Finite Element Analysis of Slabs using MATLAB/Visual-FEA and Experimental Verification using UTM

by

ANKUR NAREDA 051615

NISHANT KUMAR 051626

Guided by

Mr. ANIL DHIMAN

DEPARTMENT OF CIVIL ENGINEERING

JAYPEE UNIVERSITY OF INFORMATION TECHNOLOGY

WAKNAGHAT, SOLAN (H.P.)

May 2009

CERTIFICATE

This is to certify that the work entitled, "**Finite Element Analysis of Slabs using MATLAB/Visual-FEA and Experimental Verification using UTM**" submitted by Ankur Nareda (051615) and Nishant Kumar (051626) in partial fulfillment for the award of degree of Bachelor of Technology in Civil Engineering of Jaypee University of Information Technology has been carried out under my supervision. This work has not been submitted partially or wholly to any other University or Institute for the award of this or any other degree or diploma.

Anil Dhiman
28.5.2009

Mr. Anil Dhiman

Sr. Lecturer, Dept. of Civil Engg.

JUIT Waknaghat

Acknowledgement

We wish to express our deep regards and gratitude to **Mr. Anil Dhiman**, Sr. Lecturer, Department of Civil Engineering, Jaypee University of Information Technology, Waknaghat, Solan, for his immense help and valuable guidance, constant encouragement and inspiration throughout the progress of this project.

We are also thankful to **Dr. R.M. Vasan**, Prof. and Dean, HOD, Civil Engineering Department and to all other faculty members of civil engineering department for their support and time to time guidance. Further, we thank Mr. Upendra Singh and Mr. Pannkaj from Civil Engineering labs for the help they extended while working in the laboratory.

Ankur Nareda
(051615)

Nishant Kumar
(051626)

ABSTRACT

In many areas of engineering, one requires to perform analysis of plates subjected to in-plane and lateral loads. Today, the finite element analysis is the most versatile method to obtain results quite close to actual. In the present work, two plates – one of steel and other of RCC, are analysed by the finite element method as plate-bending problems. The Mindlin's plate theory is used in the element formulation. As the finite element analysis involves large number of nodes and elements and large-size matrices, manual calculations are not possible. In the view of this, the MATLAB and Visual-FEA are used for analysis of the plates. Further, STAAD-pro is also used to verify the results obtained from MATLAB programs developed.

The steel plate has been analysed using MATLAB program developed and the results obtained were compared with those given by analysis by STAAD.pro. The error incurred between the two deflections has been found to negligible and of the order of 1.13%. The RCC plate has been cast and tested on UTM (Universal Testing Machine) to get load-deflection curve. Same plate was modeled in Visual-FEA application to plot the load-defection curve using Finite Element Method. The curve was found to be comparable to the one given by UTM. The deflection at the centre of the plate given by UTM at a load of 105 kN was 1.23 mm, while that obtained by Visual-FEA was 1.18 mm. The studies carried out herein conclude that the Finite Element Method gives quite accurate results to an acceptable degree and is a versatile enough to analyse continuous structures like plates and shells.

CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1
INTRODUCTION

1.1 General

The limitation of the human mind is such that it cannot grasp the behavior of its complex surroundings and creations in one operation.

It is well known from the elastic theory of plates that the classical solution involves tedious calculations, especially when the plates are arbitrary shaped and are anisotropic.

1.2 Plates and Shells[1]

Plates

In many area of structural design we require analysis of plates subjected to lateral loads. It is well known from the elastic theory of plates that the classical solution involves tedious calculation especially when the plates are arbitrary shaped and are anisotropic. According to the nature of stress states the plates are classified as follows:

1. *Thick plate*, in which triaxial state of stress is developed, is defined by a complete set of differential equations of 3-dimensional theory of elasticity. Plates for which the ratio of thickness to least dimension on plan exceeds 1/10 maybe taken as belonging to this class.

2. *Thin plates with small deflection* in which the membrane stresses are very small compared to flexural stresses under deformation due to transverse loading. This class may be taken to comprise plates for which the ratio of thickness to span does not exceed 1/10 and the maximum deflection w is less than h/10-h/5.

3. <u>*Thin plates with large deflection*</u> are characterized by the fact that the flexural stresses are accompanied by relatively large tensile or compressive stresses in the middle plane. these membranes stresses significantly affect the bending moment.

Shells

In many areas of structural design we require analysis of shell subjected to different types of loads. It is well known from the theory of shells that the classical solution involves tedious calculations and is extremely difficult especially for shells of arbitrary shapes. The finite element method is very much suited for the analysis of shells of general shapes because of its flexibility in accounting for arbitrary geometry, loadings and variations in material properties.

<u>*Thin shells*</u> – If the thickness of the shells is small compared to the radii of curvature of the mid surface, the shell is referred to as geometrically thin shell.

1.3 Finite Element Method

It was the work of Turner, Clough, Martin and Topp that led to the discovery of finite element method. The basic concept is that a body or a structure may be divided into smaller elements of finite dimensions called 'Finite Elements' .The original body or the structure is then considered as an assemblage of these elements connected at a finite number of joints called 'Nodes' or 'Nodal Points'.

The properties of the elements are formulated and combined to obtain the solution for the entire body or structure. Thus, instead of solving the problem for the entire structure or body in 1 operation, in this method attention is mainly devoted to the formulation of properties of the constituent element.

The procedure for combining the elements, solution of equations and the evaluation of elements strain and stresses are the same for any type of structural system or body. Hence, the FEM offers scope for developing general purpose programs with the properties of various types of elements forming an element library and other procedures of analysis forming the common core segments.

The finite-element method originated from the needs for solving complex elasticity, structural analysis problems in civil and aeronautical engineering. Its development can be traced back to the work by Alexander Hrennikoff (1941) and Richard Courant (1942). While the approaches used by these pioneers are dramatically different, they share one essential characteristic: mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.

Hrennikoff's work discretizes the domain by using a lattice analogy while Courant's approach divides the domain into finite triangular subregions for solution of second order elliptic partial differential equations (PDEs) that arise from the problem of torsion of a cylinder .Courant's contribution was evolutionary, drawing on a large body of earlier results for PDEs developed by Rayleigh, Ritz, and Galerkin.

`Development of the finite element method began in earnest in the middle to late 1950s for airframe and structural analysis and gathered momentum at the University of Stuttgart through the work of John Argyris and at Berkeley through the work of Ray W. Clough in the 1960s for use in civil engineering. By late 1950s, the key concepts of stiffness matrix and element assembly existed essentially in the form used today[5] and NASA issued request for proposals for the development of the finite element software NASTRAN in 1965. The method was provided with a rigorous mathematical foundation in 1973 with the publication of Strang and Fix's *An Analysis of The Finite Element Method*, and has since been generalized into a branch of applied mathematics for numerical modeling of physical systems in a wide variety of engineering disciplines, e.g., electromagnetism and fluid dynamics.

Variety of specializations under the umbrella of the mechanical engineering discipline (such as aeronautical, biomechanical, and automotive industries) commonly uses integrated FEM in design and development of their products. Several modern FEM packages include specific components such as thermal, electromagnetic, fluid, and structural working environments.

In a structural simulation, FEM helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs. FEM allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements. FEM software provides a wide range of simulation options for controlling the complexity of both modeling and analysis of a system. Similarly, the desired level of accuracy required and associated computational time requirements can be managed simultaneously to address most engineering applications. FEM allows entire designs to be constructed, refined, and optimized before the design is manufactured. This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications.

The introduction of FEM has substantially decreased the time to take products from concept to the production line. It is primarily through improved initial prototype designs using FEM that testing and development have been accelerated. In summary, benefits of FEM include increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity, and increased revenue.

1.4 Objective of the Project

Following are the objectives of the project:-

- Understand Finite Element Analysis and its application to structural analysis.
- Develop differential equations for plates as per Mindlin's theory and solve the same by FEM.
- Use CAD applications like STAAD-pro and MATLAB for complex FEM problems.
- Use Visual-FEA for analysis of slab.
- Verification of analytical results with the experimental investigation.

CHAPTER 2
THEORY OF PLATE BENDING

2.1 Literature Review

A plate can be considered a two dimensional extension of a beam in simple bending. Both plates and beams support loads transverse or perpendicular to their plane and through bending action. A plate is a flat (if it were curved, it would be a shell). A beam has a single moment of resistance, while a plate resists bending about two axis and has a twisting moment.

Basic behavior of geometry and deformation[2]

Consider a thin plate in the x-y plane of thickness t measured in the z direction as shown below:

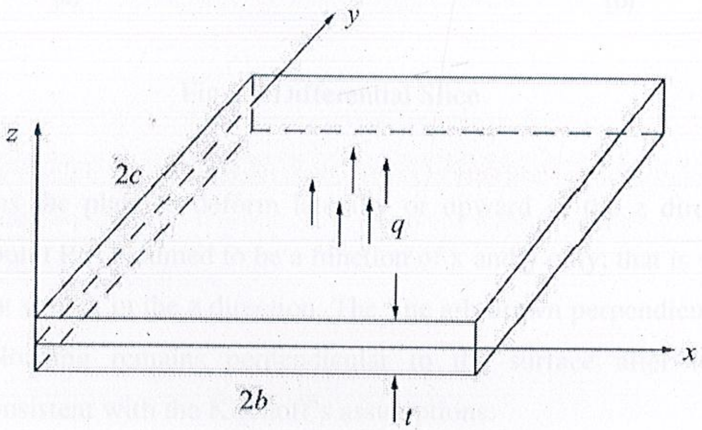

Fig. 2.1 Thin Plate

The plate surfaces are at $z = \pm t/2$, and its midsurface is at $z = 0$. The basic geometry of the plate is as follows:

1. The plate thickness is much smaller than its inplane dimensions b and c (i.e. t<<b or c). If t is more than about one-tenth of the span of the plate, then transverse shear deformation must be accounted for and the plate is said to be thick.
2. The deflection w is much less than the thickness t (i.e. w/t<<1).

2.2 Kirchoff assumptions[1]

Consider the differential slice cut from the plate by planes perpendicular to the x axis as shown in the figure below:

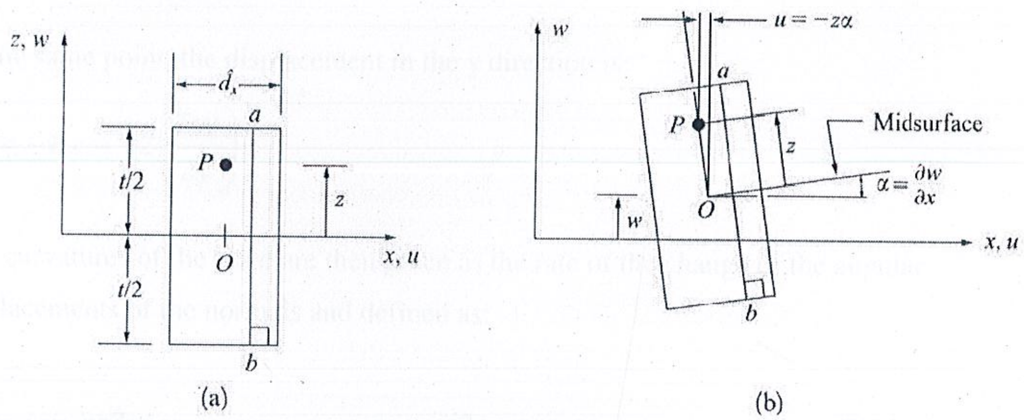

Fig 2.2 Differential Slice

Loading q causes the plate to deform laterally or upward in the z direction and, the deflection w of point P is assumed to be a function of x and y only; that is w = w(x,y) and the plate does not stretch in the z direction. The line a-b drawn perpendicular to the plate surface before loading remains perpendicular to the surface after loading. These conditions are consistent with the Kirchoff's assumptions:

1. Normal remains normal. This implies that transverse shear strains $\gamma_{yz} = 0$ and $\gamma_{xz} = 0$. However γ_{xy} = does not equals 0. Right angles in the plane of the plate may not remain right angles after loading. The plate may twist in the plane.
2. Thickness changes can be neglected and normals undergo no extension.
3. Normal stress σ_z has no effect on in-plane strains ε_x and ε_y in the stress-strain equations and is considered negligible.

4. Membrane or in-plane forces are neglected here, and the plane stress resistance can be superimposed later. Therefore, the in-plane deflections in the x and y direction at the midsurface, t=0, are assumed to be zero; u(x,y,0)=0 and v(x,y,0)=0.

Based on the Kirchoff assumptions, at any point P the displacement in the x direction due to small rotation α is:

$$u = -z\alpha = -z\left(\frac{\partial w}{\partial x}\right)$$

At the same point, the displacement in the y direction is:

$$v = -z\alpha = -z\left(\frac{\partial w}{\partial y}\right)$$

The curvatures of the plate are then given as the rate of the change of the angular displacements of the normals and defined as:

$$\kappa_x = -\frac{\partial^2 w}{\partial x^2} \qquad \kappa_y = -\frac{\partial^2 w}{\partial y^2} \qquad \kappa_{xy} = -\frac{2\partial^2 w}{\partial x \partial y}$$

Using the definitions for in-plane strains, along with the curvature relationships, the in-plane strain/displacement equations are:

$$\varepsilon_x = -z\frac{\partial^2 w}{\partial x^2} \qquad \varepsilon_y = -z\frac{\partial^2 w}{\partial y^2} \qquad \gamma_{xy} = -2z\frac{\partial^2 w}{\partial x \partial y}$$

The first of the above equations is used in beam theory. The remaining two equations are new to plate theory.

2.3 Stress-strain relationship[1]

Based on the third Kirchoff assumptions, the plane stress equations that relate in-plane stresses to in-plane strains for an isotropic material are:

$$\sigma_x = \frac{E}{1-\nu^2}\left(\varepsilon_x + \nu\varepsilon_y\right) \qquad \sigma_y = \frac{E}{1-\nu^2}\left(\varepsilon_y + \nu\varepsilon_x\right) \qquad \tau_{xy} = G\gamma_{xy}$$

The in-plane normal stresses and shear stress are shown acting on the edges of the plate shown in the figure below:

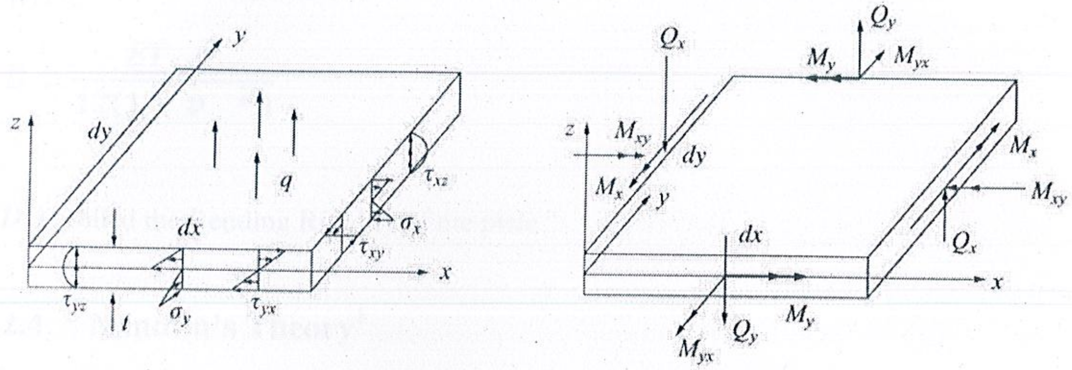

Fig 2.3 Stresses on edge of plate

Similar to stress variation in beam, the stresses vary linearly in the z direction from the midsurface of the plate. The transverse shear stresses τ_{yz} and τ_{xz} are also present, even though transverse shear deformation is neglected. These stresses vary quadratically through the plate thickness.

