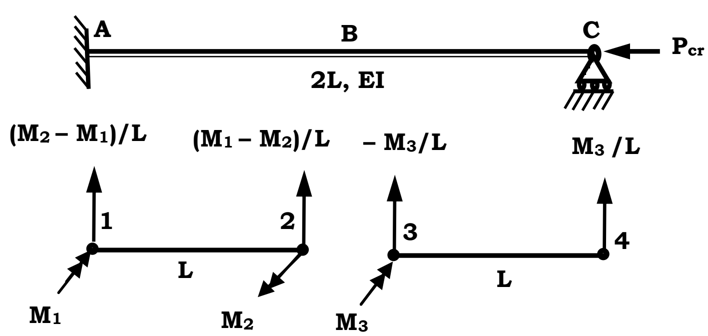


Fig. 14.2 Propped Cantilever Beam Example

Step 1 Develop equilibrium matrix [B]: After discretizing the beam into two elements and applying the equilibrium conditions, in the direction of rotation and deflection in terms of internal moments M_1, M_2 and M_3 at point B in **Fig. 14.2**, one can write the equilibrium equations in matrix form as

$$\begin{bmatrix} \frac{1}{L} & -\frac{1}{L} & -\frac{1}{L} \\ 0 & 1 & -1 \end{bmatrix} \begin{Bmatrix} M_1 \\ M_2 \\ M_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \end{Bmatrix} \qquad \text{... (14.7)}$$

which using concise notations can be written as

$$[B]\{F\} = \{P\} \qquad \text{... (14.8)}$$

Step 2 Develop displacement deformation relations (DDR): Considering δ_1 and δ_2 as lateral and rotational nodal displacements at point B respectively, which are associated with the three deformations β_1, β_2 and β_3 corresponding to internal moments M_1, M_2 and M_3 respectively, the displacement deformation relationship can be expressed in terms of nodal displacements as follows:

$$\begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \end{Bmatrix} = \begin{bmatrix} \frac{1}{L} & 0 \\ -\frac{1}{L} & 1 \\ -\frac{1}{L} & -1 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix} \quad \text{or } \{\beta\} = [B]^T\{\delta\} \qquad \text{... (14.9)}$$

Step 3 Develop the compatibility matrix [C]: Writing the above relation in equation form,

$$\beta_1 = \delta_1/L, \ \beta_2 = -(\delta_1/L) + \delta_2 \text{ and } \beta_3 = -(\delta_1/L) - \delta_2 \qquad \text{... (14.10)}$$

Eliminating displacements from the above equation, one can write

$$2\beta_1 + \beta_2 + \beta_3 = 0$$

Or in a matrix form it can be written as

$$\begin{bmatrix} 2 & 1 & 1 \end{bmatrix} \begin{Bmatrix} \beta_1 \\ \beta_2 \\ \beta_3 \end{Bmatrix} = \{0\} \qquad \text{... (14.11)}$$

This can be further written in the form as

$$[C]\{\beta\} = 0 \qquad \text{... (14.12)}$$

The null property of the matrix [C] can be checked by the product of $[B][C]^T$ which must be equal to zero for numerical stability of solution in IFM.

Step 4 Develop global flexibility matrix [G]: The global flexibility matrix is worked out using the force deformation relation, $\{\beta\} = [G]\{F\}$. For a single member having length L, flexural rigidity EI, subjected to M_1 and M_2 moment at each end, one can write

$$\beta_1 = \frac{L}{6EI}(2M_1 + M_2),\ \ \beta_2 = \frac{L}{6EI}(2M_2 + M_1) \text{ and } \beta_3 = \frac{L}{6EI}(2M_3)$$

or in matrix form,

$$\begin{Bmatrix}\beta_1\\ \beta_2\\ \beta_3\end{Bmatrix} = \frac{L}{6EI}\begin{bmatrix}2 & 1 & 0\\ 1 & 2 & 0\\ 0 & 0 & 2\end{bmatrix}\begin{Bmatrix}M_1\\ M_2\\ M_3\end{Bmatrix} = [G]\{F\} \quad \dots (14.13)$$

Where, $[G] = \frac{L}{6EI}\begin{bmatrix}2 & 1 & 0\\ 1 & 2 & 0\\ 0 & 0 & 2\end{bmatrix}$ is known as the global flexibility matrix.

... (14.14)

The concatenation of third row in the equilibrium matrix [B] can be obtained by multiplying compatibility matrix [C] by {β}. Thus one can convert all the displacement deformation relations (DDRs) into force deformation relations (FDRs). Substituting deformation vector {β} from Eq. (14.13) into Eq. (14.12), one can write

$$[C][G]\{F\} = \frac{L}{6EI}\begin{bmatrix}5 & 4 & 2\end{bmatrix}\begin{Bmatrix}M_1\\ M_2\\ M_3\end{Bmatrix} = \{0\} \quad \dots (14.15)$$

Step 5 Develop global geometric stiffness matrix [K_{gc}]: The global geometric stiffness matrix is worked out by assembling the two elemental matrices. For each element the geometric stiffness matrix in terms of P and

L is given by Eq. (14.5). Considering terms corresponding to global displacement degrees of freedom X_1 and X_2, one can write

$$[K_g] = \frac{P_{cr}}{30}\begin{bmatrix} 72 & 0 \\ 0 & 8L^2 \end{bmatrix} \quad \ldots (14.16)$$

Step 6 Calculate buckling load: Concatenating Eq. (14.15) into Eq. (14.7) gives the global equilibrium matrix [S]. Substituting all related matrices into Eq. (14.4) gives the complete global form of IFM based stability equation. Considering total span 2L of propped cantilever beam as 2m, cross-sectional dimension of beam as 0.01m x 0.01m and E of material as 2.01 x 10^8 kN/m^2, one can write

$$\begin{bmatrix} \frac{1}{L} & -\frac{1}{L} & -\frac{1}{L} \\ 0 & 1 & -1 \\ \frac{5L}{6EI} & \frac{4L}{6EI} & \frac{2L}{6EI} \end{bmatrix}\begin{bmatrix} M_1 \\ M_2 \\ M_3 \end{bmatrix} = \frac{P_{cr}}{30}\left[\left[\begin{bmatrix} 72 & 0 & 0 \\ 0 & 8L^2 & 0 \\ 0 & 0 & 0 \end{bmatrix}\right][J]\left[\frac{L}{6EI}\begin{bmatrix} 2 & 1 & 0 \\ 1 & 2 & 0 \\ 0 & 0 & 2 \end{bmatrix}\right]\right]\begin{bmatrix} M_1 \\ M_2 \\ M_3 \end{bmatrix} \quad \ldots (14.17)$$

Where $[J] = [S^{-1}]^T$.

Step 7 Calculate force mode shape: Substituting all the values of Pcr calculated from the Eq. (14.17), values of all other internal moments M_2 and M_3 can be worked out by assuming relative value of M_1 = 1 kN-m by considering any two random equation of moments by direct elimination approach.

Step 8 Calculate displacement mode shape: Substituting values of all internal moments in Eq. (14.3), the nodal displacements, which depend on the moments, can be calculated.

14.3 EXAMPLES OF BUCKLING ANALYSIS OF BEAMS

Various beam examples are considered here with one or two spans having different boundary conditions as shown in **Fig 14.3**. The length L of each element is taken as 1m. The cross sectional dimension of beam is considered as 0.01m x 0.01m with mild steel as basic material having E as of 2.01 x 10^8 kN/m^2 [ ]. Results obtained for different types of beam problems are reported here in **Table 14.1.**

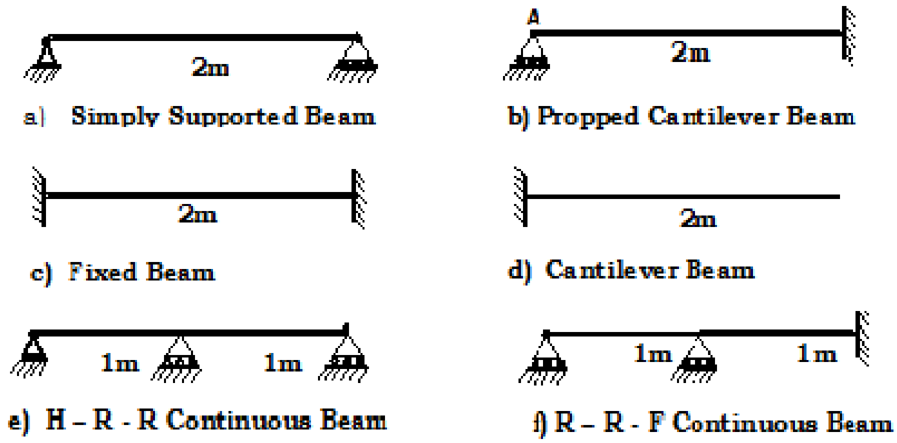


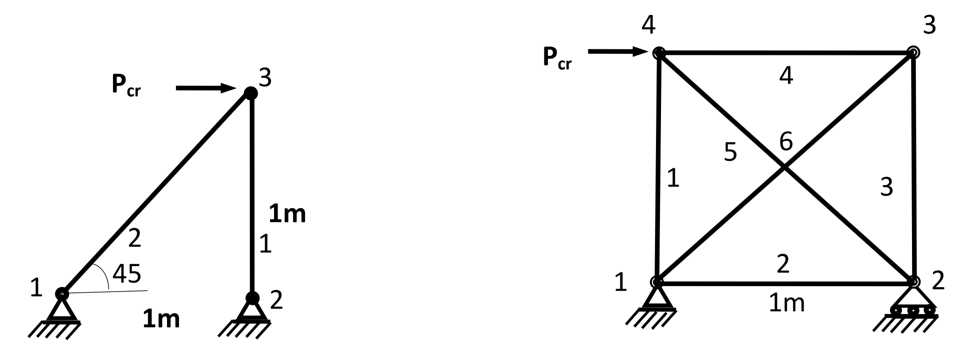
Fig.14. 3 Different Types of Beam Examples

Table 14.1 Results for Beam Examples

Buckling Load (P_{cr}) kN		Internal Moment (M)		Nodal Disp. (δ)
IFM	Euler's			
Simply Supported Beam Example				
0.42	0.41	M_1 = 1.0	M_2 = -1.0	δ_1 = 2.00
Propped Cantilever Beam Example				
0.88	0.83	M_1 = 1.0	M_2 = -0.7	δ_1 = 1.00
		M_3 = 0.8		θ_2 = 0.08
Fixed Beam Example				
1.69	1.65	M_1 = 1.0	M_2 = -0.87	δ_1 = 1.023
		M_3= 0.87	M_4 = -1.0	θ_2 = 0.00
Cantilever Beam Example				
0.11	0.10	M_1 = 1.0	M_2 = 0.62	δ_1 = 1.00
Hinged – Roller - Roller Beam Example				
1.69	1.68	M_1 = 1.0	M_2 = -1.0	δ_1 = -0.85
Hinged – Roller – Fixed Beam Example				
2.10	2.09	M_1 = 1.0	M_2 = -1.0	δ_1 = 0.37
		M_3 = 0.52		

