International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in

Vol.4., Issue.4., 2016
(July-August)

RESEARCH ARTICLE

ISSN: 2321-7758

NONLINEAR ANALYSIS OF REINFORCED CONCRETE BEAM USING ANSYS

Dr. K. RAJASEKHAR[1], ANIRUDH GOTTALA[2], M.MAHESH[3], NISHANTH PAPPALA[4]

[1,2,3]Department of Civil Engineering, Andhra University

[4]Department of Civil Engineering, Shri Venkateswara University

Visakhapatnam, Andhra Pradesh, India.

ABSTRACT

Concrete structural components exist in buildings in different forms. Understanding the response of these components during loading is crucial to the development of an overall efficient and safe structure. Different methods have been utilized to. Experimental based testing has been widely used as a means to analyse the response of structural components. While this is a method that produces real life response, it is extremely laborious. The performed study investigation attempts to compare the results from elastic analysis of a reinforced beam under transverse loading, using an analysis software package to that obtained from a normal theoretical analysis. In the present study, the non-linear response of RCC beam using FE Modeling under the incremental loading has been carried out. The variation in load displacement curve, the crack patterns, the stress distribution and the effect of the non-linear behavior of concrete and steel on the response of RCC beam were observed.

Keywords: RCC Beam, ANSYS, Finite Element Modeling, Non-Linear Analysis, Crack Pattern, Stress distribution, Load – Displacement Curve.

I. INTRODUCTION

A concrete beam is a structural element that carries load primarily in bending. Bending causes a beam to suffer compression and tension. Beams carry both vertical and horizontal loads. The loads carried by a beam are transferred to columns, walls, which is then transferred to foundations. The compression section must be designed to resist buckling and crushing, while the tension section must be able to adequately resist to the tension. Experimental based testing has been widely used as a means to analyse individual elements and the effects of concrete strength under loading.

Beams support mainly vertical loads and are small in cross-section compared with their span. A beam is structural member which spans horizontally between supports and carries loads which act right angles to the length of the beam. A beam is in pure bending when the shear stress in the beam is zero, and the bending moment is constant. A beam that is initially straight will deflect under load. In the problems that are of most interest to us the deflections will be small compared to the span of beam.

A beam is a structural element that is capable of withstanding load primarily by resisting bending. Internally, beams experience compressive, tensile and shear stresses as a result of the loads applied to them. Typically, under gravity loads, the original length of the beam is slightly reduced to enclose a large radius arc at the top of beam, resulting in compression, while the same original beam length at the bottom of the beam is slightly stretched to enclose a large radius arc and so is under tension.

In some reinforced concrete beams the concrete is entirely in compression with tensile forces taken by steel tendons. These beams are known as prestressed concrete beams and are fabricated to produce a compression more than the expected tension under loading conditions. The strength of RCC beam depends on the composite action of concrete and steel. The properties of the beams with regard to their resistance to deformation must be taken into consideration.

- Carries transverse external loads that cause bending moment, shear forces and in some cases torsion.
- Concrete is strong in compression and very weak in tension.
- Steel reinforcement is used to take up tensile stresses in reinforced concrete beams.

Concrete structural components exist in building and other structures in different forms. Understanding the response of these components during loading is crucial to the development of an overall efficient and safe structure. Different methods have been utilized to study the response of structural components. Experimental based testing has been widely used because it produces real life response, but the method is extremely time consuming. Therefore, the use of finite element analysis to study these components has been also used.

II.INTRODUCTION OF BEAMS AND METHODOLGY

A beam is a structural element that is capable of withstanding load primarily by resisting bending. The bending force induced into the material of the beam as a result of the external loads, own weight, span and external reactions to these loads is called a bending moment. Beams are characterized by their profile (shape of cross-section), their length, and their material.

A. There are three types of reinforced concrete beams

1. Singly Reinforced Beam:

A singly reinforced beam is one in which the concrete element is only reinforced near the tensile face and the reinforcement, called tension steel, is designed to resist the tension.