The bending moments acting along the edge of the edge of the plate can be related to the stresses by:

$$M_x = \int_{-t/2}^{t/2} \sigma_x z\,dz \qquad M_y = \int_{-t/2}^{t/2} \sigma_y z\,dz \qquad M_{xy} = \int_{-t/2}^{t/2} \tau_{xy} z\,dz$$

Substituting strain for stresses gives:

$$M_x = \int_{-t/2}^{t/2} z\left(\frac{E}{1-\nu^2}\left(\varepsilon_x + \nu\varepsilon_y\right)\right)dz \qquad M_y = \int_{-t/2}^{t/2} z\left(\frac{E}{1-\nu^2}\left(\varepsilon_y + \nu\varepsilon_x\right)\right)dz \qquad M_{xy} = \int_{-t/2}^{t/2} zG\gamma_{xy}dz$$

Using the strain/curvature relationships, the moment expression become:

$$M_x = D\left(\kappa_x + \nu\kappa_y\right) \qquad M_y = D\left(\kappa_y + \nu\kappa_x\right) \qquad M_{xy} = \frac{D(1-\nu)}{2}\kappa_{xy}$$

Where,

$$D = \frac{ET^3}{12(1-v^2)}$$

D is called the Bending Rigidity of the plate.

2.4 Mindlin's Theory[1]

Mindlin's approximation is that straight lines originally normal to the mid surface, before deformation, remain straight but not normal to the deformed surface, i.e., the average rotation of the section may be taken as the rotation in which normals remains perpendicular to the mid surface plus an additional rotation due to transverse shear. Thus the actual shear deformation is assumed to be equivalent to a straight line rotation representing a uniform shear strain through the thickness.

The three assumptions made in Mindlin's Theory of plates are:-

1. The deflections of the plate are small.
2. Normal to the plate mid surface before deformation remains straight but is not necessarily normal to it after deformation.
3. Stresses normal to the mid surface are negligible.

Element properties

1. Straight or curved one-dimensional elements with physical properties such as axial, bending, and torsional stiffnesses. This type of elements is suitable for modeling cables, braces, trusses, beams, stiffeners, grids and frames. Straight elements usually have two nodes, one at each end, while curved elements will

need at least three nodes including the end-nodes. The elements are positioned at the centroidal axis of the actual members.

2. Two-dimensional elements for membrane action (plane stress, plane strain) and/or bending action (plates and shells). They may have a variety of shapes such as flat or curved triangles and quadrilaterals. Nodes are usually placed at the element corners and, if needed for higher accuracy, additional nodes can be placed along the element edges or even inside the element. The elements are positioned at the mid-surface of the actual layer thickness.
3. Torus-shaped elements for axisymmetric problems such as thin, thick plates, shells, and solids. The cross-section of the elements are similar to the previously described types: one-dimensional for thin plates and shells, and two-dimensional for solids, and thick plates and shells.
4. Three-dimensional elements for modeling 3-D solids such as machine components, dams, embankments or soil masses. Common element shapes include tetrahedrals and hexahedrals. Nodes are placed at the vertexes and possibly in the element faces or within the element.

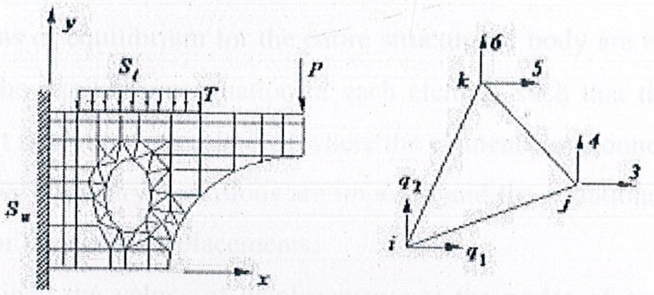

Fig. 2.4 A crude FE example mesh (with triangular and rectangular elements) for a cantilever beam with hole

Element interconnection and displacement

The elements are interconnected only at the exterior nodes, and altogether they should cover the entire domain as accurately as possible. Nodes will have nodal (vector) displacements or degrees of freedom which may include translations, rotations, and for special applications, higher order derivatives of displacements. When the nodes displace, they will *drag* the elements along in a certain manner dictated by the element formulation. In other words, displacements of any points in the element will be interpolated from the nodal displacements, and this is the main reason for the approximate nature of the solution.

Steps involved in the Finite Element Analysis[3]

1. In finite element analysis, simple functions known as 'shape functions' are chosen to approximate the variation of displacement within an element in terms of displacement at the nodes of the element.
2. It follows the concept used in the Rayleigh-Ritz procedure of functional approximation method but the difference is that the approximation to field variable is made at the element level.
3. The strains and stresses within an element will also be expressed in terms of the nodal displacement.
4. Then the *principle of virtual displacement or minimum potential energy* is used to derive the equation of equilibrium for the element and the nodal displacements will be the unknowns in the equations.
5. The equations of equilibrium for the entire structure or body are then obtained by combining the equilibrium equation of each element such that the continuity of displacement is ensured at each node where the elements are connected.
6. The necessary boundary conditions are imposed and the equations of equilibrium are solved for the nodal displacements.
7. Having obtained the values of displacements at the nodes of each element, the strains and stresses are evaluated using the element properties.

CHAPTER 3

ANALYSIS OF STEEL PLATE ELEMENT

3.1 The Problem Formulation[2]

A simply supported square plate made of steel has been considered for analyzing using finite element method. The plate is subjected to a concentrated load of 40 lb at the centre. The size of the plate is 10 inches by 10 inches and its thickness is 0.1 inches.

A program has been prepared in MATLAB for calculating the deflection of the plate at its centre parallel to the load.

3.2 Analysis using STAAD.pro

The plate has been modeled in STAAD.pro 2005 by giving the same inputs for geometry and material. The central deflection obtained due the same amount of load i.e. 40 lb, was 0.0441 inches. The comparison of this deflection is shown in the Table 3.1.

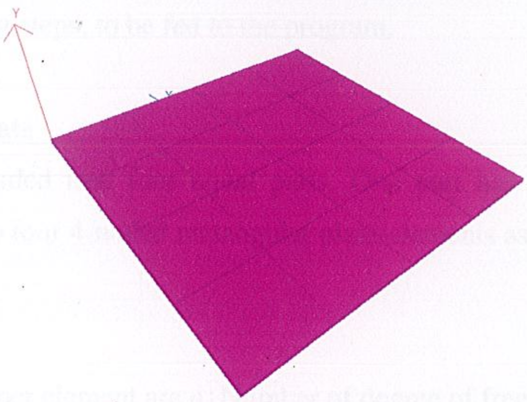

Fig.3.1 Isometric View of the Plate modeled in STAAD pro

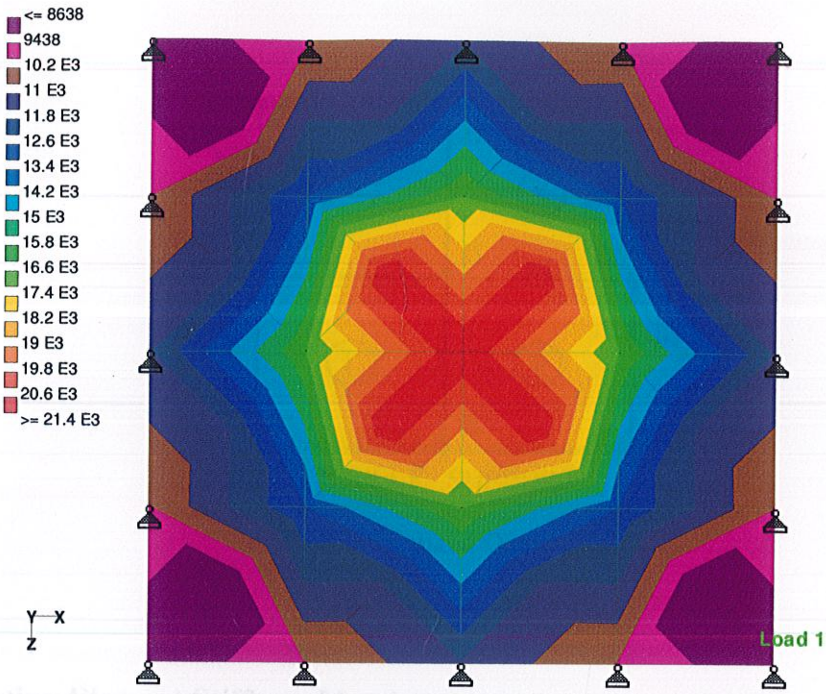

Fig.3.2 Stress contours obtained by STAAD.pro

3.3 Analysis using MATLAB program[2]

3.3.1 Preprocessing

The basic flow chart for the MATLAB program is shown in the Fig. 3.4. The input was prepared as the following steps, to be fed to the program.

Discretization of the plate

The plate has been divided into four equal parts. One part has been taken for FEM analysis and divided into four 4-noded rectangular plate-elements as shown in Fig. 3.3.

Degree of Freedom

Total numbers of nodes per element are 4. Number of degree of freedom per node is 3 i.e. in *x*, *y* and *z* direction. Total numbers of nodes in the system are 9. Therefore total number of degree of freedom in the system will be 27.

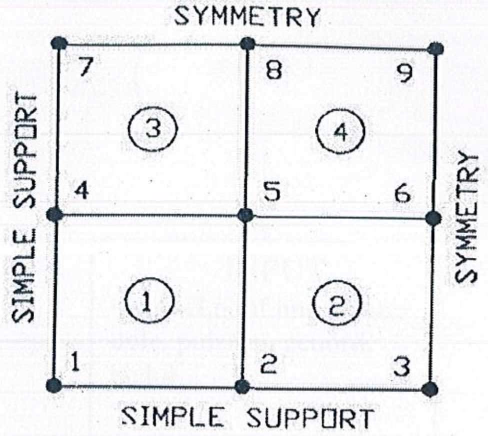

Fig.3.3 Discretized Quarter Plate

Generating Element Stiffness Matrices

Element stiffness matrix is generated by both using bending and shear. After computation both are summed up to get the element matrix.

Application of Boundary Conditions

After the generation of element stiffness matrix boundary conditions depending on the plates are applied.

3.3.2 Postprocessing

Nodal Displacement

Nodal displacement at each node is calculated by dividing nodal force vector by the element stiffness matrix which computed by combining bending and stiffness matrices

Strains and Stresses

The strain at any point may be calculated by multiplying strain –displacement matrix **B** with the nodal displacement vector obtained in the previous step. Ultimately, the stresses at various points are calculated by multiplying the property matrix **D** with the strain matrix just obtained.

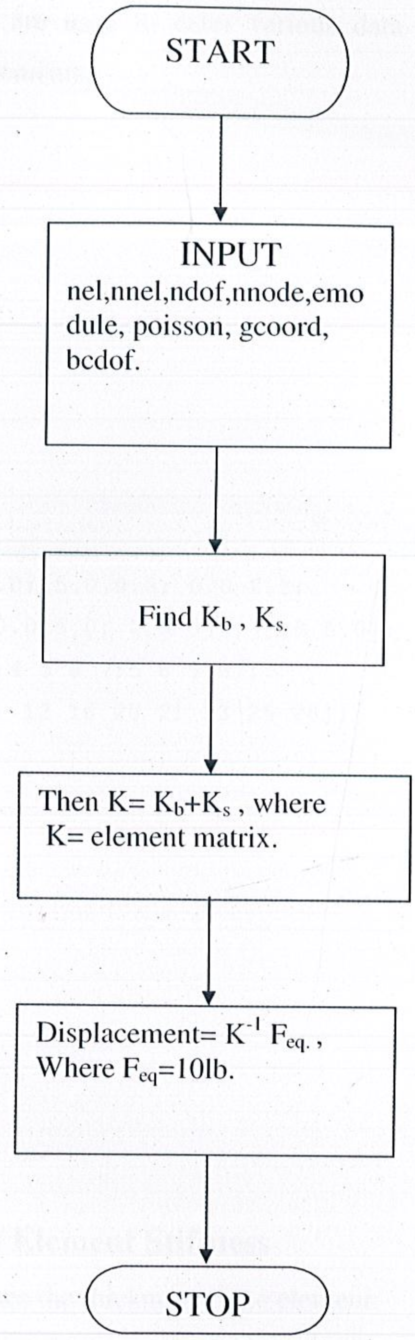

Fig. 3.4 Flow Chart for the MATLAB Program

3.4 Receiving Input Data

The following commands are used to enter various data which is required for the calculation of nodal displacements:

```
nel=4;
nnel=4;
ndof=3;
nnode=9;
sdof=nnode*ndof;
edof=ndof*nnel;
emodule=30e6;
poisson=0.3;
t=0.1;
nglxb=2; nglyb=2;
nglxs=1; nglys=1;
gcoord=[0.0 0.0; 2.5 0.0; 5.0 0.0; 0.0 2.5;
    2.5 2.5; 5.0 2.5; 0.0 5.0; 2.5 5.0; 5.0 5.0];
nodes=[1 2 5 4;2 3 6 5;4 5 8 7;5 6 9 8];
bcdof=[1 2 3 4 6 7 9 11 12 16 20 21 23 25 26];
bcval=zeros(1,15);
ff=zeros(sdof,1);
kk=zeros(sdof,sdof);
disp=zeros(sdof,1);
index=zeros(edof,1);
kinmtpb=zeros(3,edof);
matmtpb=zeros(3,3);
kinmtps=zeros(2,edof);
matmtps=zeros(2,2);
ff(27)=10;
```

3.5 Determination of Element Stiffness

The following code determines the thickness of the element:

```
for iel=1:nel
    for i=1:nnel
        nd(i)=nodes(iel,i);
        xcoord(i)=gcoord(nd(i),1);
        ycoord(i)=gcoord(nd(i),2);
```

```
    end
    k=zeros(edof,edof);
    kb=zeros(edof,edof);
    ks=zeros(edof,edof);
for intx=1:nglxb
        x=pointb(intx,1);
        wtx=weightb(intx,1);
        for inty=1:nglyb
            y=pointb(inty,2);
            wty=weightb(inty,2);
            [shape,dhdr,dhds]=feisoq4(x,y);
            jacob2=fejacob2(nnel,dhdr,dhds,xcoord,ycoord);
            detjacob=det(jacob2);
            invjacob=inv(jacob2);
            [dhdx,dhdy]=federiv2(nnel,dhdr,dhds,invjacob);
            kinmtpb=fekinepb(nnel,dhdx,dhdy);
            kb=kb+kinmtpb'*matmtpb*kinmtpb*wtx*wty*detjacob;
        end
    end
    for intx=1:nglxs
        x=points(intx,1)
        wtx=weights(intx,1)
        for inty=1:nglys
            y=points(inty,2)
            wty=weights(inty,2)
            [shape,dhdr,dhds]=feisoq4(x,y)
            jacob2=fejacob2(nnel,dhdr,dhds,xcoord,ycoord);
            detjacob=det(jacob2)
            invjacob=inv(jacob2);
            [dhdx,dhdy]=federiv2(nnel,dhdr,dhds,invjacob);
            kinmtps=fekineps(nnel,dhdx,dhdy,shape);
            ks=ks+kinmtps'*matmtps*kinmtps*wtx*wty*detjacob;
        end
    end
```

3.6 Assembling

Assembling of bending stiffness matrix and shear stiffness matrix is done as:

```
k=kb+ks;
    index=feeldof(nd,nnel,ndof);
    kk=feasmbl1(kk,k,index);
end
```

3.7 Nodal Displacements

Nodal displacements has been calculated by the following code:

```
[kk,ff]=feaplyc2(kk,ff,bcdof,bcval);

disp=kk\ff;

num=1:1:sdof;

displace=[num' disp]

contourf(displace);

colorbar;
```

3.8 Analysis of Results

After the calculation of nodal displacements, the result obtained by MATLAB is analyzed and compared with results obtained by STAAD.pro. An error of 1.133% has been found which is acceptable.