14.4 EXAMPLES OF BUCKLING ANALYSIS OF TRUSSES

Two problems of plane truss, one having two members and other having six members, are shown in **Fig 14.4**. The length L of vertical and horizontal members are considered as 1m with cross sectional dimensions of each one as 0.01m x 0.01m. The truss is made of mild steel having E as 2.01 x 10^8 kN/m^2. Using the desired matrices and following the steps outlined above, first of all the critical buckling load is evaluated and then other results are worked out. For the example of two member truss, the value of critical load is found as 14000 kN against 14090 kN based on exact solution and 14142 based on FEM based solutions **[104]**. Results obtained for the buckling load, internal forces and nodal displacements for the two truss examples are reported in **Table 14.2.**



a) Two Member Truss **b) Six Member Truss**

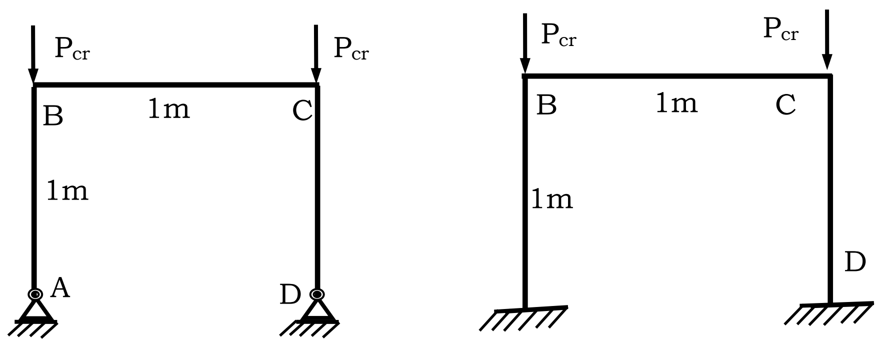
Fig 14.4 Plane Truss Examples

Table 14.2 Results for Plane Truss Examples

Buckling Load (P_{cr}) x 10^5 kN		Internal Forces (F) (kN)		Nodal Disp. (δ)
IFM	FEM			
Two Member Truss Example				
0.14	0.1414	F_1 = 1.0	F_2 = 0.0	δ_1= 2.00
				δ_2 = 0.00
Six Member Truss Example				
0.018	0.020	F_1 = 1.0	F_2 = -0.7	δ_1 = 0.414
				δ_2 = 0.000
		F_3 = 0.8	F_4 = 0.8	δ_3 = 0.002
				δ_4 = 0.00
		F_5 = 0.8	F_6 = 0.8	δ_5 = 0.001

14.5 EXAMPLES OF BUCKLING ANALYSIS OF FRAMES

Total four plane frame examples are considered here with sway and non-sway conditions. Two frames have pinned connection at each column base while the rest two have fixed column bases as shown in **Fig. 14.5**. The length of each member is considered as 1m with the cross sectional dimensions of each member as 0.01m x 0.01m and E as 2.01 x 10^8 kN/m^2. These hinged footed and fixed footed frame examples are analysed here under sway and non-sway conditions and results obtained are reported in **Table 14.3.** Results are compared with the FEM based solution **[104].**



(a) Hinged Footed **(b) Fixed Footed**

Fig. 14.5 Plane Frame Examples

Table14.3. Results for Plane Frame Examples

Buckling Load (P_{cr}) kN		Internal Moment (M)		Nodal Disp. (δ)
IFM	FEM			
Non Sway – Hinged - Hinged Frame Example				
2.5	2.45	M_1 = 1.0	M_2 = -1.0	θ_1 = -1.00
		M_3 = -1.0	M_4 = 1.0	θ_2 = 1.00
Non Sway- Fixed – Fixed Frame Example				
4.42	4.4	M_1 = 1.0	M_2 = -2.0	θ_1 = 1.00
		M_3 = 2.0	M_4 = -1.98	θ_2 = 0.08
		M_5 = -1.98	M_6 = 0.97	
Sway – Hinged – Hinged Frame Example				
0.31	0.305	M_1 = 1.0	M_2 = -1.00	θ_1 = 0.03
		M_3 = -0.9	M_4 = -0.9	θ_2 = -0.02
				δ_3 = 0.09
Sway – Fixed – Fixed Frame Example				
1.27	1.245	M_1 = 1.0	M_2 = -2.43	θ_1 = -0.31
		M_3 = 2.43	M_4 = -2.42	θ_2 = -0.29
		M_5 = -2.43	M_6 = -1.0	δ_3 = 1.03

14.6 SIGNIFICANCE OF PLATE BUCKLING

Linear Elastic Stability Analysis (LESA) is generally defined in the literature as an approach in which calculation is done for intensity of applied in plane loading, which is parallel to neutral plane. The internal distribution of orthogonal moment induced and possible nodal displacements at any point in the isotropic plate are again considered as independent for secondary linear analysis which is carried out after calculation of critical loading. Actually, practical aspects associated with the uncertain buckling based collapse involve a nonlinear aspect of instability and associated post buckling behavior with large amount of inelastic deformation. However, even in connections, LESA thoroughly describes the complete circumstances of failure, which are of design importance for number of thin structural forms. i.e. Naval and Aeronautical structures. It also furnishes the fundamental basis for large technical content of practical aspects of design methodology, even where nonlinear phenomena must be taken into account to define accurately the magnitude of load that causes failure. Thus, finally the complete form of the solution is generally provided by Linear Analysis procedure only. Three well-known classical methods by which elastic critical load can be calculated are: (i) Equilibrium Method, (ii) Energy Method, and (iii) Dynamic Method. In the present chapter, however, integrated force method is proposed for the buckling analysis of plates.

14.7 FORMULATION FOR PLATE BUCKLING

14.7.1 Element Equilibrium Matrix [Be]

The variational functional is evaluated as a portion of IFM functional which yields the basic elemental equilibrium matrix [B_e] as follows:

$$U_p = \int_D \left\{ M_x \frac{\partial^2 w}{\partial x^2} + M_y \frac{\partial^2 w}{\partial y^2} + M_{xy} \frac{\partial^2 w}{\partial x \partial y} \right\} d_x d_y \ = \{M\}^T \{\epsilon\}\ ds \qquad \text{... (14.18)}$$

where, $\{M\}^T = (M_x, M_y, M_{xy})$ are the internal moments and $\{\epsilon\}^T =$ $\left(\frac{\partial^2 w}{\partial x^2}, \frac{\partial^2 w}{\partial y^2}, 2\frac{\partial^2 w}{\partial x \partial y}\right)$ represents the curvatures corresponding to each internal moment.

A four noded, 12 displacement and 9 force degrees of freedom rectangular element of thickness t with dimensions as 2a x 2b along the x and y is considered for modeling the problem. The complete formulation for the rectangular plate bending element is already given in the Chapter 5.

As explained earlier, the element equilibrium matrix is calculated by

$$[B^e] = \int_s [Z]^T[Y]ds \qquad \ldots (14.19)$$

where [Z] = [L][N] where [L] is the matrix differential operator matrix, [N] is the displacement interpolation function matrix and [Y] is the matrix of force interpolation function matrix. Substituting necessary matrices and integrating gives a non symmetrical equilibrium matrix $[B^e]$ of size ddof x fdof. The matrix $[B^e]$ should have full row rank as mathematical property, which is required for further calculations.

14.7.2 Elemental Flexibility Matrix $[G^e]$

The basic elemental flexibility matrix for isotropic material is obtained by discretizing the complementary strain energy which gives

$$[G^e] = \int_s [Y]^T[D][Y]\, dxdy \qquad \ldots (14.20)$$

where, [Y] is force interpolation function matrix and [D] is material property matrix. Substituting these matrices and integrating provides $[G^e]$ matrix which is a symmetrical matrix of size fdof x fdof.

14.7.3 Global Compatibility Matrix [C]

The compatibility matrix is obtained from the deformation displacement relation ($\{\beta\} = [B]^T\{X\}$). In DDR all the deformations are expressed in terms of all possible nodal displacements and the ‘r’ compatibility conditions are developed in terms of internal forces i.e., F_1,------ F_{2n}, where '2n' is the total number of internal forces in a given problem. The global compatibility matrix [C] can be evaluated by multiplying the compatibility matrix and the global flexibility matrix.

14.7.4 Geometric Stiffness Matrix [Kg]

Figure 14.6 shows a rectangular plate of thickness 't' subjected to inplane compressive forces acting along the neutral plane.

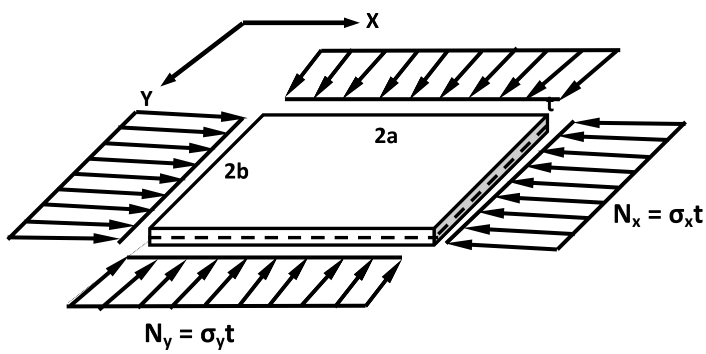


Fig. 14.6 Plate Under In-plane Forces

The geometric stiffness matrix for the RECT_9F_12D element can be written as

$$[K_g^e{}_{(x\text{-}x)}] = \int_S [N'_x]^T[N'_x]\, \sigma_x t \; dxdy \qquad \ldots (14.21)$$

where, N'_x is vector of size (12 x 1) developed by differentiating with respect to x, $\sigma_x t$ is the inplane load acting along neutral plane. Integrating within the domain gives the geometric stiffness [Kg^e] as follows; $[K_{g^e\,(x-x)}] =$