2. Doubly Reinforced Beam:

A doubly reinforced beam is one in which besides the tensile reinforcement the concrete element is also reinforced near the compressive face to help the concrete resist compression. The latter reinforcement is called compression steel. When the compression zone of a concrete is inadequate to resist the compressive moment (positive moment), extra reinforcement has to be provided if the architect limits the dimensions of the section.

3. Singly or Doubly Reinforced Flanged Beams:

In most reinforced concrete structures, concrete slabs and beams are cast monolithic. Thus beams form part of the floor system together with the slab. In a floor system consisting of several beams cast monolithically with the slab the intermediate beams act as T-beams and the end beams and beams round the staircase or lift opening act as L-beam. The top of the T-beam serves as a flange or a compression member in resisting compressive stresses. The web of the beam below the compression flange serves to resist shear stress and to provide greater separation for the coupled force of bending.

B. Stresses in Beams

Internally, beams experience compressive, tensile and shear stresses as a result of the loads applied to them. Typically, under gravity loads, the original length of the beam is slightly reduced to enclose a smaller radius arc at the top of the beam, resulting in compression, while the same original beam length at the bottom of the beam is slightly stretched to enclose a larger radius arc, and so is under tension.

The primary tool for structural analysis of beams is the Euler Bernoulli beam equation. Other mathematical methods for determining the deflection of beams include "method of virtual work" and the "slope deflection method". Engineers are interested in determining deflections because the beam may be in direct contact with a brittle material such as glass. Beam deflections are also minimized for aesthetic reasons. A visibly sagging beam, even if structurally safe, is unsightly and to be avoided. A stiffer beam (high modulus of elasticity and high second moment of area) produces less deflection. Mathematical methods for determining the beam forces (internal forces of the beam and the forces that are imposed on the beam support) include the "moment distribution method", the force or flexibility method and the direct stiffness method.

C. Modes of Failure

In reinforced concrete beam design, failure is designed for safety. If the structure fails by sudden crushing, huge damage and loss of life may take place. So, it is designed so that structure would give a warning before collapse. Reinforced concrete beam failure may take place in 3 ways.

1. Concrete failure: Crushing of concrete
2. Steel failure: Yielding by steel
3. Concrete & steel combined failure

1 Concrete Failure: In concrete failure, concrete reaches to its ultimate strength before steel reaching to its yield point. As a result concrete crushes down before steel yields. This type of beam failure is unwanted as the concrete crushes without giving any warning. Concrete failure occurs due to the following reasons.

I. If large amount of reinforcement is used.
II. Compressive strength of concrete is less than tensile strength of steel.
III. Sudden destruction occurs without giving any warning.

2. Steel Failure: In steel failure, steel reaches to its yield point before concrete reaches to its ultimate strength. So, steel yields before crushing of concrete which gives ample warning to the inhabitants. So, steel failure is preferable during structure design. Some common notes on steel failure of RCC beams are given below.

I. Relatively moderate amount of reinforcement are employed.
II. Tensile strength of steel is less than compressive strength of steel.
III. Steel yields and stretches and tension crack propagates concrete cracks.
IV. Secondary compression failure.
V. Steel failure of RCC beams gives ample warning to the inhabitants.

3. Combined Concrete Beam Failure:

In this type of failure, concrete and steel fails at the same time. That means concrete should reach to its ultimate strength just at the same time when steel fails so that both fails together. Combined failure is practically almost impossible because it is not possible to tell the exact behavior of concrete & steel so that they fail at the same time.

D. Finite Element Method (FEM)

The finite element method constitutes a general tool for the numerical solution of partial differential equations in engineering and applied science. Historically, all major practical advances of the method have taken place since the early 1950s in conjunction with the development of digital computers. However, interest in approximate solutions of field equations dates as far backs in time as the development of the classical field theories (e.g. elasticity, electro-magnetism) themselves.

E. Employment of the Finite Element Method

An establishment of appropriate finite element model for an actual practical problem depends to a large degree on the following factors: understanding of the physical problem including a qualitative knowledge of the structural response to be predicted, knowledge of the basic principles of mechanics and good understanding of the finite element procedures available for analysis.