Table 3.1: Comparison of Central Deflections

Tool →	STAAD pro	MATLAB program
Central Deflection (inches)	0.0441	0.0436
Variation in Result w.r.t. STAAD pro	1.1337 %	

CHAPTER 4
EXPERIMENTAL ANALYSIS OF RCC SLAB

4.1 Introduction

A Concrete slab is a common structural element of modern buildings. Horizontal slabs of steel reinforced concrete, typically between 10 and 50 centimetres thick, are most often used to construct floors and ceilings, while thinner slabs are also used for exterior paving.

In many domestic and industrial buildings a thick concrete slab, supported on foundations or directly on the sub soil, is used to construct the ground floor of a building. In high rises buildings and skyscrapers, thinner, pre-cast concrete slabs are slung between the steel frames to form the floors and ceilings on each level.

Concrete slabs typically have a high thermal mass. In older buildings, concrete slabs cast directly on the ground can drain heat from a room. In modern construction techniques, concrete slabs are usually cast on top of thicker layers of insulation, for example expanded polystyrene, and may contain underfloor heating. Even so their thermal mass can lead to a delay warming the room when the heating is switched on. This can be an advantage in climates with large daily temperature swings, where the slab keeps the building cool by day and warm by night.

A concrete slab may be prefabricated or in situ. Prefabricated concrete slabs are built in a factory and transported to the site, ready to be lowered into place between steel or concrete beams. They may be pre-stressed (in the factory), post-stressed (on site), or unstressed. It is vital that the supporting structure is built to the correct dimensions, or the slabs may not fit.

In situ concrete slabs are built on the building site using formwork - a type of boxing into which the wet concrete is poured. If the slab is to be reinforced, the rebars are positioned within the formwork before the concrete is poured in. Plastic tipped metal, or plastic bar chairs are used to hold the rebar away from the bottom and sides of the formwork, so that when the concrete sets it completely envelops the reinforcement. For a ground slab, the formwork may consist only of sidewalls pushed into the ground. For a suspended slab, the formwork is shaped like a tray, often supported by a temporary scaffold until the concrete sets.

The formwork is commonly built from wooden planks and boards, plastic, or steel. On commercial building sites today, plastic and steel are more common as they save labour. On low-budget sites, for instance when laying a concrete garden path, wooden planks are very common. After the concrete has set the wood may be removed, or left there permanently.

In some cases formwork is not necessary - for instance, a ground slab surrounded by brick or block foundation walls, where the walls act as the sides of the tray and hardcore acts as the base.

4.2 RCC Slab Casting

A RCC slab was cast with the following properties:

- Slab Dimensions = 60cm × 60cm × 10cm
- Volume Of slab = $0.0377m^3$
- Volume of Cubes = $3 \times (0.15)^3 = 0.0101m^3$
- Total Volume = $0.0478m^3$
- Wt of slab = 0.0478 × 2400 = 114.78 kg.

- Using M20 (1:1.5:3) concrete and Fe 415 steel bars.
- Wt of Cement = 114.78/5.5 = 20.86kg ≈ 22kg.
- Wt of Sand = 1.5 ×114.78/5.5 = 31.30kg ≈ 33kg.
- Wt of Aggregates = 3 ×114.78/5.5 = 62.60kg ≈ 66kg.
- Water/ Cement Ratio = 0.5
- Use 10 bars of 10mm ϕ spaced 100mm C/C.

4.3 Tests on Slab

The UTM (Universal testing machine) was used to measure the deflection of the slab. The capacity of the UTM was 1000 kN (100 tones). A line load was applied at the centre of the slab. The load was increased gradually upto 105 kN and the load-deflection curve was plotted.

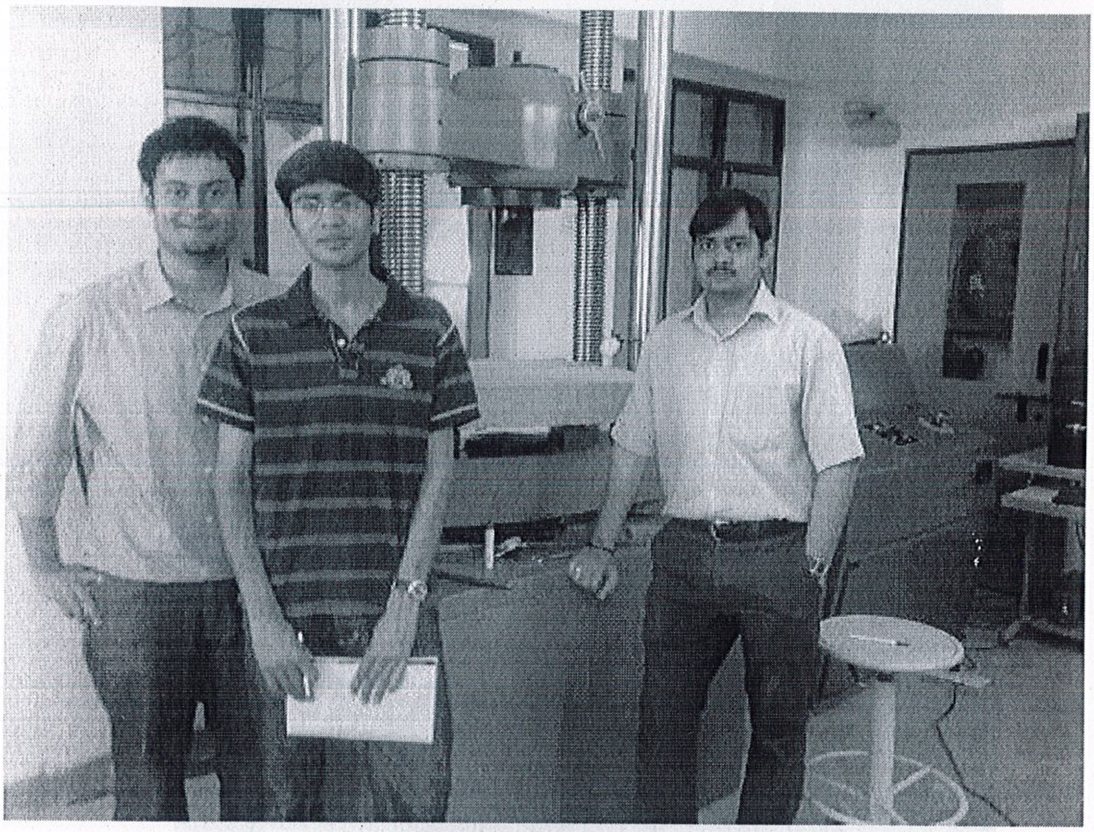

Fig.4.1 Project Team

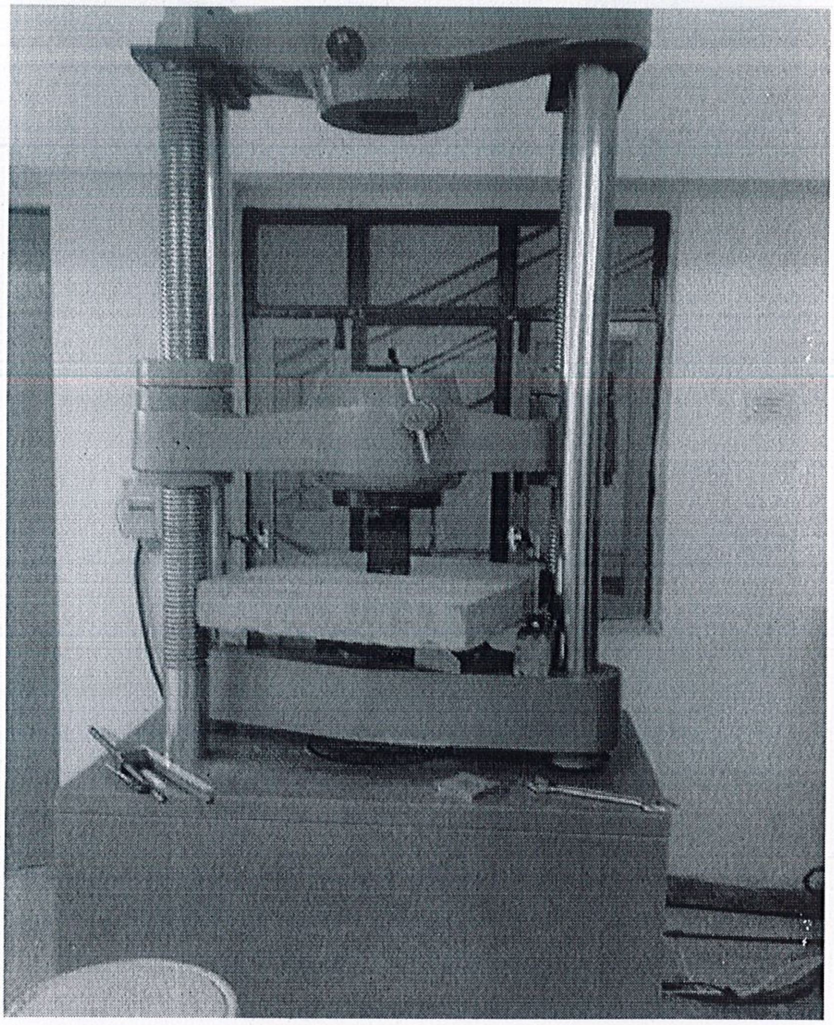

Fig.4.2 Slab being tested on UTM

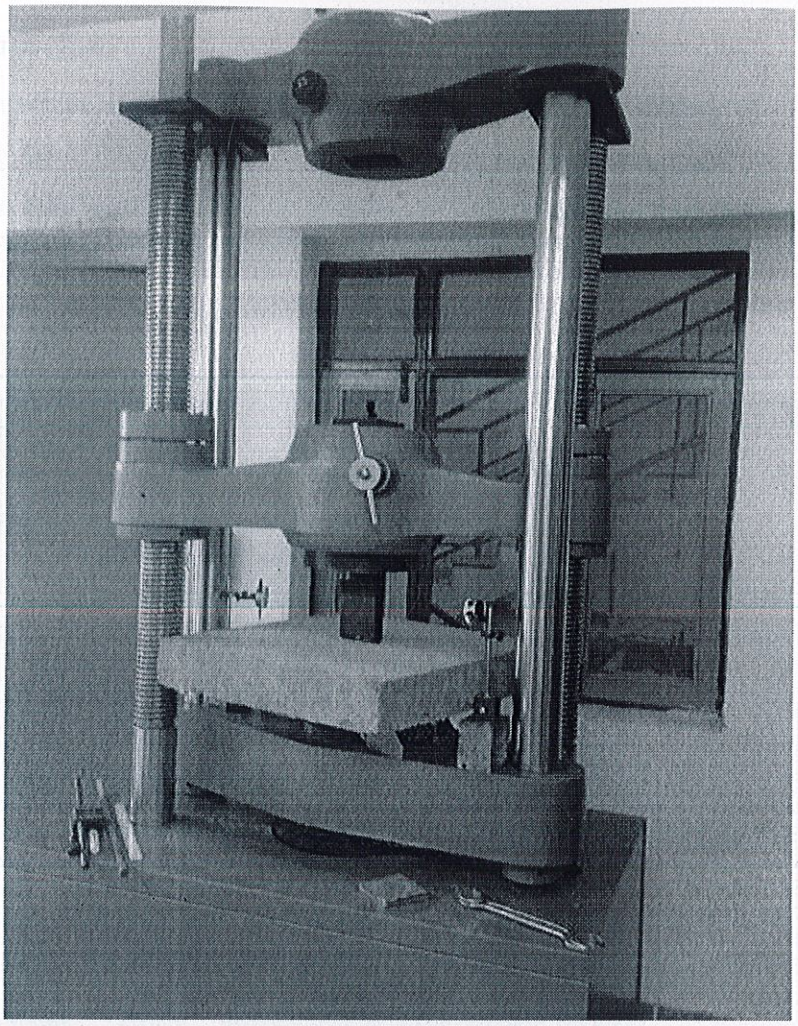

Fig.4.3 A line load is being applied using UTM at the centre of the slab

4.3 Tests on Slab

The UTM (Universal testing machine) was used to measure the deflection of the slab. The capacity of the UTM was 1000 kN (100 tones). A line load was applied at the centre of the slab. The load was increased gradually upto 105 kN and the load-deflection curve was plotted.