$$Pcrit\begin{bmatrix} \frac{46b}{105a} & \frac{b}{15} & \frac{11b^2}{105a} & -\frac{46b}{105a} & \frac{b}{15} & -\frac{11b^2}{105a} & -\frac{17b}{105a} & \frac{b}{30} & \frac{13b^2}{210a} & \frac{17b}{105a} & \frac{b}{30} & -\frac{13b^2}{210a} \\ & -\frac{8ab}{45} & 0 & -\frac{b}{15} & \frac{-2ab}{45} & 0 & \frac{b}{30} & \frac{-ab}{45} & 0 & \frac{b}{30} & \frac{4ab}{45} & 0 \\ & & \frac{4b^3}{105a} & -\frac{11b^2}{105a} & 0 & -\frac{4b^3}{105a} & \frac{13b^2}{210a} & 0 & \frac{b^3}{35a} & \frac{13b^2}{210a} & 0 & -\frac{b^3}{35a} \\ & & & \frac{46b}{105a} & -\frac{b}{15} & -\frac{11b^2}{105a} & \frac{17b}{105a} & -\frac{b}{30} & -\frac{13b^2}{210a} & -\frac{17b}{105a} & -\frac{b}{30} & \frac{13b^2}{210a} \\ & & & & \frac{8ab}{45a} & 0 & -\frac{b}{30} & \frac{4ab}{45} & 0 & \frac{b}{30} & -\frac{ab}{45} & 0 \\ & - & & & & \frac{4b^3}{105a} & \frac{13b^2}{210a} & 0 & -\frac{b^3}{35a} & -\frac{13b^2}{210a} & 0 & \frac{b^3}{35a} \\ & & & & & & \frac{46b}{105a} & -\frac{b}{15} & -\frac{11b^2}{105a} & -\frac{46b}{105a} & \frac{b}{15} & -\frac{11b^2}{210a} \\ & & & & & & & \frac{8ab}{45a} & 0 & \frac{b}{15} & \frac{-2ab}{45} & 0 \\ & & & Sym & & & & & \frac{4b^3}{105a} & \frac{11b^2}{105a} & 0 & \frac{4b^3}{105a} \\ & & & & & & & & & \frac{46b}{105a} & \frac{b}{15} & -\frac{11b^2}{105a} \\ & & & & & & & & & & \frac{8ab}{45a} & 0 \\ & & & & & & & & & & & \frac{4b^3}{105a} \end{bmatrix} \quad \ldots\ (14.22)$$

Where, P_{crit} = (σ_xt) = a critical inplane-force considered corresponding to buckling of plates.

14.8 EXAMPLES OF BUCKLING ANALYSIS OF PLATES

Total seven examples of mild steel plate are considered here to validate the proposed method under different boundary conditions i.e., Simply supported (S), Clamped (C), Free (F) and their combinations as shown in **Fig. 14.7**. In all problems, the load is considered in x-x direction only. Each plate is having geometrical dimensions as 4000 mm x 4000 mm x 200 mm. E = 2.01 x 10^{11} N/m^2 and ν as 0.23. Depending upon the type of symmetry, either quarter plate is discretized in 2x2 mesh or half plate is discretized in 4 x 2 mesh as shown in **Fig. 14.7**.

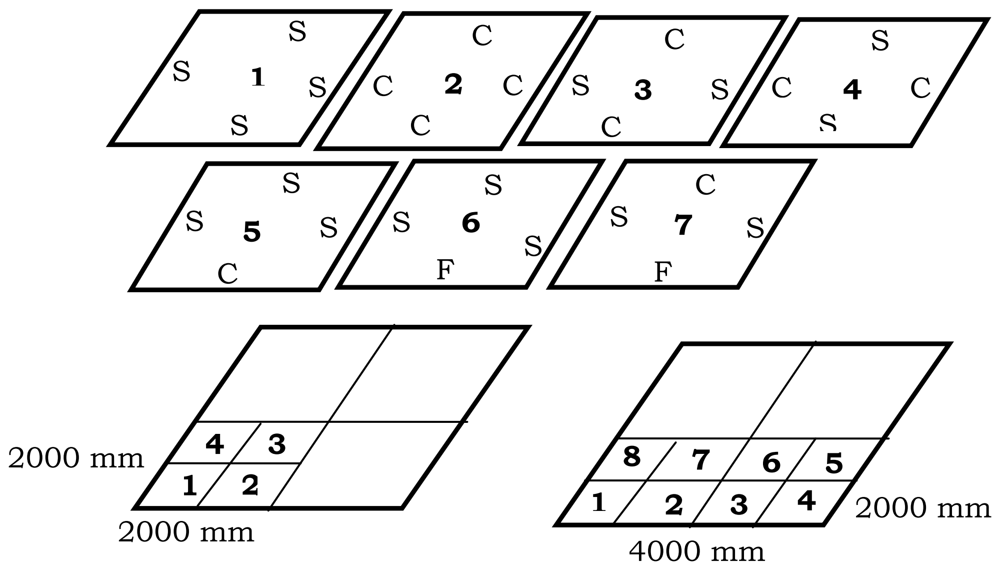


Fig 14.7 Plate Buckling Problems with Different Boundary Conditions

Steps for finding the solution to any plate buckling problem are the same. Here the steps are explained with reference to a simply supported plate subjected to inplane forces in the x-direction only.

Step 1- Develop global equilibrium matrix [B]: A four-noded rectangular element (2a x 2b) with 12 ddof and 9 fdof is used for discretizing the problem in to four elements. The elemental [B^e] matrix is obtained as explained above by substituting a = 0.5m, b = 0.5m. An assembled size of global equilibrium matrix for quarter plate will be of size 12 x 36.

Step 2- Develop global compatibility conditions [C]: The compatibility matrix for all discretized elements is obtained from the displacement deformation relations (DDR) i.e. $\beta = [B]^T\{\delta\}$. In the DDR, 36 deformations which correspond to 36 force variables are expressed in terms of 12 displacements. The problem requires 24 compatibility conditions [C] that are obtained by using auto-generated Matlab software by giving input as upper part of the global equilibrium matrix.

Step 3- Develop global flexibility matrix [G]: The flexibility matrix for the problem is obtained by diagonal concatenation of the four element flexibility matrices as;

$$[G] = \begin{bmatrix} G_{e^1} & & & \\ & G_{e^2} & & \\ & & G_{e^3} & \\ & & & G_{e^4} \end{bmatrix} \qquad \ldots (14.38)$$

Step 4- Develop global geometric stiffness matrix [K_{gc}]: The global geometric stiffness matrix is worked out by assembling the four element geometric matrices [K_g^1], to [K_g^4]. Considering standard stiffness based assembly procedure a global geometric stiffness matrix is developed having number of rows and columns for S-S-S-S case = global DOF for quarter symmetry = 12. The remaining rows and columns for [K_{gc}] are filled by zeros. Thus, the complete matrix is of size (36 x 36).

Step 5- Calculate buckling load (P_{crit}): The global CC matrix of size 24 x 36 is obtained after normalizing with respect to components of [B_e] of size 12 x 36. It is developed by multiplying [C] matrix of size 24 x 36 by global flexibility matrix of size 36 x 36. Finally by concatenations one can get the complete global equilibrium matrix of size 36 x 36. Thus substituting all necessary matrices in Eq. (14.4), one can get solution as eigen vector of size (12 x 1), corresponding to 12 global displacements of quarter plate of S-S-S-S case. The critical load is found as 3.681×10^5 kN against the exact value of 3.47×10^5 kN [ ].

Step 6: Calculate force mode shape {F} (Internal moments): The internal unknowns are auto calculated in Matlab based eigen value analysis by typing [F, Pcrit] = eig (Smatrix, KJG) where [F] is the matrix of size 36 x 36, Pcrit is the diagonal matrix of size 36 x 36 in which first 12 elements and

KJG is the product of global geometric stiffness matrix [K_g] of size 36 x 36, Jmatrix and [G] which is the global flexibility matrix of size 36 x 36. Taking sixth column of [F] matrix which is corresponding to minimum critical load and substituting in moment equation, all moments at points I, F, C, G are worked out as per each segment 1, 2, 3 and 4 (F_1 to F_9), (F_{10} to F_{18}) etc., as shown in **Fig 14.8.** The values normalized with respect to point C are depicted in **Table 14.4.**

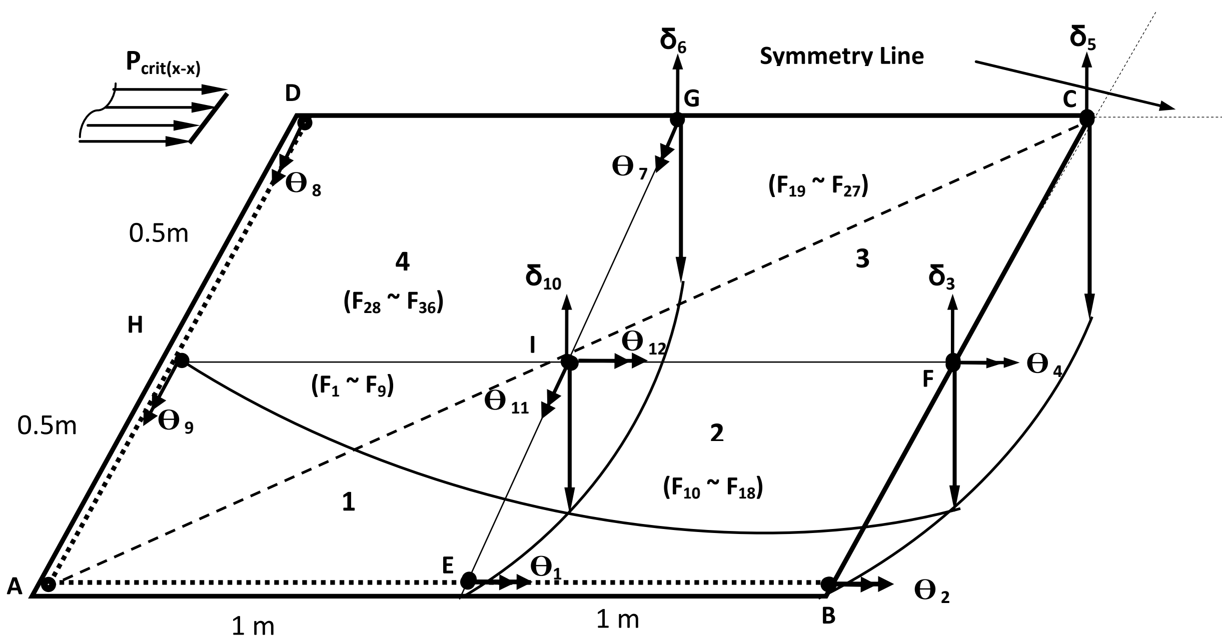


Fig 14.8 Four Element Discretization Scheme

Step 8: Calculate displacement mode shape (δ): Substituting values of all internal forces (F_1 to F_{36}) for lowest critical buckling load in Eq. (14.3) gives the normalized displacements with respect to $δ_5$ as follows.