Discretization of the domain into finite elements is the first step in the finite element method. This is equivalent to replacing the domain having an infinite number of degrees freedoms by a system having finite number degrees of freedom. The shape, size, number and configuration of elements having had to be chosen carefully so that the original body or domain is simulated as closely as possible without increasing the computational effort needed for the solution. The various considerations taken in the discretization process are:

- Type of element

- Size of element
- Location of nodes
- Number of elements
- Simplifications afforded by the physical configuration of the body
- Finite representation of infinite bodies
- Node numbering

After meshing the body it is necessary to add the material properties, external loads, and apply the boundary conditions. Before start of the problem, only parameters of the calculation regime should be added to the input file.

1.Type of Element: Often the type elements to be used will be evident from the physical problem itself and geometry of the body. Let's consider briefly various types of finite elements which are subject to certain static and kinematic assumptions.

1.1 Truss and Beam Elements: Truss and beam elements are very widely used elements in structural engineering.

1.2 Plane Stress and Plane Strain Elements: Plane stress elements are employed to model membranes', the in-plane action of the beams and plates and so on. In each of these cases a 2-D stress situation exists in the x-y plane with the stresses equal to zero. Plane strain elements are used to represent a slice (of unit thickness) of a structure in which the strain components are zero. This situation arises in the analysis of long dam retaining wall and so on.

1.3 Plate and Shell Elements: The basic proportion in the plate and shell analysis is that the structure is thin in one dimensional and therefore the following assumptions can be made:

1. The stress through the thickness of plate or shell is zero.
2. Material particles that are originally on a straight line perpendicular to the mid-surface of the plate/shell remain on a straight line during deformation.

2 Size of Elements

Size of elements influences the convergence of the solution directly and hence it has to be chosen with care. If the size of elements is small the final solution is expected to be more accurate. Another characteristic related to the size of elements, which affects the finite element solution, is the aspect ratio of elements. The aspect ratio describes the shape of element in the assemblages of elements. For two-dimensional elements the aspect ratio is taken as the ratio of the largest dimension of the element of the smallest dimension. Elements with an aspect ratio of nearly unity generally yield best results.

3 Location of Nodes: If the body has no abrupt changes in geometry, material properties and external conditions, the body can be divided into equal subdivisions and hence the spacing of nodes can be uniform. On the other hand, if there are any discontinuities in the problem, nodes have to be introduced obviously.

4 Number of Elements : The number of elements to be chosen for idealization is related to the accuracy desired, size of elements and number of degrees of freedom involved. An increase in the number of elements generally means more accurate result. However the use of large number of elements involves large number of degrees of freedom.

F. Finite Element Program Packages: The ANSYS is computer software for finite element analysis and design. The ANSYS is a general purpose software, meaning that you can use it for almost any type of finite element analysis in virtually and industry automobiles, aerospace, railways, machinery, electronics etc, to mention just a few. "General purpose "also refers to the fact that the program can be used in all disciplines of engineering structural, mechanical, electrical, fluid and biomedical.

G. Performing a Typical ANSYS Analysis: The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines. The next few sections of this chapter cover general steps that are common to most analyses.

A typical ANSYS analysis has three distinct steps:

1.1 Building a Model: Building a finite element model requires more of an ANSYS user's time than any other part of the analysis. First, you specify a job name and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.The ANSYS element library contains more than 100 different element types. Each element type has a unique number and a prefix that identifies the element category. The following element categories are available: BEAM4 for example, has six structural degrees of freedom is a line element, and can be modeled in 3-D space. PLANE77 has a thermal degree of freedom (TEMP), is an eight-node quadrilateral element, and can be modeled only in 2-D space.

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. For example, real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS). Most element types require material properties. Depending on the application, material properties may be:

- Linear or nonlinear
- Isotropic, orthotropic, or anisotropic
- Constant temperature or temperature-dependent.

2 Apply Loads and Obtain the Solution: In this step, we use SOLUTION menu to define the analysis type and analysis options, apply loads, specify load step options and initiate the finite element solution. The analysis type is chosen based on the loading conditions and the response we wish to calculate. You can apply loads on the model in a variety of ways in the ANSYS program. Also, with the help of load step options, you can control how the loads are actually used during solution.