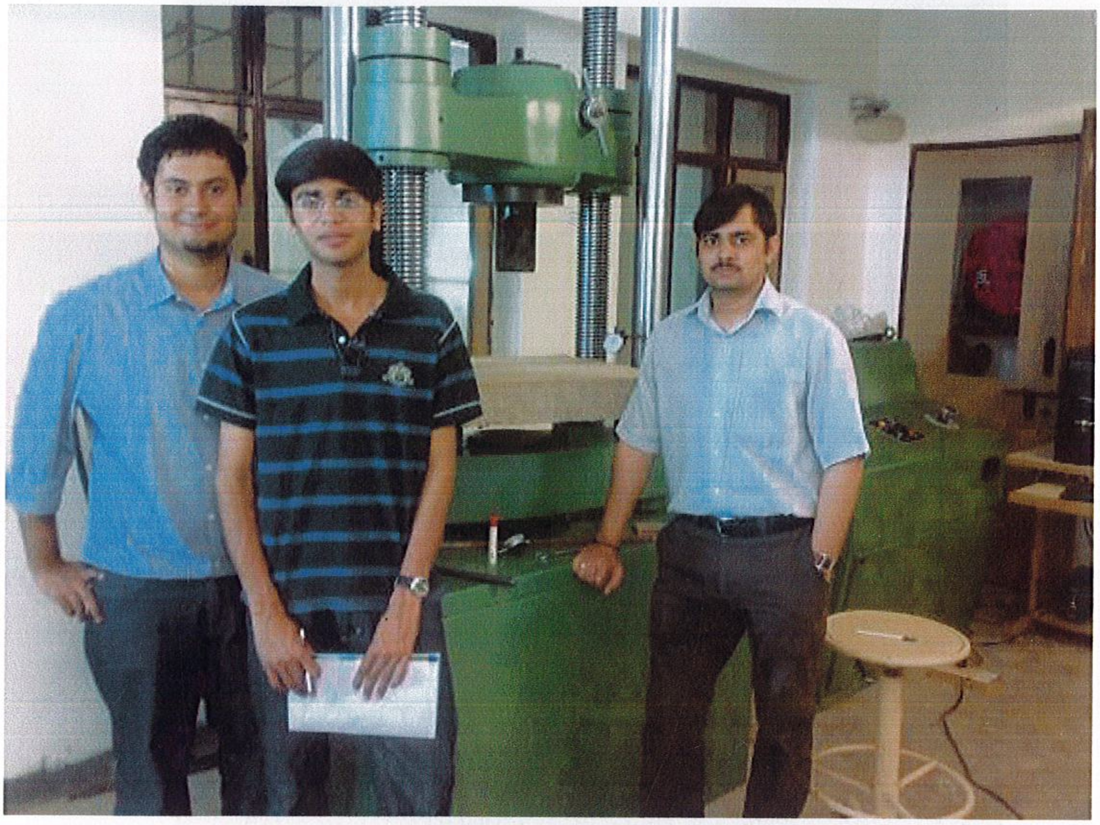

Fig.4.1 Project Team

Fig.4.2 Slab being tested on UTM

Fig.4.3 A line load is being applied using UTM at the centre of the slab

Table 4.1: Load-Deflection values Obtained by UTM

Load (kN)	Deflection (mm)
0	0.000
10	0.100
20	0.212
30	0.480
40	0.590
50	0.750
60	0.961
70	1.000
80	1.030
90	1.150
100	1.200
110	1.330
120	1.410

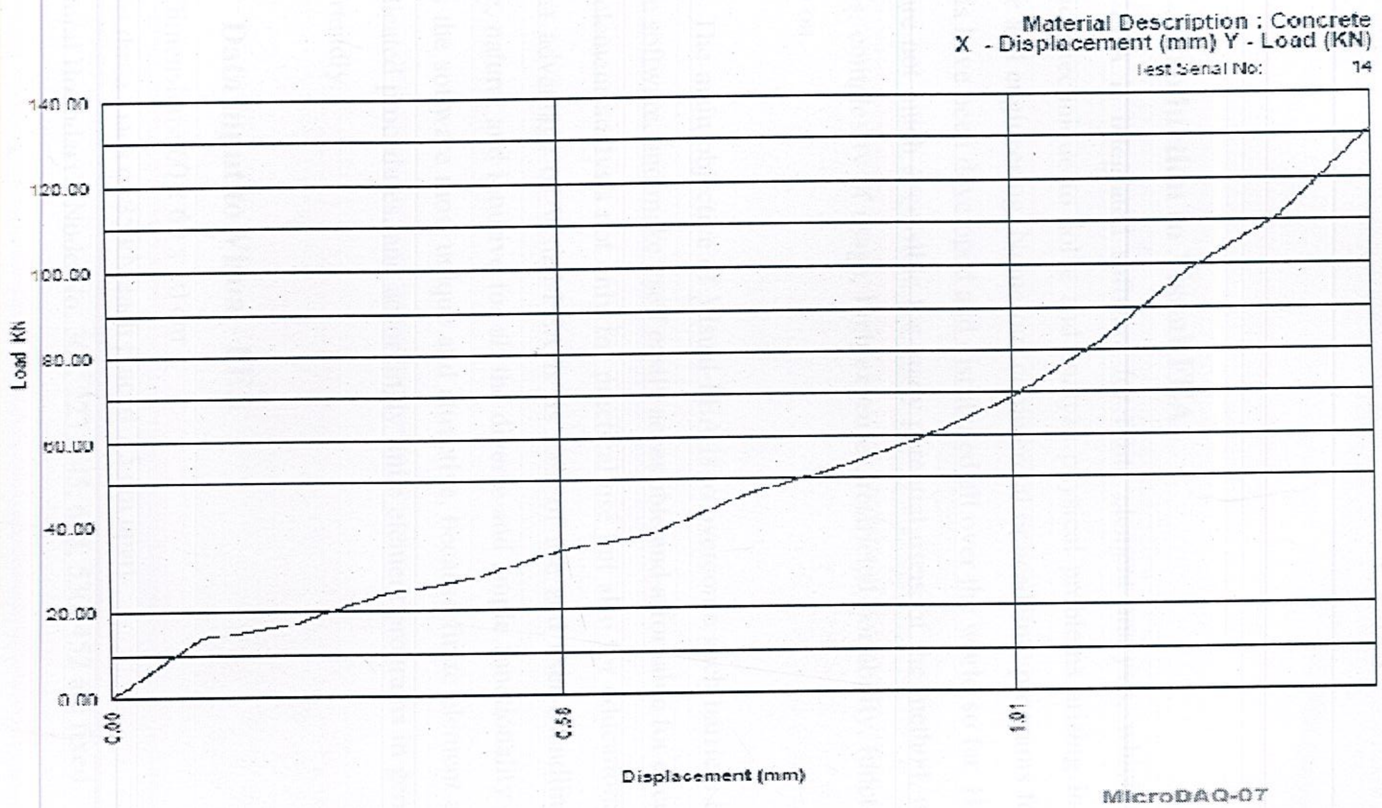

Fig.4.4 Load-deflection curve obtained by UTM

CHAPTER 5

ANALYSIS OF RCC SLAB WITH VISUAL-FEA

5.1 Introduction to Visual-FEA

Visual-FEA is integrated software for finite element analysis, which is an advanced numerical technique to solve and analyze physical problems arising in many fields of science and engineering. Numerous commercial or academic programs for finite element analysis have been developed and distributed all over the world so far. However, most of them are not much accessible for many potential users of the method, owing to various reasons: complexity of usage, high expenses, restricted portability, functional limitations and so on.

The main objective of Visual-FEA is to overcome such barriers between the user and the software, and make itself easily accessible and affordable for everyone who needs finite element analysis not only for practical use but also for educational purposes. The greatest advantage of Visual-FEA is its ease of use and user-friendliness. Its usage is simple, natural and intuitive for all the diverse and ample functionality it provides. This makes the software most unique and attractive, because finite element analysis involves complicated procedures, and accordingly finite element programs in general are far from user-friendly.

5.2 Data input to Visual- FEA

Slab Dimension= 60 x 60 x 10 cm

Loads= three loads of 35 kN each placed 7.5 cm apart.

Structural Boundary= Node No. 301, 223, 145, 608, 530, 452 are fixed.

5.3 Load-deflection curves

Given below is the load-deflection curve obtained by analyzing the RCC slab with Visual-FEA.

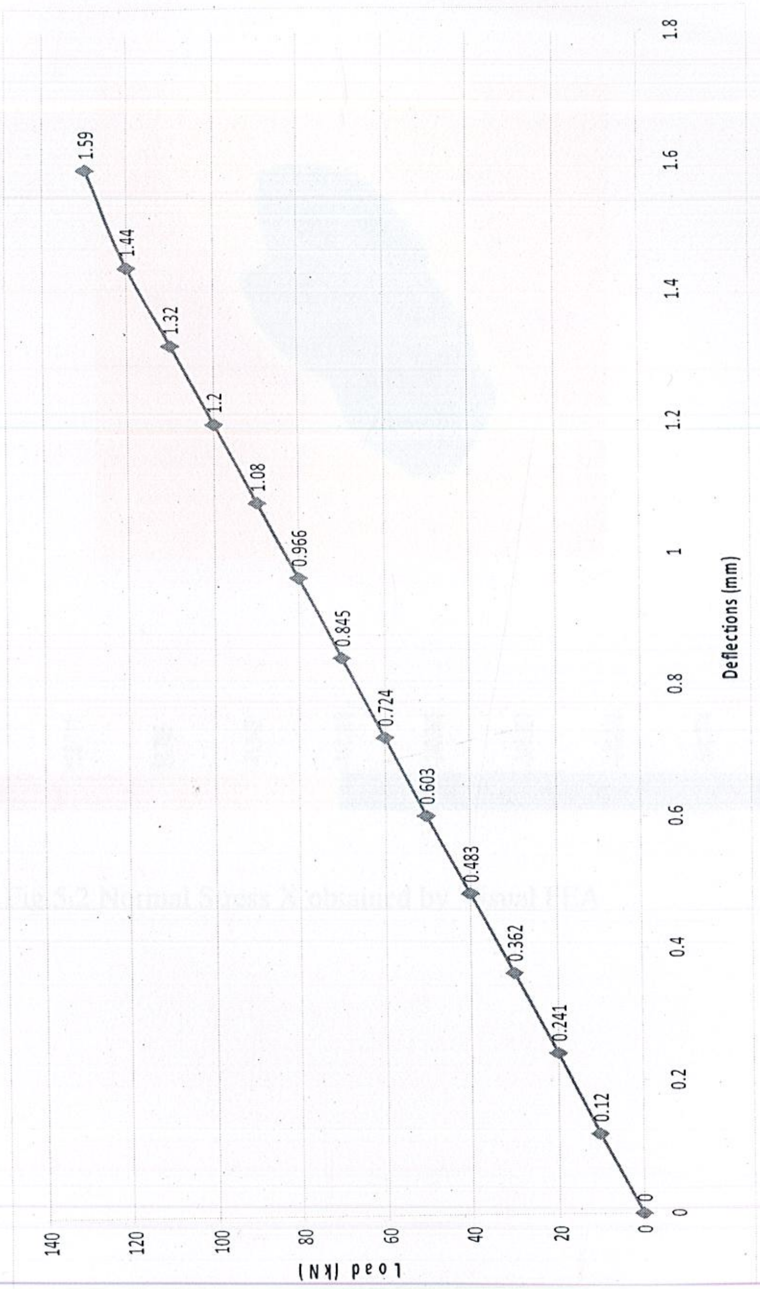

Fig.5.1 Load-deflection curve obtained by Visual-FEA

5.4 Stress and Deflection Contours

The stress contours obtained by Visual-FEA are given in this section, for normal stresses in X (Fig. 5.2) and Z-axis (Fig. 5.3) and displacement in X,Y and Z-axis (Fig. 5.4).