Θ1 = -0.6651	Θ2 = -1.136	δ3 = 0.992	Θ4 = -0.7718
δ5 = 1.4313	δ6 = 1.0000	Θ7 = 0.8506	Θ8 = -1.0611
Θ9 = -0.7956	δ10 = 0.6553	Θ11 = 0.5076	Θ12 = 0.6377

Table 14.4 Normalized moments for S_S_S_S Case

Moments	Normalized Moments with Respect to node C			
	Node I	Node F	Node C	Node G
M_x	0.2854	0.9935	1.000	0.5862
M_y	0.09244	0.6321	1.000	0.6931
M_{xy}	0.8668	3.5133	1.000	3.6198

Similarly linear buckling analysis is carried of all other plate problems by using either one way or two way symmetry depending on the boundary conditions. **Table 14.5** shows IFM based critical load and exact solution **[99]** for the plates subjected to uniaxial loading.

Table 14.5 Buckling Loads for Different Boundary Conditions

Case	Buckling Load (Pcr) x 10^5 kN	
	IFM	Exact
C_C_C_C	8.53	8.10
C_S_C_S	8.00	7.81
S_C_S_C	6.098	5.89
C_S_S_S	5.37	5.13
F_S_S_S	1.31	1.24
F_S_C_S	1.48	1.46

A simply supported plate subjected to biaxial loading is also considered for buckling analysis. Because of two way symmetry only quarter of the plate is discretized into 2x2 mesh. Global geometric stiffness matrix is calculated by considering N_x and N_y of equal intensity i.e. by considering load ratio $\lambda = 1$. IFM based analysis gives the value of critical load as 1.78 x 10^5 kN whereas the exact value is 1.73 x 10^5 KN **[99].**

14.9 DISCUSSION OF RESULTS

- ❖ Total six problems of beam are solved for calculating the critical buckling loading, but only minimum load is considered for further calculation of non-dimensional internal moments and nodal displacements. With two element discretization for all the problems, result for critical load is found in good agreement with Euler's buckling load. The value for cantilever beam is found slightly higher which may be due to flexibility of free end. The non-dimensional moments are auto-calculated using Matlab module which provide an approximate idea about values of moments at the intermediate points with respect to unit value at one point. Non-dimensional nodal displacements also worked which give an idea about displacements at other points with respect to unit ordinate at far end for the first mode of buckled form.

- ❖ Two problems of pin jointed plane truss are attempted in which critical load is worked out using buckling form of any one of the sensitive members of the pin jointed truss. Result obtained for two member truss is found to match exactly with the FEM result whereas the result for 6 member truss is found slightly lower than the FEM solution for critical load.
- ❖ Two problems of plane frame are considered for calculating the first mode buckling load under sway and non sway conditions. Considering each member of the frame as one element, results obtained for critical load for all the frame problems are found within 2 to 4% to the value obtained using displacement based finite element method
- ❖ RECT_9F_12D element which is used to solve 7 plate buckling problems under uniaxial loading is found to provide critical load quite near to the exact value with 2x2 discretization in case of one way symmetry and with 4x2 discretization in case of dual symmetry. In case of simply supported plate the moments are calculated for all the four elements and are normalized with respect to the value at centre of plate. As the in-plane loading is in the x-x direction, the value of Mxx are found higher than the values of Myy.
- ❖ In case of a fully simply supported plate subjected to biaxial loading also, with 2x2 discretization, the value of critical load found using IFM is found quite close to the exact value.

CHAPTER 15

CONCLUSIONS AND CONTRIBUTION

15.1 SUMMARY

The two most fundamental approaches for analysing the discrete structures are the force (Flexibility) and displacement (Stiffness) methods. The relative merits of these methods were discussed intensively during the early evolution of computer automated structural analysis. The displacement method won out for computer automation whereas the classical force method proved inconvenient to automate and computationally more costly than the displacement method. Since then a number of attempts have been made to improve the classical force method; the integrated force method is the outcome of the one of such attempts.

The integrated force method was introduced by Patnaik and his team at Ohio Aerospace Institute with the aim to make its automation as convenient as the displacement method for the analysis of framed structures and to make it as versatile as the finite element method for the analysis of continuum structures. The same method was further investigated in the present work to find its suitability and to look at any possibility of further development.

Using the concepts of IFM, formulations were developed for static and dynamic analysis of 6 types of framed structures. After the computer implementation of the solution steps, results obtained for a variety of problems were compared with the known solutions. A dual form of IFM, called as DIFM, was also developed and computer implemented.

Rectangular, triangular and curved elements were then developed to facilitate analysis of continuum structures such as plane stress, plane strain and plate bending problems. Pre-, Main-and Post-processors were developed in VB6 and VB.NET environment using where required the facilities available in Matlab software. Results obtained using these numerical and graphical processors for the static and dynamic analysis of rectangular plates were than compared with the available solutions. Plates were analysed considering variety of loading such as central point load, uniform lateral pressure, patch and hydrostatic loading under a variety of support conditions such as fully simply supported, fully clamped and their combinations. Plotting of moment contours and deflection profile was achieved through Matlab software.

In addition to isotropic plate problems, formulation and programs were also developed for linear elastic analysis of orthotropic plates. Dynamic analysis of orthotropic plate problems was also carried out and the results were compared with the available solutions.

Also, formulation was developed for IFM based analysis of circular and annular plates under lateral pressure and line loading along inner or outer periphery of annular plate. Computer implementation of the same was validated by comparing with the available solutions. Convergence study was also carried out.

Finally, IFM based formulation was developed for computer aided buckling analysis of framed structures and rectangular plates. Results obtained were presented in tabular form and the results were compared with the available solution.

15.2 CONCLUSIONS

1. A visual programming environment provides all features that are required to develop a graphical user interface. Here selection of VB6 and VB.NET environment has helped considerably in developing the menu driven programs with provisions for all the symbolic manipulations needed for the development of different types of matrices along with the capability to provide the output in quite attractive form.

2. The development of compatibility conditions is the most important part of the IFM based formulation. It is generated from displacement-deformation relation of the original structure without any reference to static or kinematic indeterminacy. However, It is always necessary to check the compatibility conditions which could be easily verified mathematically by $[B][C]^T$ resulting in a null matrix.

3. The IFM is a true force method as it is free from the concepts of selecting the redundants and basic determinate structure, due to which an automation of the IFM is possible. The methodology adopted in IFM is unusual but attractive. The inclusion of secondary effects in terms of temperature variation, support settlement and prestrain in the analysis is also easy.

4. For small scale structures involving combination of various rigidities to resist different types of forces, finding the solution may be difficult in terms of development of CCs. It is simplified in the present work by implementing an algorithm for auto-generation of the compatibility conditions where all the CCs consisting contributions of all deformations based on nodal displacements satisfies $[C]\{\beta\} = 0$. This also helps in dealing with sparse matrix effectively.

5. IFM bestows appropriate emphasis on equilibrium equations and compatibility conditions, whereas stiffness method emphasizes only on equilibrium equations. Therefore, the results obtained using IFM are more accurate, in some of the cases, compared to other displacement based numerical methods.

6. Under transverse loading for beam type of structures two major unknowns are shearing forces and moments. In IFM shearing forces are directly represented in terms of length of each member, which is a numerical constant. Hence the major unknowns are moments only

which reduces the computational effort required for finding the solution in Integrated Force based methodology.

7. IFM based procedure for plane framed structures, considering and neglecting axial deformation in members, does not require major alteration in basic equilibrium matrix [B]. The axial force in element matrix contributes only positive or negative values depending upon the loading without effecting the number of unknowns and computation procedure which has been clearly demonstrated with the help of an example. Also, as IFM concentrates on only internal unknowns of the members, no transformation is required for inclined members.

8. Results obtained by using IFM are found to match fully, for most of the framed structure problems, with the solutions obtained by Flexibility Technique and commercially available STAAD.pro software.

9. The extension of integrated force method from plane structures to space structures is found quite straight forward. Also, as there is no need to make any transformation and as some of the internal actions are indirectly considered in IFM, the computational effort required for finding the solution is less.

10. The dual form of IFM (DIFM) is analytically equivalent to IFM and produces identical results for displacements and internal actions. Development of compatibility conditions manually in IFM, is cumbersome for being trial-and-error approach. It is totally removed in DIFM. Although programming of DIFM is easier compared to IFM, it requires some extra matrix operations to get the final solution.

11. The traditional displacement based finite element approach always uses the differentiation of an approximate displacement function to calculate stresses; which is totally removed in DIFM. Various necessary matrices i.e. Element Equilibrium matrix [B_e], and Element Flexibility Matrix [G_e] are developed for all types of framed structures.

Computer implementation of the same is found to give same results as obtained by IFM, and therefore, only some demonstrative examples of DIFM were included in the present work.

12. IFM based formulation is also successfully extended to carry out frequency analysis of framed type structures by considering eigen values analysis. The complete mathematical procedure is found to be very straight forward and easy to understand. Different types of beams are analysed under different loading criteria i.e. Direct Nodal Lumping Mass (DNLM), Lumped Mass (LM) and Consistent Mass (CM) criteria. The DNLM predicts behavior of structural member under constant loading at different frequency values, which are found to match fully with the exact solution obtained based on simple Mass-Stiffness formulas. By using the frequency values internal modal moments and nodal displacements are directly obtained using the facility available in Matlab. The frequency values for LM and CM cases are also compared with the available solution in the literature where a good agreement is found.

13. Solutions for frequency values using DNLM, LM and CM approaches are obtained for two plane trusses and for the first frequency value internal forces and relative nodal displacements are worked out. Using the secondary parameters, one can guess generally the behavior of the structural members under the excitational frequency. In two panel type truss example the structure is found much stiffer and, therefore resulted in higher frequency values.

14. The solution obtained for fixed footed frame problem, under different lumping criteria is found to match fully with the exact solution available in the literature. As first frequency is in horizontal direction, where axial effect in the beam is neglected, it requires less amount of the excitation force. As it is a direct measure of the first modal frequency, it indicates that the portal frame is more flexible in

horizontal direction compared to the vertical direction. Thus, it gives direct insight to lateral stiffness of a multistoreyed building under seismic excitation. The values of internal moments and nodal displacements are found to match fully with the overall deflection pattern of the portal frame under first frequency value. The inbuilt procedure provided by Matlab for calculating the internal forces and moments based on frequency values is found to be advantageous against the development of a separate routine for finding the solution of homogeneous equations.