Loads are divided into six categories: DOF constraints, forces (concentrated loads), surface loads, body loads, inertia loads, and coupled-field loads.

- A DOFconstraint fixes a degree of freedom (DOF) to a known value. Examples of constraints are specified displacements and symmetry boundary conditions in a structural analysis, prescribed temperatures in a thermal analysis, and flux-parallel boundary conditions.
- A force is a concentrated load applied at a node in the model. Examples are forces and moments in a structural analysis, heat flow rates in a thermal analysis, and current segments in a magnetic field analysis.
- A surfaceload is a distributed load applied over a surface. Examples are pressures in a structural analysis and convections and heat fluxes in a thermal analysis.
- A bodyload is a volumetric or field load. Examples are temperatures and fluencies in a structural analysis, heat generation rates in a thermal analysis, and current densities in a magnetic field analysis.
- Inertialoads are those attributable to the inertia (mass matrix) of a body, such as gravitational acceleration, angular velocity, and angular acceleration. You use them mainly in a structural analysis.
- Coupled-fieldloads are simply a special case of one of the above loads, where results from one analysis are used as loads in another analysis. For example, you can apply magnetic forces calculated in a magnetic field analysis as force loads in a structural analysis.

Most of these can be applied either on the solid model (key points, lines and areas) or the finite element model (nodes and elements)

3 Review the Results

After SOLVE command the ANSYS program takes model and loading information from the database and calculates results. Results are written to the result file and also to the database. The difference is that only one set of results can reside in database at one time, whereas all sets of results (for all sub steps) can be written to the result file. After building the model and obtaining the solution, the postprocessors in the ANSYS program helps to review the results.

3.1 Application of FEA

The following is a more specific list (but by no means comprehensive) of possible analysis types and applications:

- Linear and non-linear (e.g. elastic-plastic) static stress/displacement analysis (buckling, thermal loading, quasi-static analysis)
- Dynamic stress/displacement analysis (modal and transient modal dynamic analysis, non-linear transient analysis e.g. impact)
- Non-linear stress analysis for the simulation of manufacturing processes, e.g. metal forming
- Contact analyses to simulate contact between parts and assemblies
- Analyses of a welded and/or bolted connections
- Fatigue and fracture analysis
- Sub-modeling to study a local part of a larger model
- Roll-over and falling-object protective structures
- Steel & reinforced concrete buildings/frames.
- Analyses of structures due to fire.

3.11 Principles of FEA

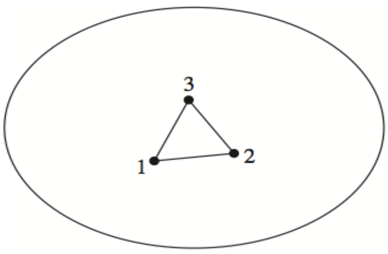
- The finite element method (FEM) or finite element analysis (FEA) is a computational technique used to obtain approximate solutionsof boundary value problemsin engineering.
- Boundary value problems are also called field problems. The field is the domain of interest and most often represents a physical structure.
- The field variablesare the dependent variables of interest governed by the differential equation.
- The boundary conditionsare the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field.

3.12 Shape Function

- The values of the field variable computed at the nodes are used to approximate the values at non-nodal points (that is, in the element interior) by interpolation of the nodal values. For the three-node triangle example, the field variable is described by the approximate relation
  $\Phi(x, y) = N1(x, y)\ \phi1 + N2(x, y)\ \phi2 + N3(x, y)\ \phi3$
- Where ϕ1, ϕ2, and ϕ3are the values of the field variable at the nodes, and N1, N2, and N3 are the interpolation functions, also known as shape functions or blending functions.
- In the finite element approach, the nodal values of the field variable are treated as unknown constantsthat are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes.
- The interpolation functions are predetermined, knownfunctions of the independent variables; and these functions describe the variation of the field variable within the finite element.

3.13 Degrees of Freedom

Again a two-dimensional case with a single field variable ϕ(x, y). The triangular element described is said to have 3 degrees of freedom, as three nodal values of the field variable are required to describe the field variable everywhere in the element (scalar).