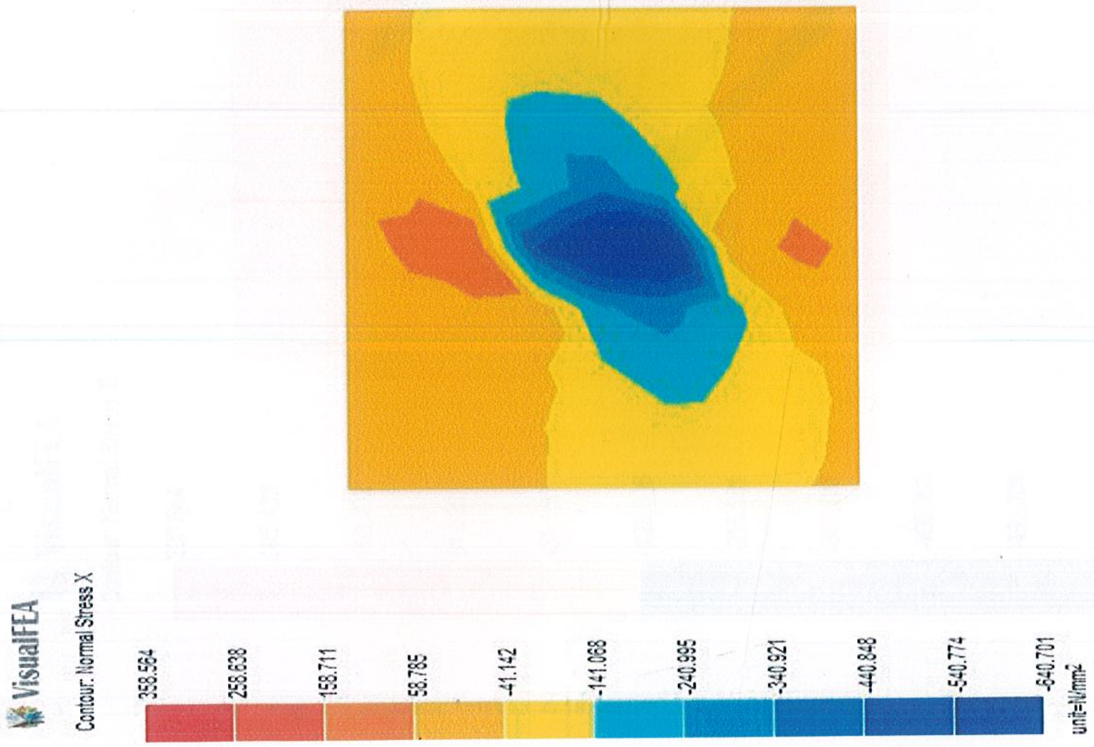

Fig.5.2 Normal Stress X obtained by Visual FEA

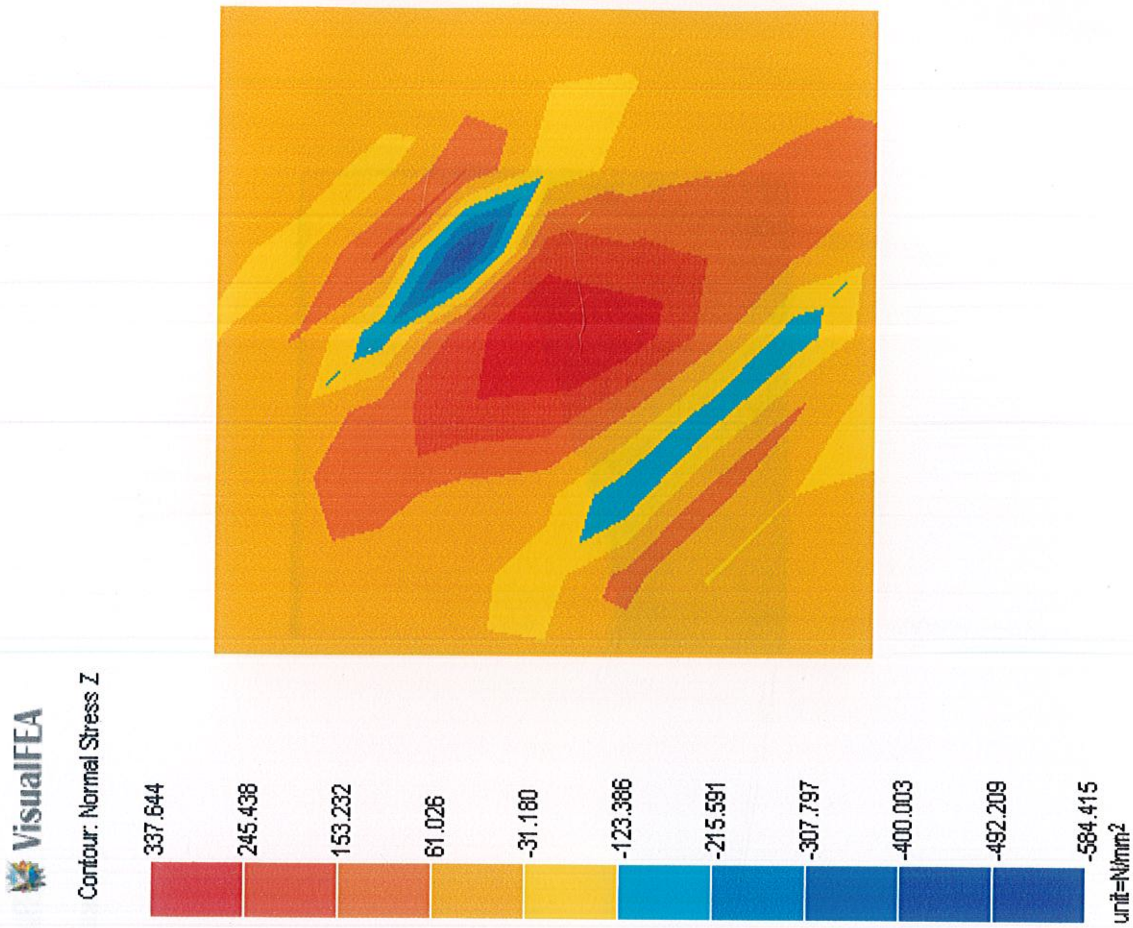

Fig.5.3 Normal Stress Z obtained by Visual FEA

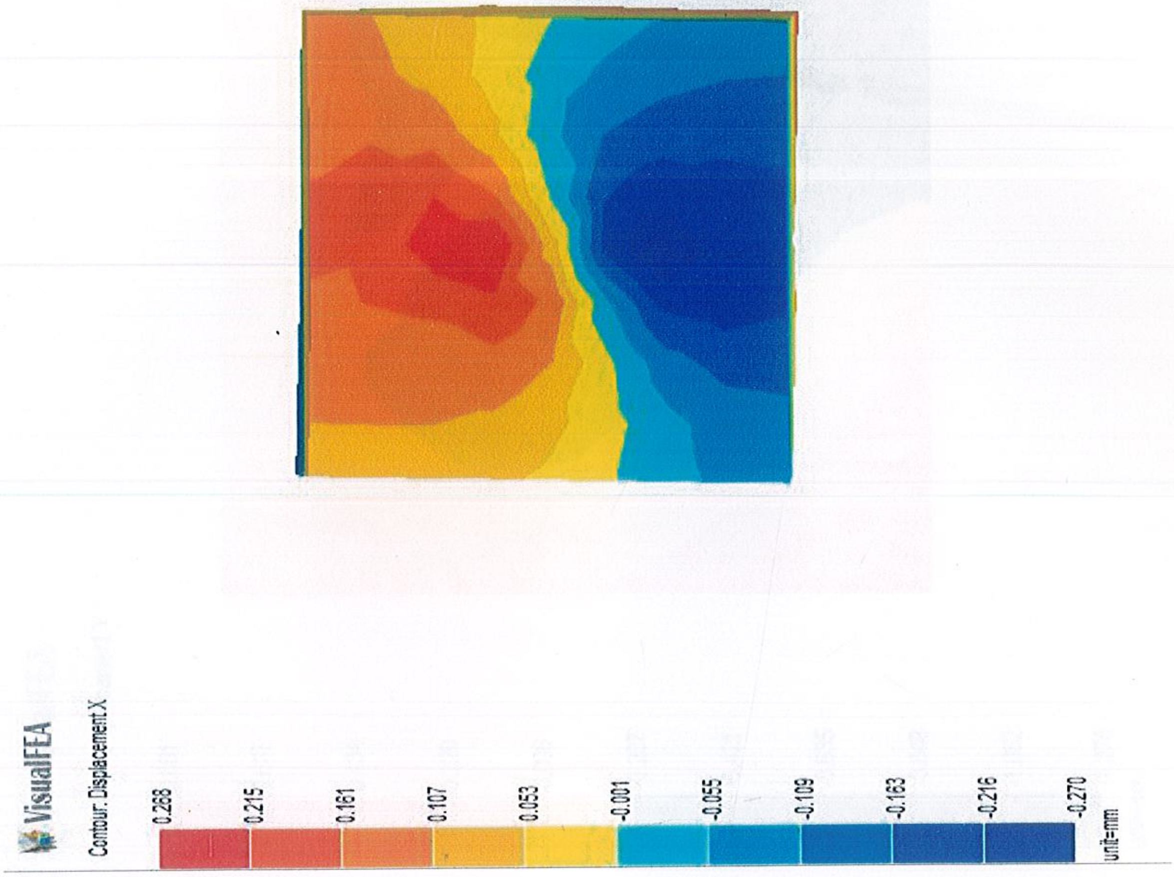

Fig.5.4 Displacement X obtained by Visual FEA

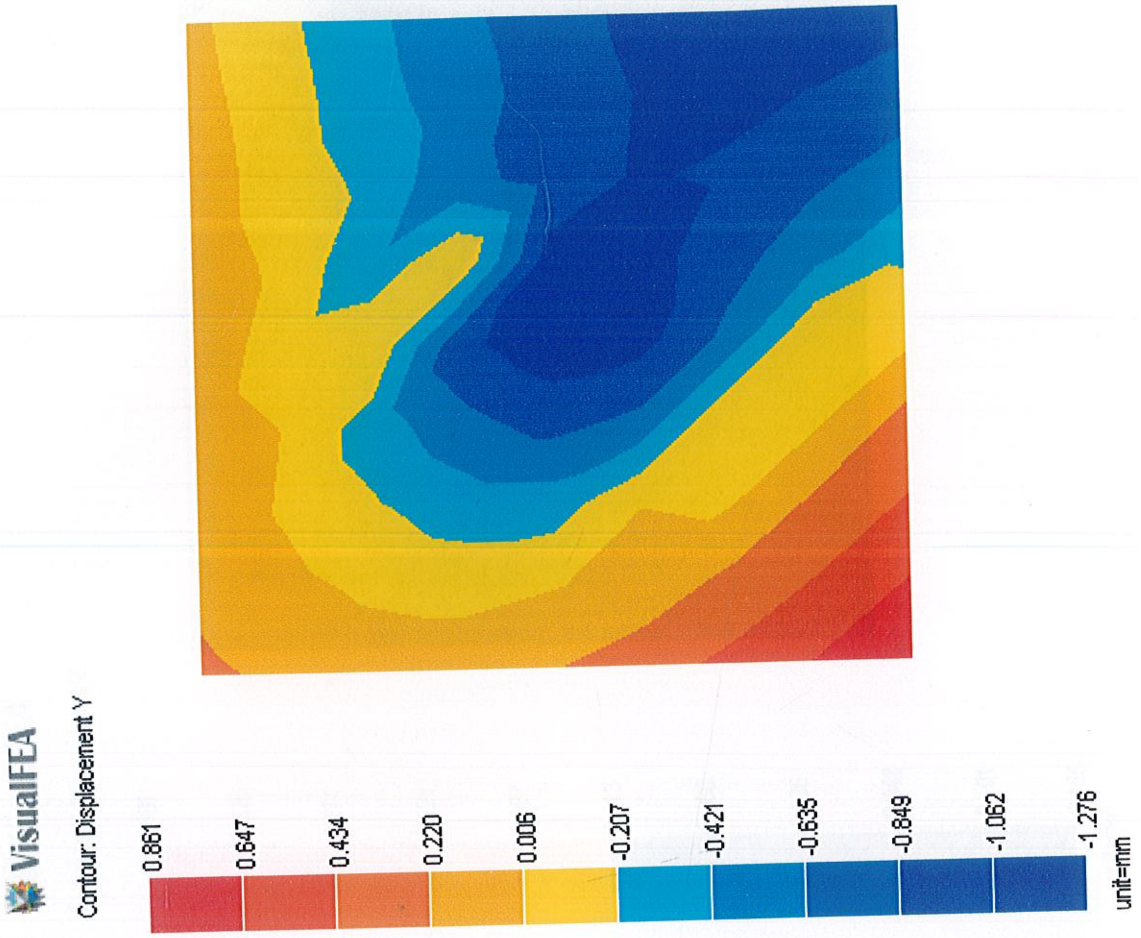

Fig.5.5 Displacement Y obtained by Visual FEA

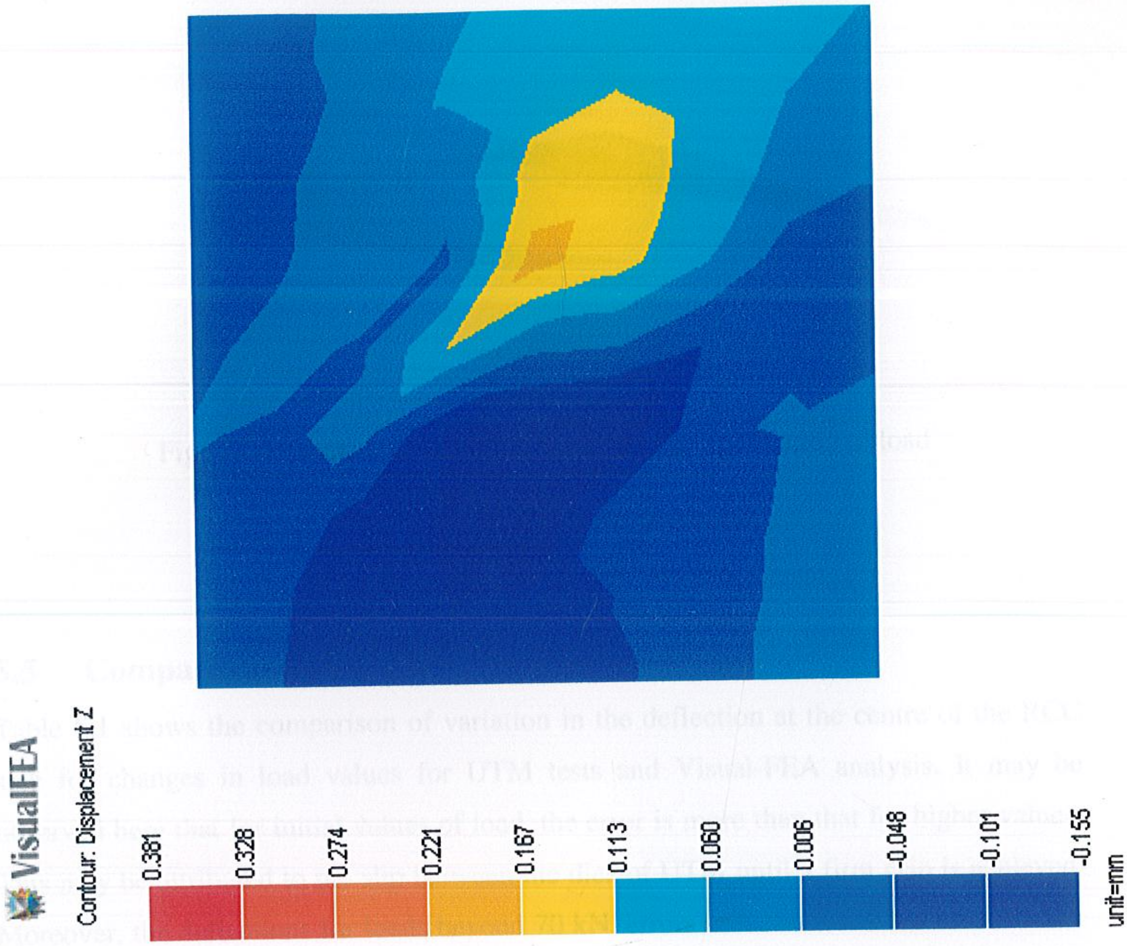

Fig.5.6 Displacement Z obtained by Visual FEA

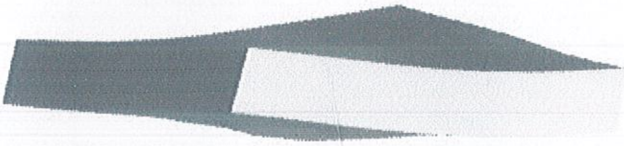

Fig.5.7 Deformed Shape of the slab after the application of load

5.5 Comparison of Results

Table 5.1 shows the comparison of variation in the deflection at the centre of the RCC slab for changes in load values for UTM tests and Visual-FEA analysis. It may be observed here that for initial values of load, the error is more than that for higher values. This may be attributed to the slip between the dies of UTM until a firm grip is achieved. Moreover, the deflections for loads beyond 70 kN, errors reduces. It is even zero for 100 kN load.

Table 5.2 shows the comparison of static deflections at the centre of the RCC slab for the two cases: test on UTM and analysis by Visual-FEA. The deflection obtained by UTM testing for a load of 105 kN applied at the centre was 1.23 mm and the deflection obtained by Visual-FEA was 1.18 mm; with an error of 4%.

The errors introduced herein may be attributed to the differences in the homogeneity of concrete in practice and that in the modeling. Moreover, location of the supports and loads may not be exactly simulated.