15. For a four membered grid example, solved for frequency analysis, there are two major displacements i.e. vertical displacement and two orthogonal rotations corresponding to bending and twisting. The vertical direction under excitation proves to be weaker hence smaller value of frequency is necessary for developing the first mode. The moments and nodal displacement at the central joint and at the end of the each member are worked out. The values are found to be consistent with the support conditions.

16. Although IFM formulation provides strong futuristic support to flexibility method, it has certain disadvantages such as the approach for different types of structures is not the same. Releasing procedure for internal forces in a pin jointed structure may cause instability. Also, in some of the cases, IFM does not provide symmetric and banded matrices.

17. IFM gives stress based behavior by calculating the internal moments rather than the displacements for various types of structures under dynamic loading. It also provides an overall idea regarding the time dependent stress animation and failure prone zone at higher modes of vibration, which is strongly desirable for uncertainty analysis particularly for lightweight structures.

18. IFM is similar to the Completed Beltrami-Michells Formulation (CBMF) in elasticity with the governing equation as [S]{F} = {P} in which equilibrium Cauchy operator is analogous to [B] matrix, Compatibility operator of St. Venant controls the strain which is analogus to [C]{β} = 0, and Hooke's constitutive matrix is related to [G] matrix. Thus, the IFM has a capability to obtain the exact stress solution because of classical mechanics background.

19. The generation of CCs by hand calculations is always a trial- and-error approach which is not only time consuming but also laborious and tedious. Auto–generation of the compatibility conditions using Matlab software is facilitated in the present work by developing an algorithm and linking it to the programming languages VB6 and VB.NET. The generation of field and boundary compatibility conditions for any large scale continuum problem with finer discretization may require more time, where the suggested scheme may prove very efficient.

20. When the development of basic equilibrium matrix [Be] and flexibility matrix [Ge] is carried out directly using Matlab based integration/differentiation technique, it has an advantage of minimizing the computer coding. Also, the overall time required for finding the solution for complex problems can be minimized by directly supplying the equilibrium and flexibility matrices through a text file.

21. The IFM based plane stress element for deep cantilever beam problem shows lower bound convergence for nodal displacements and internal stresses with 2, 3, 4, and 5 element discretization schemes. With 5 elements the nodal displacements obtained using IFM and DIFM are found in close agreement to that of the exact displacement.

22. IFM based RECT_5F_8D element shows good performance in calculation of internal stresses as well as nodal displacements under plane stress condition. The computation time required compared to

the well known Finite Element Method is less for the same degree of accuracy. For a propped deep cantilever beam subjected to pure bending, with 2 x 2 discretization of quarter plate, IFM gives Displacement Ratio as 0.998 with respect to exact value while its value is 1.045 with respect to FEM solution.

23. A triangular element (TRI_3F_6D) is used for finding the solution to a deep propped cantilever plate subjected to a point load; after verifying the properties of the triangular element in terms of full row rank, shear locking criteria etc. The solution obtained is found to be matching with the available FEM based solution. Thus, the suggested triangular element (TRI_3F_6D) can be safely used for solving similar the plane stress problems of similar types

24. CURV_5F_8D element is developed and checked for its validity using curved beam example subjected to a moment at free end. Polar coordinate system is explored for the development of the formulation. Results obtained for stresses and displacements are found within 6 % of that available in the literature.

25. For a plane strain problem of box culvert which is attempted using RECT_5F_8D element with appropriate [D] matrix in the formulation, results are found in good agreement with the solution available in the literature.

26. The proposed Airy stress based functions can handle different types of 2D continuum problems under different types of loading and support conditions. However, improper choice of stress interpolation function may make discretized model more rigid or flexible and may provide accordingly the lower or upper bound solution.

27. For large size problems, involving large number of unknowns, if the numerical difference between the components of equilibrium and compatibility part is more, it may change the displacement to

uncertain values. So, before proceeding further a normalization of the compatibility conditions is strongly recommended. However, it does not make much difference in the calculation of internal force vector {F}.

28. In IFM, the percentage variation in solution is controlled by compatibility conditions rather than equilibrium equations. A minor modification in coefficients of compatibility conditions leads to a large amount of error which may be lower or upper bound with respect to the exact solution.

29. Static analysis is carried out of square plate bending problems under central point loading and uniform lateral pressure for simply supported and clamped boundary conditions by using RECT_9F_12D element. Considering (2 x 2), (3 x 3), (4 x 4) and (5 x 5) discretization schemes for the lower left quadrant, due to dual symmetry, the solution for central deflection using IFM based formulation is found to converge from the higher side while central moments are found to converge from lower side with finer discretization. The 5 x 5 discretization used for the remaining plate bending problems is found to provide quite good results. Values of the moments at the boundaries are also found to be consistent with the support conditions. Results obtained for rectangular plate bending problems having aspect ratio as 2 are also found within acceptable limit when compared with the available results based on the energy method. Finer discretization is required in such cases to get more accurate results.

30. The Matlab based CC developer gives 24 number of compatibility conditions using only 12 equilibrium equations which makes the global equilibrium matrix a square matrix. The value of z.cTrasposeB is of the order of 10^{-13} to 10^{-14} which is almost equal to zero that is very much needed in IFM based formulation. Thus, $[B][C]^T = 0$ is satisfied by integrated force based methodology.

31. The 2D and 3D graphics capability of Matlab enabled plotting of moment contours as well deformation patterns for a variety of plate bending problems. The tool option of the matlab graphics module also allows to find necessary number of contours required in the specified domain. All these features have been found very useful in proper representation and interpretation of the results.

32. For patch and Hydrostatic loadings, with 5 x 5 and 10 x 5 discretization schemes respectively, results are found in good agreement with the available results. Different 2D and 3D plots are also included which are also found to satisfy boundary condition of plates as well as maxima /minima criteria of the deflection pattern.

33. Total four square plate bending problems are attempted for IFM based frequency analysis by using Lumped and Consistent Mass approaches. A force based eigen value approach is used instead of traditional displacement frequency calculation. After convergence study for first frequency for simply supported plate, 5 x 5 discretization scheme was adopted, which is found to give satisfactory results. The internal forces based on each frequency are readily obtained by simply typing [Fmatrix, ω] = eig(Smatrix, MJG) in Matlab module. The normalized nodal displacements with respect to the value at the centre of plate for the first frequency provide a general idea about the first modal deflection pattern. Matlab based surf module has been successfully explored, with necessary built in features, for this particular purpose.

34. The values of natural frequencies for simply supported plate using lumped mass criteria are found to be converging from the lower side with respect to analytical solution, while using consistent mass concept with same discretization pattern is found to be converging from the upper side. Frequency calculation using consistent mass includes contribution of rotary inertia in off diagonal terms which gives

values more than the lumped mass approach which has only diagonal terms with all off diagonal terms being zero.

35. For a clamped plate with 5 x 5 discretization first frequency is found to have higher value due to support constrains against rotation in the first four deflection patterns drawn using Matlab based surf module. The first modal deflection pattern shows very less value at the centre of the plate; which also follows for the rest of the modes. Frequency ratio with respect to exact value is found equal to unity. For the first modal pattern, moment values and nodal displacements along the diagonal lines are normalized with respect to the values at the centre. These are also found comparable with respect to deflection pattern with hogging nature for the elements nearer to the support. Frequency value for the clamped plate under consistent mass is also found to be matching with the available result.

36. Two problems of mixed boundary conditions are attempted using lumped mass criteria. The value of FR is found to differ slightly from unity but for higher frequencies it is found to be matching with the exact solution. Along x-x axis, being clamped, some resisting moment is indicated whereas along y-y axis, for being simply supported edge, zero value of moments is indicated. The second and third mode indicate different values compared to the first in the second case having simply supported and clamped boundary conditions all over. The combination of simply supported and free edges is found more flexible hence it required less amount of excitation energy and thereby gives lesser value of first modal frequency. The Frequency Ratio (FR) shows more deviation with respect to unit value but again for higher mode it re-converges to the unit value. This is possible due to each nodal mass is having slight deviation except the vertical.

37. Total ten orthotropic plate problems are analysed under central point loading and uniform lateral pressure considering different

combinations of boundary conditions. In all the cases, the comparison of calculated deflection values and moments indicated a good agreement with the available classical solutions. Moment contours and elastic curves plotted for all the plate problems are found to help in visualizing the behaviour of the plate under different type of support and loading conditions.

38. As the modulus of elasticity Ex of the Fiber Reinforced Composite plate is higher than Ey, the values of moment along x-direction are found more than the values in the y-direction. However, results for moment under uniform lateral pressure are found more closer to the exact value compared to the moments obtained under central point loading. In all the orthotropic plate problems also moment and deflection values are found consistent with the support conditions.

39. The Matlab based module for CC developer in case of orthotropic plates generated 249 compatibility conditions using 151 equilibrium equations within few seconds with null property check giving value between 10^{-12} to 10^{-14} which is quite remarkable. Plate examples of S_F_S_S_CPL and S_F_S_S_ULP show that due to free edge the maximum deflection is near the free edge as can seen in 3D plot. M_x shows less number of contours nearer to free edge compared to SS edge. Plate examples of F_S_F_S_CPL and F_S_F_S_ULP show behavior nearly analogous to SS beam having zero value at edges and maximum moment and deflection values at the centre of the plate while Mxy due to shearing effect show more number of contours near the support. F_C_F_C_CPL and F_C_F_C_ULP cases show stiff behavior compared to just previous case with lesser deflection values but with large number of moment contours.

40. Total eight problems of rectangular orthotropic plate are solved considering aspect ratio as 1.5 and 2.0. Compared to the square plate,

results are found less accurate which may be improved by finer discretization or by using higher order rectangular element.

41. The IFM based formulation developed in polar coordinates to deal with the axisymmetrically loaded circular plate bending problems is found quite effective. A large number of circular and annular plate problems are solved; results are found in close agreement with the available analytical solution. Deflection profile looks quite attractive particularly in case of annular plates subjected to uniform lateral pressure and line loading acting along inner or outer periphery of the plate.

42. The equations of IFM and DIFM are mathematically equivalent hence the natural frequencies, forces and displacements obtained by either of the methods are identical. Also, by providing minor changes in material matrix different types of problems related to different types of orthotropy can be easily handled. Results obtained for both static and dynamic analysis of orthotropic plates are found in close agreement with the available solutions.

43. The buckling analysis through IFM is facilitated by developing Geometric Stiffness matrix [Kg]. Various types of beam-column problems are successfully attempted using IFM based formulation where the result for critical load is found very close to Euler's critical buckling load.

44. Two problems of pin jointed plane truss are attempted using IFM based formulation to find critical load. In case of two member truss, the value of buckling load is found fully matching with the available solution whereas in case of six member truss it is found to differ by about 10 % from the FEM based solution.