In general, the number of degrees of freedom associated with a finite element is equal to the product of the number of nodes and the number ofvalues of the field variable(and possibly its derivatives) that must be computed at each node.

Φ(x, y) = N1(x, y) ϕ1 + N2(x, y) ϕ2 + N3(x, y) ϕ3

IV EXPERIMENTAL ANALYSIS, MODELING AND DISCUSSION OF RESULTS

The objective of the experimental investigation is to assess the properties of concrete mix used in the study. The tests are conducted to obtain the deflection, stresses and behavior of the beam in the laboratory. The grade of concrete adopted for all concrete mixes was M20 since it is widely used structural concrete all over the world

A. Materials: The various materials used in the experimental analysis are namely cement, fine aggregate, coarse aggregate. The specifications and properties of these materials were presented in subsequent order. All the materials used in study were tested in accordance to the Indian standards

1 Cement (IS 4112): Cement used in the experimental study was Ordinary Portland Cement of 43 grade. Cement used was fresh, of uniform color, free from any lumps and foreign matter, and from the same batch. The properties of the cement used were as shown in Table 4.1.

2 Course aggregate: Course aggregate used are from local sources of crushed granite. To obtain a reasonably good grading, 50% of the aggregate passing through 20mm I.S sieve and retained on 12.5mm I.S sieve and 50% of the aggregate passing through 12.5mm I.S sieve and retained on 10mm I.S sieve was used. The properties and test results of coarse conventional aggregate used are:

3 Water: Water is an important ingredient of concrete as it actively participates in chemical reaction with cement. Since quality of water affects the strength, it is necessary for us to use the pure and quality of water. Portable water from taps in laboratory was used for the mix preparation. The same water was used for mixing of concrete and curing of beams. Water from same source was used for curing.

B. Proportioning of Concrete Mix: The mix design M20 grade concrete adopted was in accordance to IS10262:2009. The mix is designed based on strength criteria, durability criteria for mild environment exposure and for good quality control.

1 Mix Proportion (M20): Ordinary Portland cement confirming to IS 12269 of 43 grade was used for producing concrete. The specific gravity of cement was found to be 3.13. River sand with a specific gravity of 2.60 and fineness modulus of 2.52 was used as fine aggregate. Coarse aggregates of size 20mm down and of average specific gravity 2.84 were used.

2 Concrete mix proportions:

- Grade designation: M20
- Type of cement : ordinary Portland cement 43 grade
- Maximum nominal size Of aggregate: 20mm
- Minimum cement content : 320Kg/m^3
- Maximum water cement ratio : 0.56
- Workability : 75
- Exposure condition : moderate
- Degree of supervision : good
- Type of aggregate : crushed angular aggregate 20mm
- Maximum cement content : 320 Kg/m^3

4.3 Tests and Test Specimens: Test specimens were casted for M20 mix to arrive at the strength. Study was carried out to arrive at the increment of the flexural strength of M20 concrete. The strength property aspect studied in this experimentation was as follows.

4.3.1 Casting of Beam Specimens : The steel mould of size 1000x230x300mm is taken and properly oiled. The reinforcement of 4 12mm bars at bottom and 2 12mm bars as hanger bars is placed in the mould. The design mix M20 grade concrete is slowly poured into the mould till the mould is completely filled as shown in Fig.4.2. Specimens are demoulded after 24 hours and placed in the water bath for curing till the day of test.

4.3.2 Procedure: Test specimens are removed from water bath at a temperature of 24^{o}c to 30^{o}c before testing. They are tested immediately on removal from the water bath. The dimensions of the beam should be noted before testing. No preparation of

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in
Vol.4., Issue.4., 2016
(July-August)

the surfaces is required. It is tested in the UTM by single point load.