Table 5.1: Comparison of Load-Deflection values			
Load (kN)	Deflection at Centre (mm)		Difference (mm)
	UTM	**Visual-FEA**	
0	0.000	0.000	0
10	0.100	0.120	-0.02
20	0.212	0.241	-0.029
30	0.480	0.362	0.118
40	0.590	0.483	0.107
50	0.750	0.603	0.147
60	0.961	0.724	0.237
70	1.000	0.845	0.155
80	1.030	0.966	0.064
90	1.150	1.080	0.07
100	1.200	1.200	0
110	1.330	1.320	0.01
120	1.410	1.440	-0.03

Table 5.2: Comparison of Central Deflection		
Tool →	UTM	Visual FEA
Central Deflection	1.23 mm	1.18 mm
Variation in result w.r.t. Visual FEA	4 %	

CHAPTER 6
CONCLUSIONS

6.1 Conclusions

The analytical and experimental studies carried out in this report lead to the conclusions as given below:

- The program developed in MATLAB gives fairly accurate results. The deflection at the center of the steel plate is 0.0436 inches given by the program; which is verified with the result from STAAD-pro which is 0.0441 inches. The error incurred is merely 1.13%.
- The stress and deflection contours obtained from the Visual-FEA reveal the behaviour of the slab as expected practically.
- The deflection of the RCC slab at the centre is 1.23 mm obtained by Universal Testing Machine at a load of 105 kN, while the deflection at the centre of plate obtained using Visual-FEA is 1.18 mm, which are close to an acceptable degree.
- The load-deflection curve obtained by flexure tests by UTM is found to be follow the Experimental results obtained by Visual FEA are verified by the analytical results obtained by.
- For the initial values of load, the error is more than that for higher values. This may be attributed to the slip between the dies of UTM until a firm grip is achieved. Moreover, the deflections for loads beyond 70 kN, errors reduce. It is even zero for 100 kN load.

The accuracy of the results could be increased by increasing the number of elements and by choosing higher-order displacement model and elements. Lastly, it may be stated that the Finite Element Method is a very powerful tool to analyse discrete as well as continuous structures as plates.

APPENDIX A: MATLAB and STAAD.pro Source codes

```
nel=4;
nnel=4;
ndof=3;
nnode=9;
sdof=nnode*ndof;
edof=ndof*nnel;
emodule=30e6;
poisson=0.3;
t=0.1;
nglxb=2; nglyb=2;
nglxs=1; nglys=1;
gcoord=[0.0 0.0; 2.5 0.0; 5.0 0.0; 0.0 2.5;
    2.5 2.5; 5.0 2.5; 0.0 5.0; 2.5 5.0; 5.0 5.0];
nodes=[1 2 5 4;2 3 6 5;4 5 8 7;5 6 9 8];
bcdof=[1 2 3 4 6 7 9 11 12 16 20 21 23 25 26];
bcval=zeros(1,15);
ff=zeros(sdof,1);
kk=zeros(sdof,sdof);
disp=zeros(sdof,1);
index=zeros(edof,1);
kinmtpb=zeros(3,edof);
matmtpb=zeros(3,3);
kinmtps=zeros(2,edof);
matmtps=zeros(2,2);
ff(27)=10;
[pointb,weightb]=feglqd2(nglxb,nglyb);
matmtpb=fematiso(1,emodule,poisson)*(t*t*t)/12;
[points,weights]=feglqd2(nglxs,nglys);
shearm=0.5*emodule/(1.0+poisson);
shcof=5/6;
matmtps=shearm*shcof*t*[1 0;0 1];

for iel=1:nel
    for i=1:nnel
        nd(i)=nodes(iel,i);
```

```
    xcoord(i)=gcoord(nd(i),1);
    ycoord(i)=gcoord(nd(i),2);
end
k=zeros(edof,edof);
kb=zeros(edof,edof);
ks=zeros(edof,edof);
for intx=1:nglxb
    x=pointb(intx,1);
    wtx=weightb(intx,1);
    for inty=1:nglyb
        y=pointb(inty,2);
        wty=weightb(inty,2);
        [shape,dhdr,dhds]=feisoq4(x,y);
        jacob2=fejacob2(nnel,dhdr,dhds,xcoord,ycoord);
        detjacob=det(jacob2);
        invjacob=inv(jacob2);
        [dhdx,dhdy]=federiv2(nnel,dhdr,dhds,invjacob);
        kinmtpb=fekinepb(nnel,dhdx,dhdy);
        kb=kb+kinmtpb'*matmtpb*kinmtpb*wtx*wty*detjacob;
    end
end
for intx=1:nglxs
    x=points(intx,1)
    wtx=weights(intx,1)
    for inty=1:nglys
        y=points(inty,2)
        wty=weights(inty,2)
        [shape,dhdr,dhds]=feisoq4(x,y)
        jacob2=fejacob2(nnel,dhdr,dhds,xcoord,ycoord);
        detjacob=det(jacob2)
        invjacob=inv(jacob2);
        [dhdx,dhdy]=federiv2(nnel,dhdr,dhds,invjacob);
        kinmtps=fekineps(nnel,dhdx,dhdy,shape);
        ks=ks+kinmtps'*matmtps*kinmtps*wtx*wty*detjacob;
    end
end
k=kb+ks;
index=feeldof(nd,nnel,ndof);
```

```
   kk=feasmbl1(kk,k,index);
end
[kk,ff]=feaplyc2(kk,ff,bcdof,bcval);

disp=kk\ff;

num=1:1:sdof;

displace=[num' disp]

contourf(displace);

colorbar;
```

Source Code of Plate modeled in STAAD.pro 2005:

STAAD SPACE
START JOB INFORMATION
ENGINEER DATE 20-Nov-08
END JOB INFORMATION
INPUT WIDTH 79
UNIT INCHES POUND
JOINT COORDINATES
1 0 0 0; 2 10 0 0; 3 10 0 10; 4 0 0 10; 5 2.5 0 0; 6 2.5 0 2.5;
7 0 0 2.5; 8 5 0 0; 9 5 0 2.5; 10 7.5 0 0; 11 7.5 0 2.5; 12 10 0 2.5;
13 2.5 0 5; 14 0 0 5; 15 5 0 5; 16 7.5 0 5; 17 10 0 5; 18 2.5 0 7.5;
19 0 0 7.5; 20 5 0 7.5; 21 7.5 0 7.5; 22 10 0 7.5; 23 2.5 0 10;
24 5 0 10; 25 7.5 0 10;
ELEMENT INCIDENCES SHELL
2 1 5 6 7; 3 5 8 9 6; 4 8 10 11 9; 5 10 2 12 11; 6 7 6 13 14;
7 6 9 15 13; 8 9 11 16 15; 9 11 12 17 16; 10 14 13 18 19;
11 13 15 20 18; 12 15 16 21 20; 13 16 17 22 21; 14 19 18 23 4;

```
15 18 20 24 23; 16 20 21 25 24; 17 21 22 3 25;
ELEMENT PROPERTY
2 TO 17 THICKNESS 0.1
DEFINE MATERIAL START
ISOTROPIC STEEL
E 2.97327e+007
POISSON 0.3
DENSITY 0.283
ALPHA 1.2e-005
DAMP 0.03
END DEFINE MATERIAL
CONSTANTS
MATERIAL STEEL MEMB 2 TO 17
SUPPORTS
1 TO 5 7 8 10 12 14 17 19 22 TO 25 PINNED
LOAD 1 LOADTYPE Live TITLE LL
JOINT LOAD
15 FY -40
PERFORM ANALYSIS
LOAD LIST ALL
PRINT ANALYSIS RESULTS
PRINT JOINT DISPLACEMENTS LIST 1 TO 25
FINISH
```

APPENDIX B: Overview of MATLAB and STAAD-pro

Overview of MATLAB

MATLAB is interactive software which has been used recently in various areas of engineering and scientific applications. The power of MATLAB is represented by the length and simplicity of the code. MATLAB provides Graphical User Interface (GUI) as well 3-D graphical animation.

In general, MATLAB is a useful tool for vector and matrix manipulations. Since the majority of the engineering systems are represented by matrix and vector equations, we can relieve our workload to a significant extent using MATLAB. The Finite Element Method is a well defined candidate for which MATLAB can be very useful solution tool. Matrix and vector manipulations are essential parts in the method.

Overview of STAAD.pro

STAAD Pro is the World's leading Structural Analysis and Design Package for Structural Engineers. Incorporating design codes for 15 different countries it is also the most comprehensive and universal. The STAAD/Pro Suite of software incorporates many aspects aimed at making the Engineers' working life as easy as possible.

The new graphical interface uses the latest in object oriented programming techniques and allows STAAD users to quickly and easily manage models of all sizes. In addition to all the features available in the Space Frame section STAAD has a range of Automatic Mesh Generators for slabs, complex shapes etc.

The STAAD Analysis Engine has 2D and 3D capabilities for solving problems containing Beams, plate elements and 8 noded bricks. The general nature of the solution engine allows beam models using the stiffness matrix method to be combined with finite elements. A wide range of support conditions, load types and various other member/element specifications are available for combination with these features.

APPENDIX C: Illustrative Steps Followed in Visual-FEA

Step 1: Open Visual FEA. Select new option from File menu. A project setup dialog box will appear, select 3D solid. Select the units as kN/mm. Click O.K.

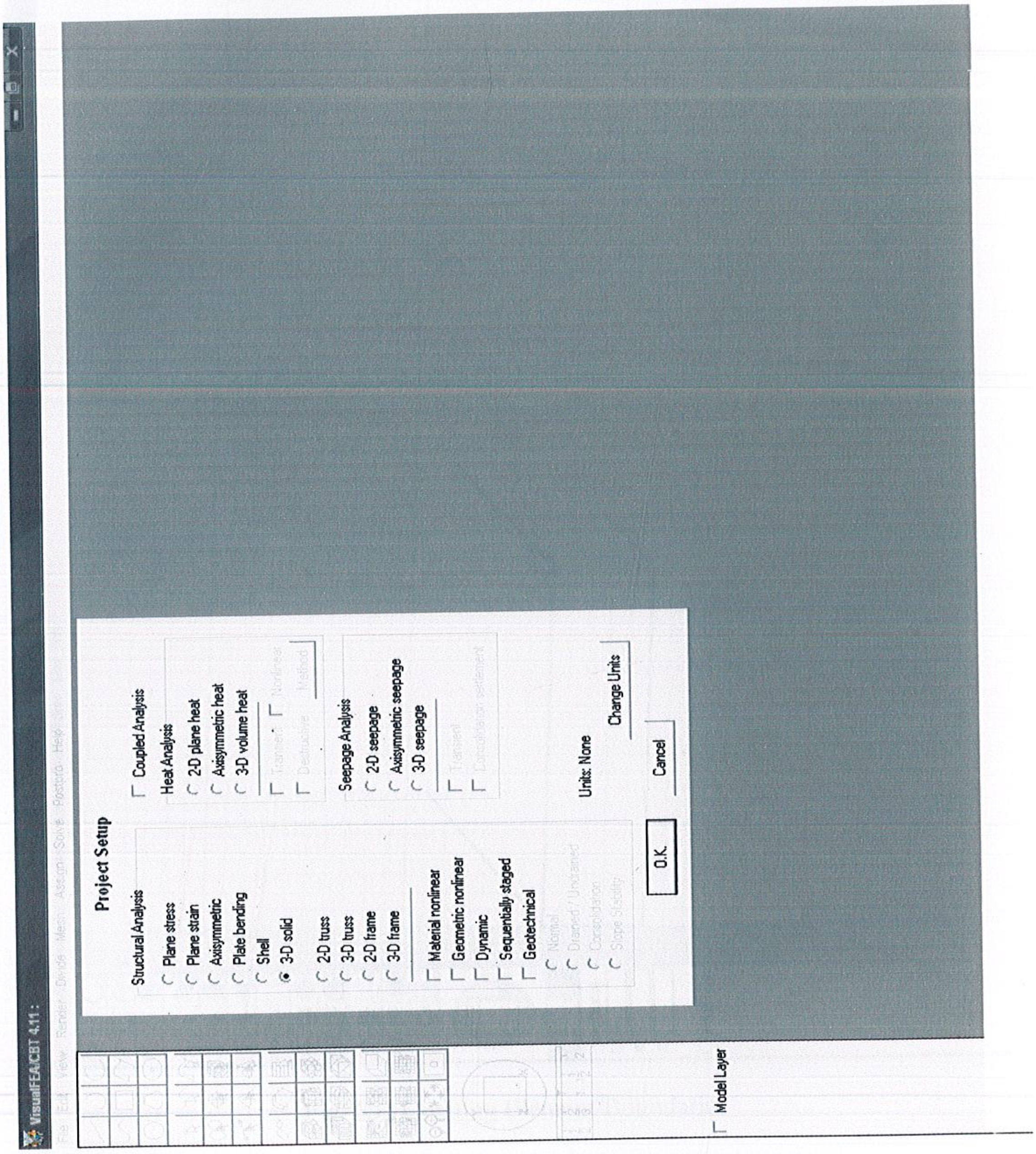

Fig. C-1 Creating a New File and Setting Units

Step 2: A grid view will appear select *line* option from the tool box at the left side of the window. Enter the coordinate values for the slab to define the boundaries.

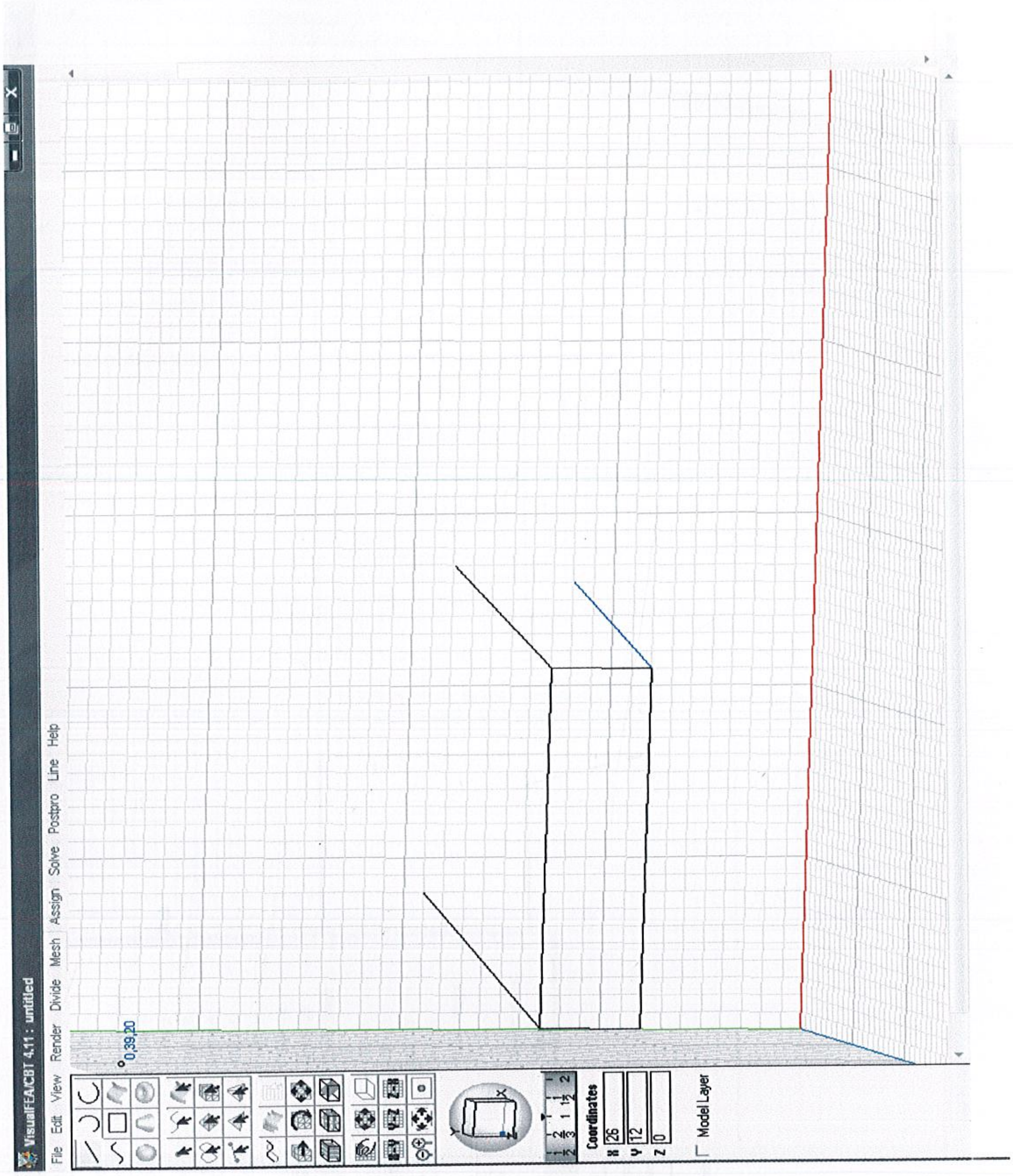

Fig. C-2 Creating Element Boundaries

Step 3: Select *Divide* option and divide all the line elements into desired numbers. Finer the division, finer will be the mesh and the accuracy of the result.