45. Four problems of 3 member plane frame are considered for calculating the first mode buckling load under sway and non sway conditions of hinged and fixed footed portal frames considering each member as one

element. IFM based results are found in close agreement with those obtained using displacement based finite element method.

46. A number of plate buckling problems are attempted under uniform compressive loading in x direction. A variety of support conditions are considered. The result for critical buckling load is found to vary from 2 to 6 % from the exact solution. Thus the correctness of IFM based formulation of geometric stiffness matrix for RECT_9F_12D element is ascertained.

15.3 CONTRIBUTION

- For an analysis thru classical force method an auxiliary statically determinate structure and a set of redundant forces must be selected. To make the process of selection easily adapted to computer automation, several attempts have been made. However, all these procedures resulted in certain undesired properties which made them unattractive and led to the demise of the classical force method. Recently, an alternate method, named as Integrated Force Method, has been developed in which all independent forces are treated as unknown quantities that are calculated by simultaneous imposition of both equilibrium and compatibility conditions. In IFM compatibility conditions are generated from displacement deformation relation of the original structure without any reference to static or kinematic indeterminacy of the structure which is the most attractive but difficult part of the IFM formulation. In the present work the generation of the compatibility conditions by the use of Linear Independent Unknown Technique, which can be readily invoked through programming language of Matlab software, was proposed. It is found to make not only the complete process automatic but also found to develop successfully the required number of compatibility conditions even when the input matrix [Be] is sparse and the values in the matrix are randomly placed. Also, through the proposed procedure, the necessary condition of null property check is found to be satisfied for any feasible framed and continuum structure. For one of the examples, in the present work, it has been demonstrated to generate successfully 291 compatibility conditions form 151 equilibrium equations with null property check of the order 1.66 $x10^{-12}$ which is almost equal to zero.

- A variety of examples of skeletal structures included in the thesis has clearly demonstrated that IFM can be effectively used for the analysis of any types of framed structure. Concept of symmetry can be easily used. Secondary effects can also be taken into account. The method is as versatile as the stiffness method of analysis with an additional advantage of no transformation is required for finding the solution. Also, a force based frequency analysis of framed structures is achieved and a concept of using direct nodal lumping mass is introduced with screen shots of computer implementation which is efficiently managed in the present work by interlinking VB6 and VB.NET with some of the modules of the Matlab software
- A number of elements are formulated for the analysis of plane stress, plane strain and plate bending problems. Steps required for finding the solution are computer implemented. The validity of the formulation and its computer implementation is confirmed by comparing the results obtained with the available classical and/or numerical solutions. Because of the stress based approach, it is found to give zero moments at the simply supported edges unlike finite element method. As the displacement and stress field within the element are independently approximated, the user has full control to have desired accuracy which is one of the most important points in favour of IFM

- A modified form of IFM, called as DIFM, is also formulated and computer implemented. Both the methods are found to give identical results. DIFM has advantage of being similar to the stiffness method for framed structures and displacement based finite element method for continuum structures. However, it requires some extra matrix operations compared to IFM.
- A number of plate examples included in the thesis clearly indicated that the proposed IFM based element can also be easily used for the dynamic analysis of plates by using the suitable mass matrix. Versatility of the method is further proved by solving not only the examples of static and dynamic analysis of isotropic rectangular plates but also of the materially and structurally orthotropic plates.
- For variety of 1D and 2D elements, most of matrices are explicitly derived and a large number of small size problems are manually worked out to clearly explain the steps involved in finding the solution which will be certainly helpful in popularizing the integrated force method among the students and structural engineers. It is indeed the need of the hour to see some such developments are carried out for wider acceptability of IFM.
- Plotting of moment contours and deflection profiles based on the IFM based results is attempted first time in the thesis by judicious use of programming environment and Matlab software. These plots are found very useful in interpretation of results.
- A new element, named as CIRC_2F_4D, proposed in the thesis is also found to provide very good results for axisymmetrically loaded circular and annular plates under variety of support and loading conditions.
- Geometric stiffness matrix developed based on the integrated force based methodology is also found to provide very good results in case of buckling analysis of 1D (framed structures) and 2D (Plate) problems.

- Thus, the results presented in the present thesis for a variety of 1D and 2D problems, confirm that the integrated force method can be used successfully and efficiently in structural analysis and it has good potential to become a viable alternative to the displacement based stiffness and finite element methods of analysis.

15.4 FUTURE SCOPE

1. Although a number of 2D elements such as rectangular element, triangular element, curved element have been developed to deal with plane stress, plane strain and plate bending problems, development of higher order elements and their computer implementation is desirable for better representation of the problem and to get more accurate solution with relatively a coarser mesh.
2. Similarly development of 3D elements based on IFM is desirable to deal with both regular and arbitrary geometry problems of solid structures which can not be idealized either as 1D or 2D problems.
3. In the present work a line element, named as CIRC_2F_4D was developed in polar coordinates to deal with axisymmetric circular and annular plate problems. Development of a 2D sector element to handle asymmetric plate problems of circular geometry will be certainly an useful extension of the present work.
4. In the present work VB6, VB.NET and Matlab were judiciously used for numerical and graphical processing. Although Matlab software considerably reduced the length of program, it is desirable to carry out the complete processing in the same environment.
5. Separate program were developed to tackle different types of problems in the present work. Development of a general purpose program, although difficult, may be tried.
6. As the equations of IFM and DIFM are mathematically equivalent and hence the natural frequencies, forces and displacements obtained by either of the methods are identical. Therefore, in the present work,

only very few problems were attempted using DIFM. It is however desirable to explore it fully to find its merits and demerits against IFM and other contemporary methods.

7. IFM based formulation can be extended to take into account shear deformation theory of different order and thus some new elements can be developed by proper stress and displacement filed representation.
8. Materially orthotropic plate problems were handled in the present work. By using suitable [D] matrix, the application of IFM can be extended to deal with a variety of structurally orthotropic plate problems such as idealized stiffened orthotropic plate problems and problems of slab reinforced with equidistant ribs and problems of corrugated sheets.
9. In the present work, plate bending formulation was completely based on small deflection theory. One may think of development the formulation for large deflection analysis of thin plates.
10. Geometric stiffness matrix was developed in the present work to deal with buckling problems. Study of post buckling behavior of arches and thin plates may be thought of by extending the IFM based formulation.
11. The suggested integrated force method can be readily extended to steady state, transient and random vibration problems by making minor changes in the formulation. Also, it shows future scope for stochastic analysis which with direct stress mode facility may prove quite attractive.

REFERENCES

1. Gere J. M. and Weaver W., "Analysis of Framed Structures", NonStrand Co., Inc, New York, 1965.
2. Hayrettin, K., "Elementary Matrix Analysis Structures", Tata McGraw Hill Book Company, New York, 1974.
3. Shaw, F. S., "Virtual Displacements and Analysis of Structures", Prentice Hall, Inc., Eaglewood Cliff, New Jercy, 1972.
4. Argyris, J. N., "Cosmos – A Computer Program for Structural Analysis", Document D2-4513, The Boeing Company, Jan. 1962.
5. Robinson, J. S., "Structural Matrix Analysis for the Engineers", John Wiley and Sons, New York, 1966.
6. Patel C. M. and Patodi S. C., "Bending Force Method for Rigid Jointed Framed Structures", Journal of Institution of Engineers (India), Civil Engineering Division, Vol. 72, pp. 200-203, Jan. 1992.
7. Patnaik S. N., "An Integrated Force Method for Discrete Analysis", International Journal of Numerical Methods in Engineering, Vol. 6, pp. 237-251, 1973.
8. Cross H., "Analysis of Continuous Frames by Distributed Fixed End Moments", Proceeding of American Society of Civil Engineers, Vol. 96, pp. 1-15, 1932.
9. Courant R., "Variational Methods for the Solution of Problems of Equilibrium and Vibrations", Bulletin of American Mathematical Society, Vol. 49, No. 1-23, 1943.
10. Southwell R. V., "On Relaxation Method: A Mathematics for Engineering Science", Proceeding oF Royal Society of London of Mathematical and Physical Science, Vol. 184, No. 99, 1945.

11. Jenen V. P. and Dill D. G, "Matrix Methods of Analysis Applied to Wing Structures", Douglas Aircraft Co., Report E. S. 6840, Jan.1944.

12. Kempner J., "Application of a Numerical Procedure to the Stress Analysis of Stringer Reinforced Panels". National Advisory Committee for Aeronautics Report, L 5090, pp. 1- 32, 19, March 1945.

13. Benscoter S. U., "Numerical Transformation Procedures for Shear Flow Calculations", Journal of Aerospace Science, Vol. 13, No. 8, pp. 438, 1946.

14. Plunkett R., "A Matrix Method for Calculating Propeller Blade Moments and Deflections", Journal of Applied Mechanics, Vol. 16, No. 6, pp. 361, 1949

15. Falknheiner, "Systematic Analysis of Redundant Elastic Structures By means of Matrix Calculus", Journal of Aero Science, Vol. 20, No. 4, pp. 293, C 1953.

16. Levy S., "Computation of Influence coefficients for Aircraft Structures with Discontinuities and Sweepback", Journal of Aero Science, Vol. 14, No.10, pp. 547-560, Oct. 1947.

17. Langefors B., "Analysis of Elastic Structures by Matrix Transformation with Special Regards to Semimonocoque Structures", Journal of Aero. Science, Vol. 19, No. 10, pp. 451-458, July 1952,

18. Wehle L. B. and Lansing W., "A Method for Reducing the Analysis of Complex Redundant Structures to a Routine Procedure", Journal of Aero Science, Vol. 19, No.10, pp. 677, 1952.

19. Lang A. L. and Bisplinghoff R. L., "Some Results of Swept Back Wing Structural Studies", Journal of Aero Science, Vol. 18, No.11, pp. 705 - 717, Nov. 1951.

20. Langefors B., "Exact Reductions and Solution by Parts of Equations for Elastic Structures", SAAB TN 24, 1953, AMR 8 Rev. 2699, 1955.

21. Kron G., “Solving Highly Complex Elastic Structures in Easy Steps”, Journal of Applied Mechanics, Vol. 22, No. 2, pp. 235, 1955.

22. Denke R. H., “A Matrix Method of Structural Analysis”, Proc of 2nd U. S. National Congress on Applied Mechanics, ASMCE, pp. 445-457, June, 1954.

23. Denke P. H., “The Matrix Solution of Certain Nonlinear Problems in Structural Analysis”, Journal of Aero Science, Vol. 23, No. 3, pp. 231, 1956..

24. Levy S., “Structural Analysis and Influence Coefficients for Detla Wings”, Journal of Aero Science, Vol. 20, No.7, pp. 449, 1953.