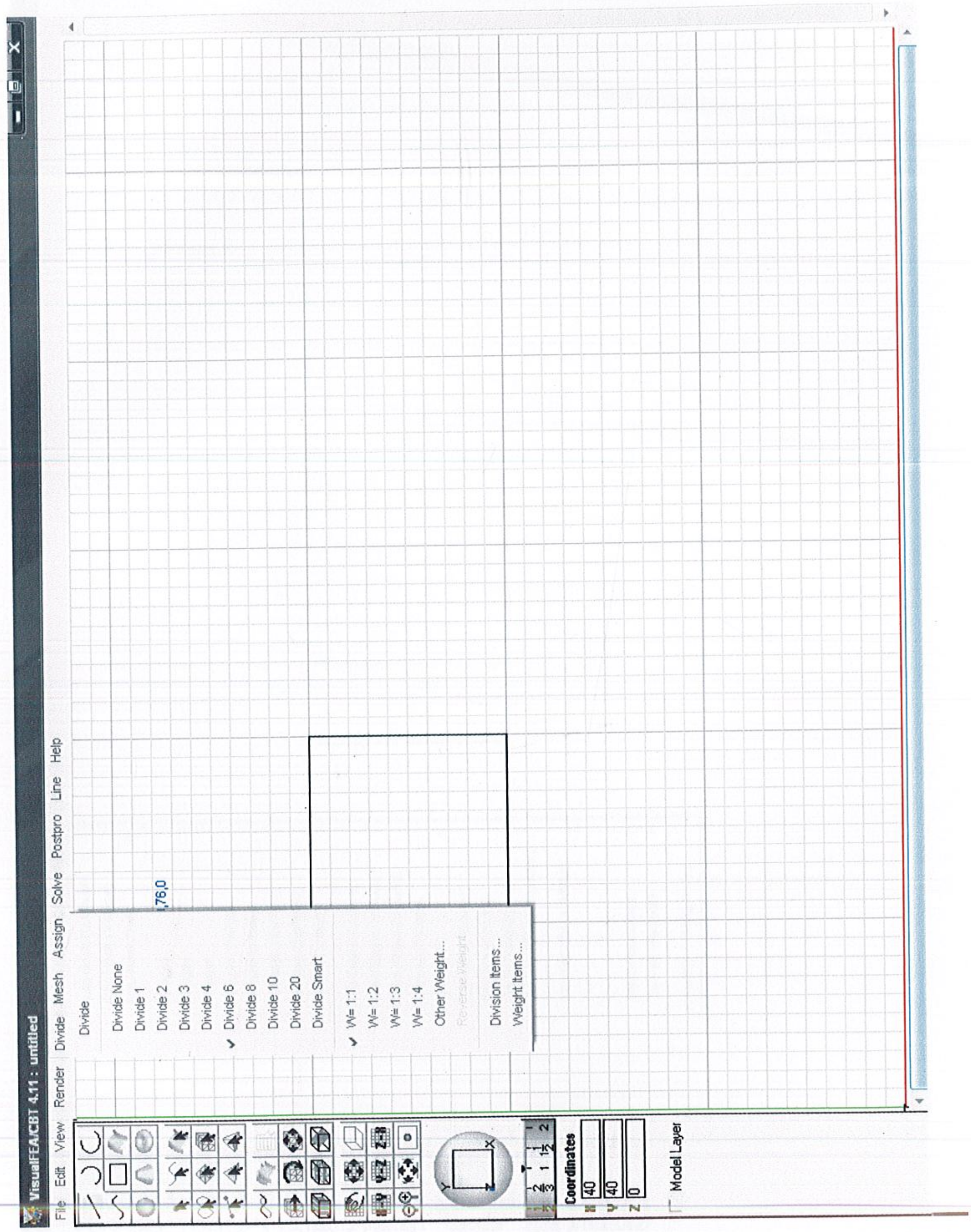

Fig. C-3 Dividing the Element for Mesh Generation

Step 4: Select the type of mesh from *Mesh* menu.

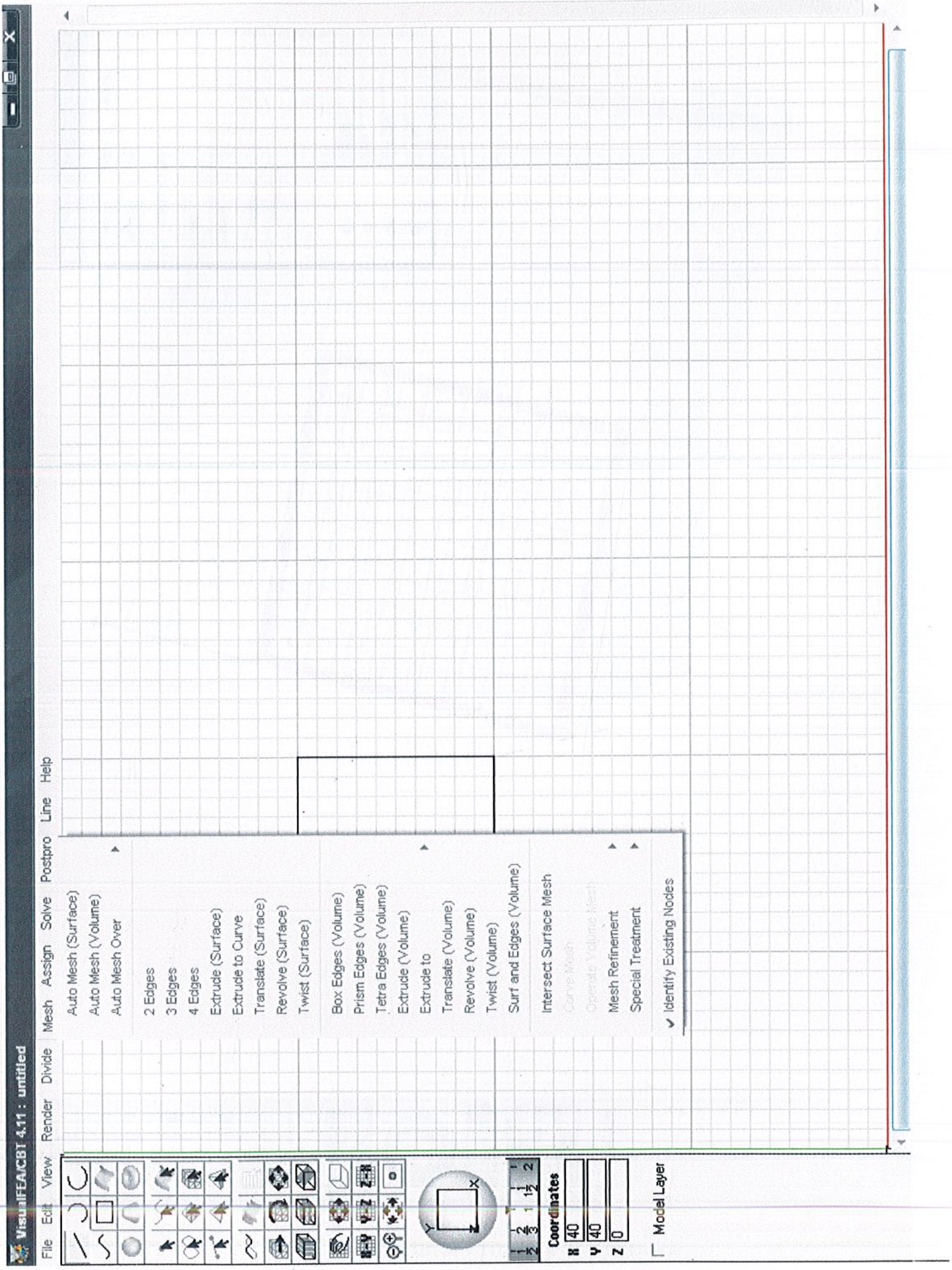

Fig. C-4 Meshing of the element

Step 5: Assign the structural properties of Concrete and Steel as shown in the figure below.

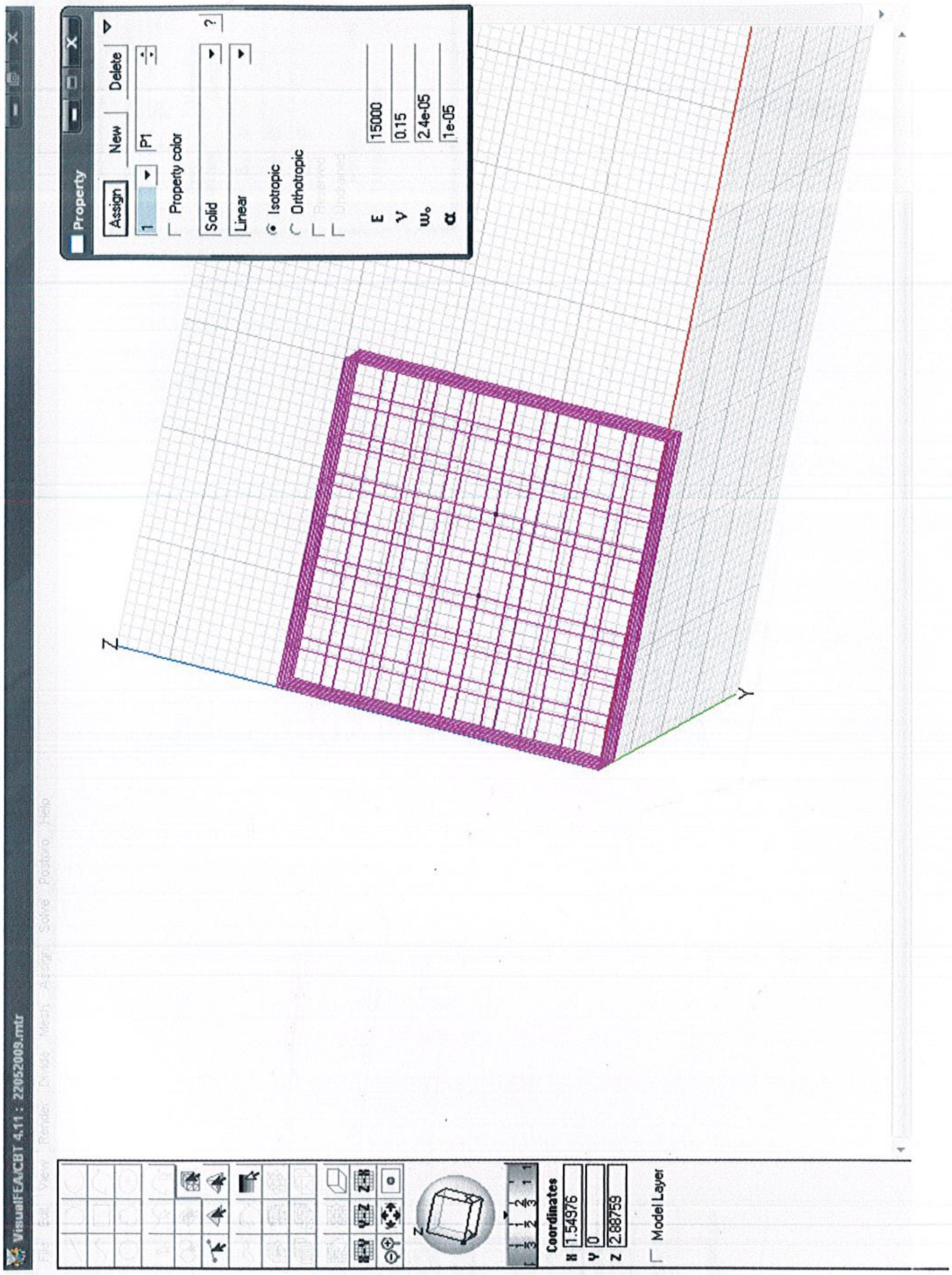

Fig. C-5 Structural Property assignment

Step 6: Apply the boundary conditions i.e. fixed or roller or hinged.

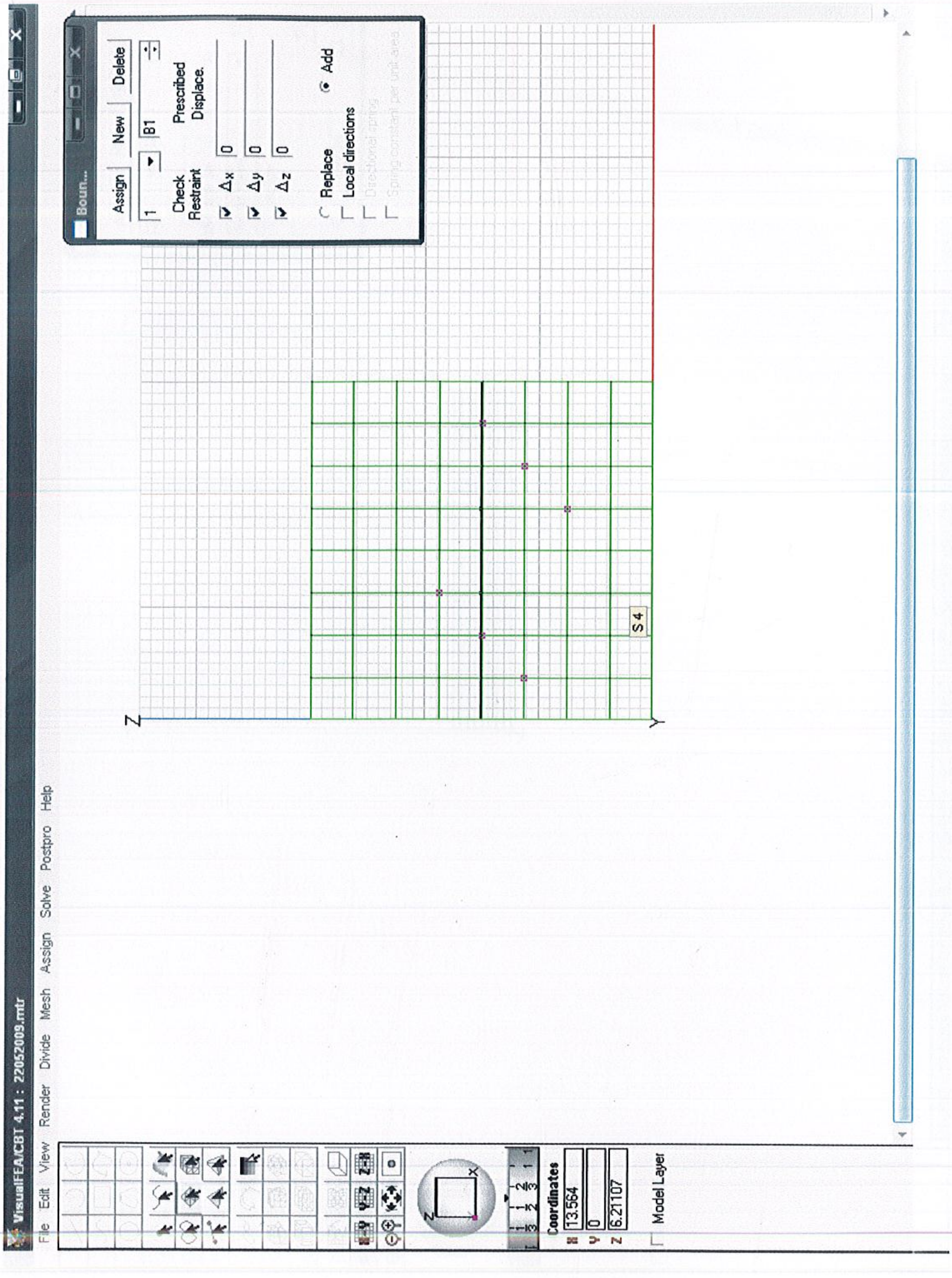

Fig. C-6 Applying Boundary Conditions

Step 7: Apply the load conditions as desired.

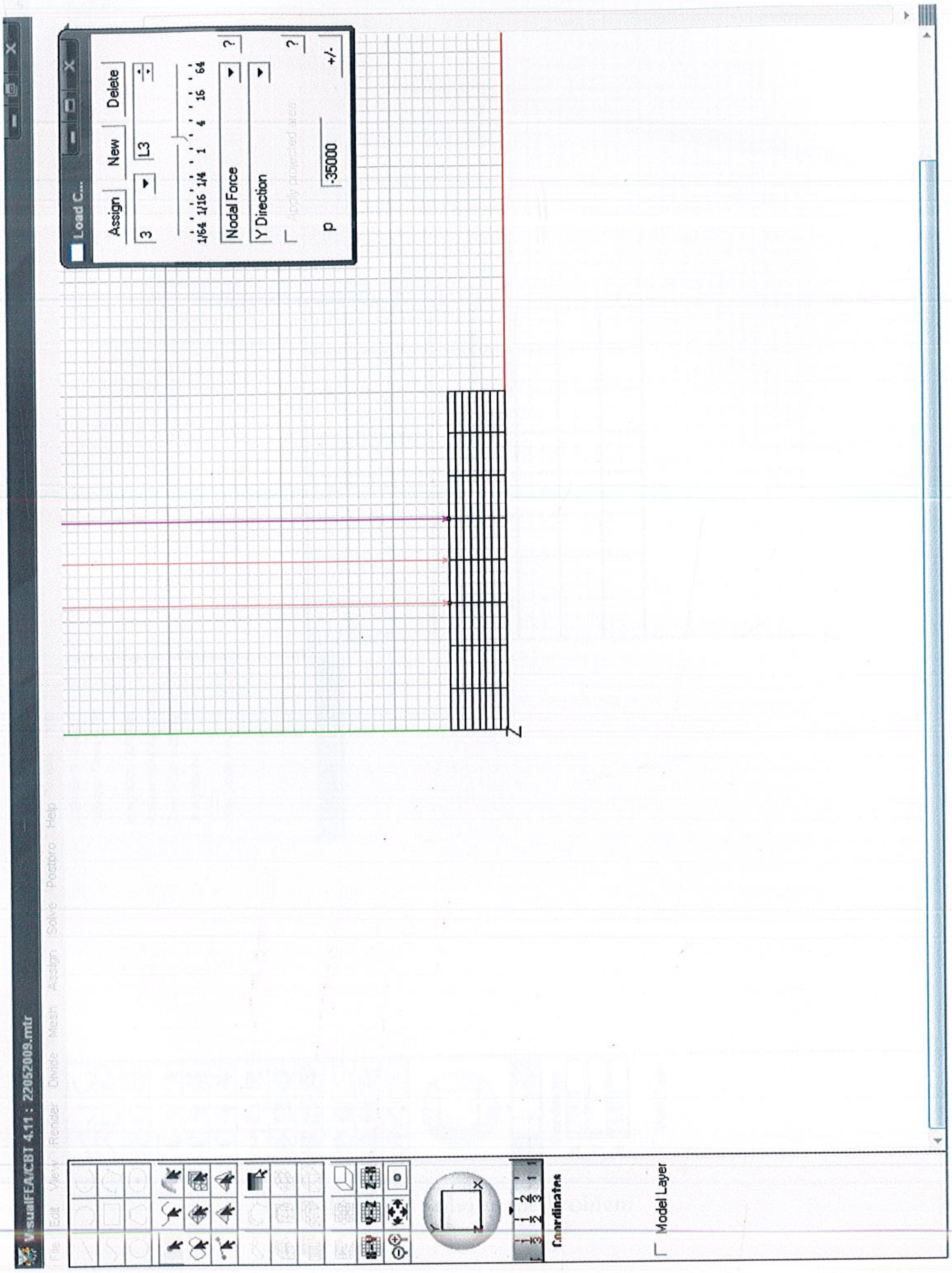

Fig. C-7 Application of loads

Step 8: From *Solve* menu select 'solve'.

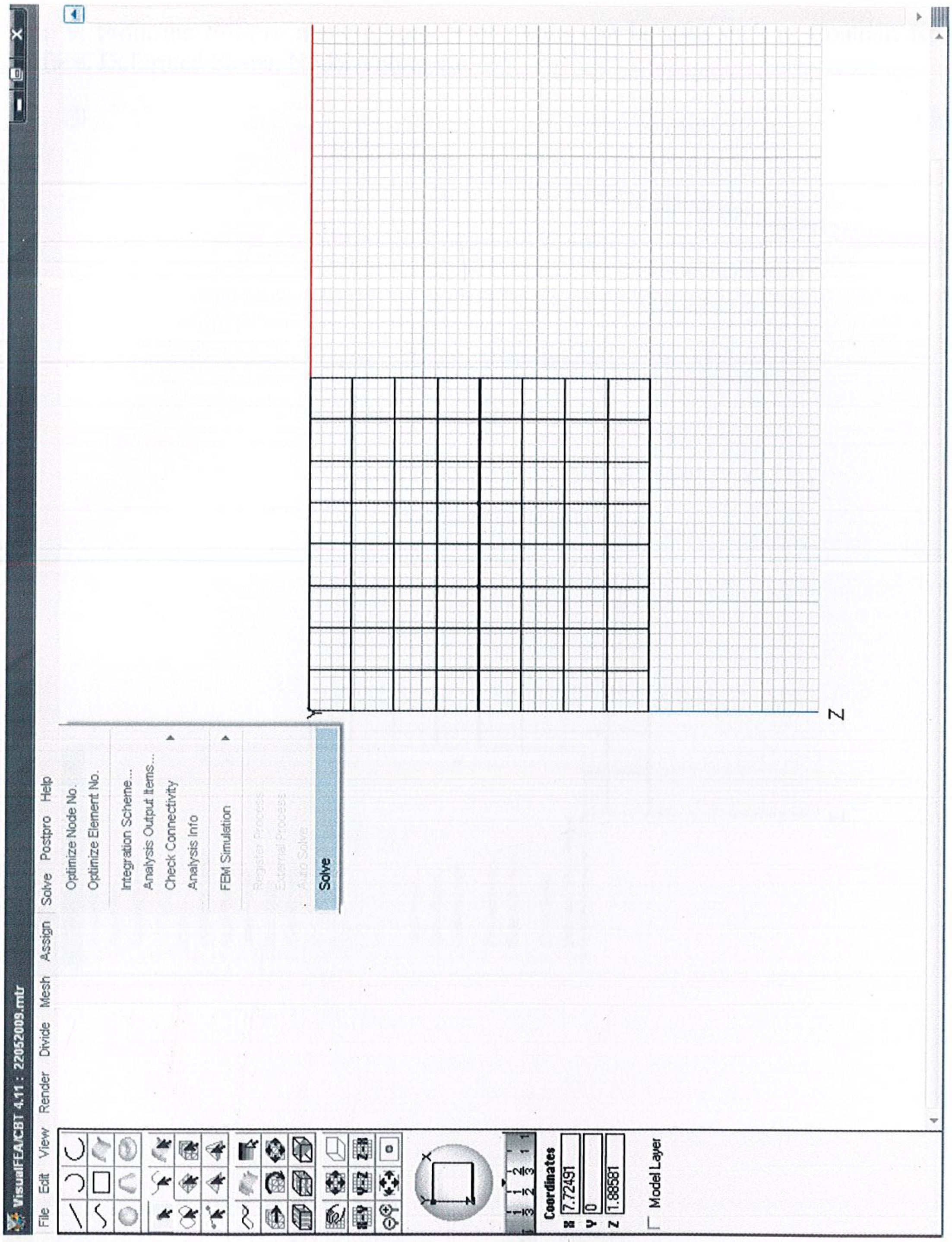

Fig. C-8 Solving the Problem

Step 9: From the *Postpro* menu a number of results can be seen such as Contour, Iso-surface, Deformed Shape, Nodal Values etc.

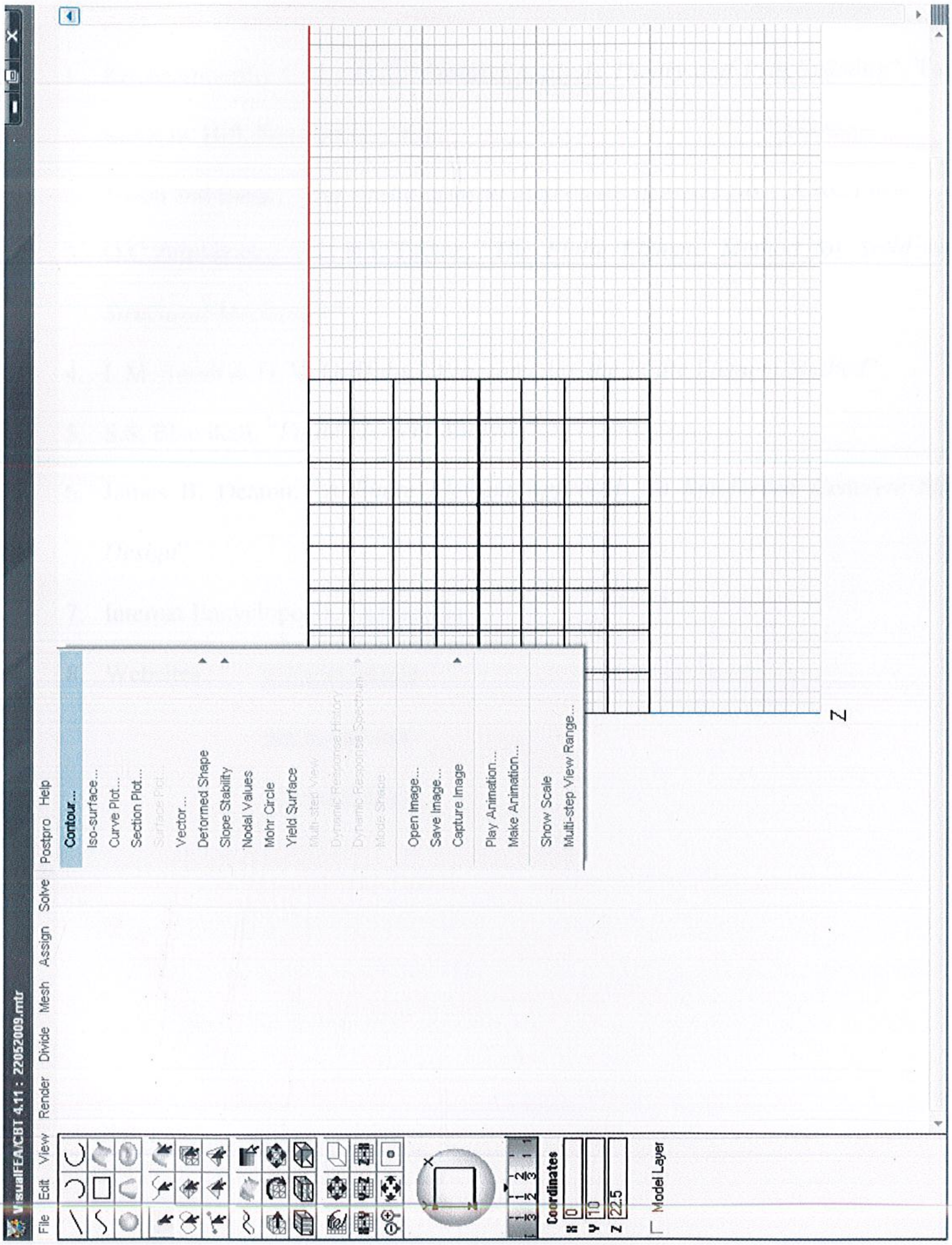

Fig. C-9 Visual FEA post processing

REFERENCES

1. Krishnamoorthy C S, "*Finite Element Analysis Theory and Programming*", Tata McGraw Hill, New Delhi, 2001.
2. Kwon and Bang , "*The Finite Element Method using MATLAB*" , CRC Press,
3. O.C.Zeinkiewicz and R.L.Taylor, "*The Finite Element Method for Solid and Structural Mechanics*".
4. I. M. Smith & D. V. Griffiths, "*Programming the Finite Element Method*"
5. S.S. Bhavikati, "*Finite Element Analysis*"
6. James B. Deaton, "*A Finite Element Approach To Reinforced Concrete Slab Design*"
7. Internet Encyclopedia – wikipedia
8. Websites : www.sv.vt.edu

   www.asce.org

   www.astm.org