25. Livesley R. K., “The Automatic Design of Structural Frame”, The Quarterly Journal of Mechanics and Applied Mathematics, Vol. 9, No. 3, pp. 257-278, 1956.

26. Argyris D. E., "Energy Theorems and Structural Analysis: A Generalized Discourse with Applications on Energy Principles of Structural Analysis Including the Effects of Temperature and Non-Linear Stress-Strain Relations", Aircraft Engineering and Aerospace Technology, Vol. 26 (10), pp. 347 – 356, 1954.

27. Argyris D.E. and Kelsey, S., "The Matrix Method of Structural Analysis and Some New Applications", Report of Aeronautical Research Council, R & M No. 3034, pp. 1-42, London, 1957.

28. Argyris J. N. and Kelsey S., “Energy Theorems and Structural Analysis”, Butterworth Scientific Publications, London, 1960.

29. Klen B., “A Simple Method of Structural Analysis”, Journal of Aero Science, Vol. 24, No.1, pp. 39, 1957.

30. Klen B., “A Simple Method of Structural Analysis Part II, Effects of taper and consideration of Curvature”, Journal of Aero Science, Vol. 24, No.11, pp. 813, 1957.

31. Kron G., “Equivalent Circuits for Elastic Fields”, Journal of Applied Mechanics, Vol. 11, No. 3, P A 149, 1959.
32. Bazant Z. P., “Analysis of Framed Structures Part I & II”,Applied Mechanics Surveys, Spartan Books, Washington, D. C., pp. 45-464, 1966.
33. Przemieniecki J. S., “Theory of Matrix Structural Analysis”, Tata Mc Graw- Hill, Inc. USA, 1968
34. Turner M. J., Cough R. W., Martin H. C. and Top. J.,“Stiffness and Deflection Analysis of Complex Structures “, Journal of Aeronautical Science, Vol. 23, No. 9, pp. 805-824, Sep. 1956.
35. Clough R. W., “The Finite Element in Plane Stress Analysis”, Proceeding of ASCE Congress on Electronic Computation, Pittsburgh, Sep. 1960.
36. Melosh R. J., “Basis for Derivation of Matices for Direct Stiffness Method”, Journal of American Institute of Aeronautical Astronomy, Vol. 1, No. 7, pp.1631-1637, July 1963.
37. Besseling J. F., “The complete Analogy between the Matrix Equation and the Continuous Field Equations of Structural Analysis”, International Symposium on Analogue Digital Technology Applications in Aeronautics, Belgium, Sep. 1963.
38. Venbeke F. B., “Upper and Lower Bound in Matrix Structural Analysis”, AGARD Report, Vol. 72, pp. 165-201, 1965.
39. Jones R. E., “A Generation of Direct Stiffness Method of Structural Analysis”, Journal of American Institute Aeronautical Astronomy, Vol. 2, pp. 821-826, 1964.
40. Pian T. H. H., “Derivation of Element Stiffness Matrices” Journal of American Institute of Aeronautical Astronomy, Vol. 2, pp. 576-577, 1964.
41. Zinkiewicz O. C. and Cheung Y. K., “The Finite Element Method in Structural and Continuum Mechanics”, McGraw-Hill, New Yrok, 1967.

42. Archer J. S., “Consistent Matrix Formulations for Structural Analysis Using Influence Coefficient Technique, Journal of American Institute of Aeronautical Astronomy, Vol. 64, pp. 488, June 1963.

43. Guyan R. J., “Reduction of Stiffness and Mass Matrices”, Journal of American Institute of Aeronautical Astronomy, Vol. 3, No. 2, pp. 380, Feb. 1965.

44. Hurty W. C., “Dynamic Analysis of Structural Systems using component modes”, Journal of American Institute of Aeronautical Astronomy, Vol. 3, No. 4, pp. 678-685, April. 1965.

45. Burton C. B. and Sokolowski, M., “On Certain Two Dimensional Application of the Couple Stress Theory, International Journal of Solid and Structures, Vol. 4, No. 1, pp. 15-29, 1966.

46. Felippa C. A. and Clough R. W., “A Refined Quadrilateral Element for Analysis of Plate Bending”, California University of Berkely, PDF url : ADA447741, 1968.

47. Gallagher R. H., “The Finite Element Method in Elastic Instability Analysis”, ISD/ISSC Symposium on Finite Element Techniques, Stuttgart, June 1969.

48. Gallagher R. H., “The Finite Element Method in Elastic Instability Analysis”, ISD/ISSC Symposium on Finite Element Techniques, Stuttgart, June 1969.

49. Neale B. K. Henshel R. D. and Edwards., “Hybrid Plate Bending Elements”, Journal of Sound and Vibration, Vol. 23, No. 1, pp. 101-112, 1972.

50. Kikuchi F., “Convergence of ACM Finite Elements Schemes for Plate Bending Problems, Publications RIMS, Kyoto University, Vol. 11, pp. 247-265, 1975.

51. Anderson, C. M., “Mixed Isoparametric Elements for Saint-Venant Torsion Problems”, Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 6, No. 2, pp. 195-218, 1975.

52. Gupta R. S. and Rao S. S., "Finite Element Eigen Value Analysis of Tapered and Twisted Timoshenko Beams", Journal of Sound and Vibration, Vol. 56, No. 2, pp.187-200, 1978.
53. Gallagher R. H. and Ding H., "Approximate Force Method Reanalysis Techniques in Structural Analysis", International Journal for Numerical Methods in Engineering, Vol. 21, No. 7, pp. 1253-1267, 1985.
54. Argyris J. and Lazarus T., "A Natural Triangular Layered Element for Bending Analysis of Isotropic, Sandwitch and Laminated Composite and Hybrid Plates", Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 109, No. 3-4, pp. 197-218, 1993.
55. Wilson E. L., "Automation of Finite Element Method – A Personal Historical View", Journal of Finite Elements in Analysis and Design, Vol. 13, No. 2-3, pp. 91-104, 1993.
56. Kosmtka J. B. and Friedman Z., "An Improved two noded Timoshenko Beam Finite Element", Journal of Computers and Structures, Vol. 47, No. 3, pp. 473-481, 1993.
57. Gupta K. K. and Meek J. K., " A Brief History of the Finite Element Method", International Journal for Numerical Method in Engineering, Vol. 39, pp. 3761-3774, 1996.
58. Fellipa, C. A. and Park, K. C., "A Variational Framework for Solution Method Developments in Structural Mechanics", Journal of Applied Mechanics, Vol. 65, No.1, pp.242-249, 1998.
59. Fellipa, C. A, "A Historical Outline of Matrix Structural Analysis: A Play in Three Acts", Centre for Aerospace Structures, CU-CAS-00-13, pp.1-15, 2000.
60. Kikuchi F., "Theory and Examples of Partial Approximation in the Finite Element Method", International Journal of Numerical Methods in Engineering, Vol. 10, Online Doc: doi: 10.1002/nme.1620100109, pp. 115-122, 2005.

61. Samuelsson A, and Zinkiewicz O. Z., "History of Stiffness Method", International Journal for Numerical Methods in Engineering, Vol. 67, pp. 149-157, 2006.

62. Ozotorun. N. K., "A Rectangular Finite Element Formulation", Journal of Finite Elements in Analysis and Design, Vol. 42, No. 12, 2006.

63. Kuoa S. R, Chih C. Y. and Chang J. R., " A Reliable Three Node Triangular Plate Element Satifying Rigid Body Rule Incremental Force Equilibrium Condition, Journal of Chinese Institute of Engineers, Vol. 29, No. 4, 2006.

64. Kaveh A., Koohestanis. K and Taghizadiehs, N., "Force Method for Finite Element Models with Indeterminate Support Conditions", Asian Journal of Civil Engineering (Building and Housing), Vol. 8, No. 4, pp. 389-403, 2007.

65. Lang C. and Yan Y., "The Reinforced Concrete Rectangular Plate with Complex Boundary supported", International Conference on Electrical Technology and Civil Engineering, ICETCE 2011, pp. 6340-6344, 2011.

66. Patnaik S. N. and Yadagiri S., "Design for Frequency by the Integrated Force Method", Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 16, pp. 213-230, April 1978.

67. Patnaik S. N. and Joseph K. T., "Generation of Compatibility Matrix in the Integrated Force Method", Journal of Computer Methods in Applied Mechanics and Engineering, Vol. 55, pp. 239-257, 1986.

68. Patnaik S. N., "The Integrated Force Method Versus the Standard Force Method", Journal of Computers and Structures, Vol. 22, No. 2, pp.151-163, 1986.

69. Patnaik S. N., “Analytical Initial Design for Structural Optimization via The Integrated Force Method”, Journal of Computers and Structures, Vol. 33, No. 1, pp. 265-289, 1989.

70. Nagabhusanam J. and Patnaik S. N., “General Purpose Program to Generate Compatibility Matrix for the Integrated Force Method”, Journal of AIAA, Vol. 28, No. 10, pp. 1838-1842, 1989.

71. Patnaik S. N, Hopkins D. A, Ajello R. A., and Berke L., “Improved Accuracy for Finite Element Structural Analysis via Integrated Force Method”, Journal of Computers and Structures, Vol. 45, No. 3, pp. 521-542, 1992.

72. Patnaik S. N, and Satish G., “Analysis of Continuum Using Boundary Compatibility Conditions via Integrated Force Method”, Journal of Computers and Structures, Vol. 34, No. 2, pp. 287-295, 1990.

73. Patnaik S. N. Hopkins D. A. and Kaljevic I., “Treatment of Initial Deformations in the Integrated Force Method”, Journal of Computer Methods in Applied Mechanics in Engineering, Vol. 140, pp. 281-289, 1997.

74. Patnaik S. N, Hopkins D. A, and Kaljevic I., “Development of Finite Elements for Two-Dimensional Structural Analysis Using the Integrated Force Method”, NASA Technical Memorandum 4655, 1996.

75. Patnaik S. N, Hopkins D. A, and Coroneos R. M., “Dynamic Analysis with Stress Mode Animation by the Integrated Force Method”, NASA Technical Memorandum 4729, 1997.

76. Patnaik S. N, Hopkins D. A, and Halford G. R., “Integrated Force Method Solution to Indeterminate Structural Mechanics Problems”, NASA Technical Memorandum TP – 207430, E-106810, March 2004.

77. Dhananjaya H. R, Pandey P. C and Nagabhushanam J., “Mindlin-Reissner Theory based New 4-Noded Quadrilateral Plate Bending

Element For Integrated Force Method”, Journal of Structural Engineering, Vol. 32, No. 3, pp. 179-194, Aug. 2005.

78. Dhananjaya H. R, Nagabhushanam J and Pandey P. C, “Application of Integrated Force Method to Laminated Composite Plates”, Proceedings of 1st National Conference on Recent Developments in Structural Engineering (RDSE 2005), pp. 109-116, Manipal, 2005.

79. Dhananjaya H. R. Nagabhushanam J., and Pandey P. C., “Automatic Generation of Plate Bending Finite Elements Equilibrium and Flexibility Matrices Using Integrated Force Method”, Proceedings of 1st National Conference on Recent Developments in Structural Engineering (RDSE 2005), pp. 67-72, Manipal, 2005.

80. Dhananjaya H. R., Pandey, P. C, and Nagabhushanam J., “Closed Form Solutions of Equilibrium and Flexibility Matrices for Rectangular Plate Bending Elements Using Integrated Force Method, Proceedings of International Conference on Recent Developments in Structural Engineering (RDSE 2007), pp. 256-274, Manipal, 2007.

81. Patnaik S. N, and Pai S. S., “Boundary Compatibility Condition and Rotation in Elasticity”, International Journal of Physical Sciences, Vol. 1, No. 2, pp. 081-084, Oct. 2006.

82. Dhananjaya H. R., and Nagabhushanam J., “Closed Form Solutions for Elements Matrices of 4-Noded Rectangular Plate Elements using IFM”, Electronic Journal of Structural Engineering, Vol. 10, pp. 1-9, 2010.

83. Logan D. L., A First Course of Finite Element Methods, 3rd Ed, Thomson Asia Ltd., USA, 2002.

84. Pian T. H. H. and Cheng D., “On the Suppression of Zero Energy Deformation Modes”, International Journal of Numerical Methods in Engg, Vol. 19. pp.1741-1752, 1983.

85. Tong P., and Pian T.H.H., “A Variational Principle and the Convergence of Finite Element Method Based on Assumed Stress Distribution”, International Journal of Solids and Structures, Vol. 5, pp. 463-472, 1969.

86. Ghallaghar R. H., Finite Element Analysis Fundamentals, Prentice hall inc., Englewood cliff, NJ, 1975.

87. Robinson J., Integrated Theory of Finite Element Methods. John Wiley and Sons, Inc. London. 1973.

88. Suo Z., Notes on Solid Mechanics of Plane Elasticity Part – I, ES 240, 2008.

89. Przemieniecki J. S., Theory of Matrix Structural Analysis. Dover Publications, Inc., New York. 1985.

90. Martin H. C., Introduction to Matrix Method of Structural Analysis. McGraw Hill Book Co., New York, 1966.

91. Kaljevic, Patnaik S. N, and Hopkins D. A, “Development of finite elements for two-dimensional structural analysis using the integrated force method”, Computers and Structures, NASA TM-4655, Vol. 59. No. 4, pp.691-706,1996.

92. Weaver W, and Gere J. M., “Matrix Analysis of Framed Structures”, CBS Publishers & Distributors, Delhi., 1986.

93. Paz M, Structural Dynamics Theory and Computation, CBS Publishers & Distributors, Delhi., 1987.

94. Bansal R. K., A text book on Strength of Material, Laxmi Publication, New Delhi, 2002.

95. Timoshenko S, Strength of Materials, Part I, Elementary Theory and Problems, D. Van Nostrand Company, 3rd Ed., New Jercy, 1955.

96. Desai C. S, and Abel, J. F. Introduction to the Finite Element Method, Affiliated East West Press Pvt. Ltd, New Delhi, 2002.

97. Timoshenko S. P, and Goodier J. N., Theory of Elasticity. Mc Hill Book Co. New York., 1970.

98. Love A. E. H., A Treatise on the Mathematical Theory of elasticity, 4^{th} Ed., Dover Publication. Inc., New York 1927.

99. Pilkey, W. D, Formulas for Stress, Strain and Structural Matrices, 2^{nd} Edition, John Wiley & Sons Inc New Jercy, 2005.

100. Szilard R, Theories and Applications of Plate Analysis, John Wiley & Sons Inc New Jercy, 2004

101. Chandrashekhara K, Theory of Plates, 1^{st} Edition, Universities Press (India) Ltd., India, 2001.

102. Bairagi N. K, A Text Book of Plate Analysis, 1^{st} Edition, Khanna Publishers, New Delhi, 1986.

103. Timoshenko, S. P and Krieger, S. W., Theory of Plates and Shells, 2^{nd} Edition, McGraw-Hill Book Co. Inc., New York, 1955.

104. Rajasekaran S, Finite Element Analysis in Engineering Design, S Chand, New Delhi, 2003.

LIST OF PAPERS PUBLISHED

1. Doiphode G. S., Patodi S. C. and Salat K. N.: "Integrated Force Method for Analysis of Framed Structures", National Conference on Emerging Technology and Development in Civil Engineering, Amaravati, pp. V-88 to V-94, March 2007.
2. Doiphode G. S., Salat K. N. and Patodi S. C.: "An Integrated Force Method: A Novel Tool for Indeterminate Structural Mechanics Problems", International Conference on Recent Developments in Structural Engineering, MIT, Manipal, pp.165 (pp 248-255 e-Proceeding), Sep. 2007.
3. Doiphode G. S., Dhyani D. and Patodi S. C.: "Integrated Force Method: A Matrix Method for Space Frame Structures", National Conference on Currents Trends in Technology. Nirma Institute of Technology, Ahmedabad, pp. 472-475, Dec. 2007.
4. Doiphode G. S., Salat K. N. and Patodi S. C.: "Secondary Effects in Framed Structures Via Integrated Force Method", Journal of Tech. & Science, Vidyanagar, Vol. 20, pp. 6-11, Dec. 2007.
5. Doiphode G. S., Kulkarni S. M. and Patodi S. C.: "Formulating a Plane Stress Rectangular Element based on Integrated Force Method", National Conference on Currents Trends in Technology. Nirma Institute of Technology, Ahmedabad, pp. 410-413, Nov. 2008.
6. Doiphode G. S., Kulkarni, S. M. and Patodi S. C.: "Improving Plate Bending Solutions using Integrated Force Method", 6th Structural Engineering Convention, Chennai, pp. 227–235, Dec. 2008.
7. Doiphode G. S., Kulkarni S. M. and Patodi S. C.: "Dual Integrated Force Method: A Displacement Based Matrix Approach for Framed Structures", National Conference on Emerging Vistas of Technology in 21st Century, Parul Institute of Engineering and Technology, Limda, Vadodara, pp. 6-10, Sep. 2009.

8. Kulkarni S. M., Patodi S. C. and Doiphode G. S.: "Development of Processors for the Analysis of 1D and 2D Structures using Dual Integrated Force Method", National Conference on Emerging Vistas of Technology in 21st Century, Parul Institute of Engineering and Technology, Limda, Vadodara, pp 46-50, Sep. 2009.
9. Doiphode G. S., Kulkarni S. M. and Patodi S. C.: "Formulation of 2D Plane Stress Curved Element based on Integrated Force Method", 4th National Conference on Currents Trends in Technology, NIT, Ahmedabad, pp. 25-28, Nov. 2009.
10. Doiphode G. S., Kulkarni S. M. and Patodi S. C.: "Development of Integrated Force Method based Formulation for Plane Stress Triangular Element", International Conference on Advances in Concrete, Structural and Geotechnical Engineering, Pilani, pp. 192, Dec. 2009.
11. Doiphode G. S., Prajapati J. K. and Patodi S. C.: "Automated Development of Compatibility Conditions for Framed Structures using Matlab", National Conference on Current Trends of Research and Development in Civil and Environmental Engineering, Sardar Patel Institute of Technology, Vasad, pp. 22, Jan. 2010.
12. Doiphode G. S., Kulkarni, S. M. and Patodi, S. C.: "Airy Stress Function based Integrated Force Method for 2D Plane Stress/Strain Problems", National Conference on Current Trends of Research and Development in Civil and Environmental Engineering, Sardar Patel Institute of Technology, Vasad, pp. 22, Jan. 2010.
13. Doiphode G. S., Patel V. A. and Patodi S. C.: "Calculating Natural Frequencies of a Plane Stress Problem using Integrated and Dual Integrated Force Methods", National Conference on Current Trends of Research and Development in Civil and Environmental Engineering, Sardar Patel Institute of Technology, Vasad, pp. 23, Jan. 2010.
14. Doiphode G. S., Patel V. A. and Patodi S. C.: "Frequency Analysis of Framed Structures using Integrated Force Method and its Modified Form", International Conference on Innovative World of Structural

Engineering, Govt. College of Engineering, Aurangabad, pp. 251-259, Sep. 2010.

15. Doiphode G. S., Kulkarni S. M. and Patodi S. C.: "Integrated Force Method: A Direct Moment Determination Approach for Orthotropic Plates", 2nd National Conference on Emerging Vistas of Technology in 21st Century, Parul Institute of Engineering and Technology, Limda, Vadodara, pp. 1-6, Dec. 2010.
16. Doiphode G. S., Patel V. A. and Patodi S. C.: " IFM and its Dual Form for Eigen Value Analysis of Plate Bending Problems", Nirma University Journal of Engineering and Technology, Vol. 1, No.2, pp. 3- 8, Dec. 2010.
17. Doiphode G. S. and Patodi S. C.: "Free Vibration Analysis of Orthotropic Plates using Integrated Force Method", Journal of A. D. Patel Institute of Technology, Vol. 8, No.1, pp 83-88, Sep. 2011.
18. Doiphode G. S. and Patodi S. C.: "Integrated Force Method for Fiber Reinforced Composite Plate Bending Problems", International Journal of Advanced Engineering Technology, Vol. II, pp 285-295, Oct. 2011.
19. Doiphode G. S. and Patodi S. C.: "A Hybrid Method for Annular Plate Bending Problems", International Journal of Engineering Research and Applications, Vol. 1, No. 3, pp 730-740, Oct. 2011.
20. Doiphode G. S. and Patodi S. C.: "Analysis of Plates under Hydrostatic and Patch Loading using Integrated Force Method", 2nd International Conference on Currents Trends in Technology, Nirma Institute of Technology, Ahmedabad, pp. 1-5, Dec. 2011.
21. Doiphode G. S. and Patodi S. C.: "Development of Integrated Force Method for Axisymmetric Plates with Circular Geometry", International Association of Bridge and Structural Engineering Journal, Vol. 42, No. 1, pp 9-19, March 2012.
22. Doiphode G. S. and Patodi S. C.: "Integrated Force Method for Buckling Analysis of Skeletal Structures", The Indian Journal of Technical Education, pp 143-150, April 2012