Fig.4.1. Universal Testing Machine

Fig.4.2. Beam Specimen

Fig.4.3 RCC Beam placed in UTM for testing

Fig.4.4 crack developed in the RCC beam subjected to loading

4.3.3 Placing the specimens in the testing machine:

- The bearing surfaces of the supporting and loading rollers are wiped clean, any loose sand or other materials removed from the surfaces of the specimen where they are to make contact with rollers.
- The specimen is placed in the machine in such a manner that the load is applied to the uppermost surfaces as cast in the mould, along two lines spaced apart.
- The axis of the specimen is carefully aligned with the axis of the loading device.
- No packing is used between the bearing surfaces of the specimen and the rollers. The load is increased until the specimen fails, and the maximum load is applied to the specimen during the test recorded.
- The appearance of the fractured faces of concrete and any usual features is noted

4.4 Steps Involved in the Finite Element Analysis:

4.4.1 Preprocessing

Define the geometric domain of the problem.
Define the element type(s) to be used.
Define the material properties of the elements.
Define the geometric properties of the elements (length, area, and the like).
Define the model of beam (mesh the model).
Define the physical constraints (boundary conditions).
Define the loadings.

4.4.2Solution

Computes the unknown values of the primary field variable(s)

Computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, deflections and heat flow.

4.4.3Post processing

Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution.

4.5 Modeling Procedure:

Preprocessing

4.5.1.1 Defining the problem:

In this problem we have to go to file option in the menu controls and title of the problem is changed.

4.5.1.2 Define element types: Element types that are used in creating the model are presented in Table 4.4

Table 4.4 Element types for modeling

Material type element	ANSYS
Concrete	Solid 65
Steel reinforcement	Beam188

SOLID65 allows the presence of four different materials within each element, one matrix material (*e.g*. concrete) and a maximum of three independent reinforcing materials. Concrete material is capable of directional integration point cracking and crushing besides incorporating plastic and creep behavior. Reinforcement (which also incorporates creep and plasticity) has uniaxial stiffness only and is assumed to be smeared throughout the element. Directional orientation is accomplished through user specified angles. Concrete material is assumed to be initially isotropic. The Solid65 element was used to model the concrete. This element has eight nodes with three degrees of freedom at each node translations in the nodal *x*, *y*, and *z* directions. This element is capable of plastic deformation, cracking in three orthogonal directions, and crushing. A schematic of the element is shown in Fig.4.5.

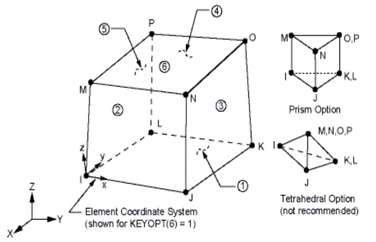


Fig 4.5 Element Solid65 3-D reinforced concrete

A BEAM188 element was used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degrees of freedom – translations in the nodal *x*-, *y*-, and *z*-directions, being also capable of plastic deformation. This element is shown in Fig.4.6.

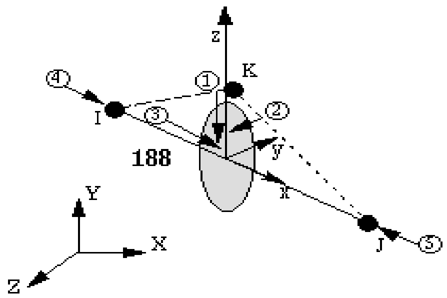


Fig 4.6 Beam 188 3-D spar element

4.5.1.3 Define real constants of the elements:

Individual elements contain different real constants. No real constant set exists for the Solid45 element. Real Constant Set 1 is used for the Solid65 element. It requires real constants for rebar assuming a smeared model. Values can be entered for material number, volume ratio, and orientation angles. The material number refers to the type of material for the reinforcement. The volume ratio refers to the ratio of steel to concrete in the element. The orientation angles refer to the orientation of the reinforcement in the smeared model.

ANSYS allows the user to enter three rebar materials in the concrete. Each material corresponds to *x*-, *y*-, and *z*-directions in the element. The reinforcement has uniaxial stiffness and the directional orientation is defined by the user. In the present study the beam is modelled using discrete reinforcement. Therefore, a value of zero was entered for all real constants which turned the smeared reinforcement capability of the Solid65 element off. Real Constant Sets 2, 3, 4, and 5 are defined for the Link8 element. BEAM 188 elements does not require any real constant.

Table 4.5 Real Constants

Real constant set	Element Type		Real constants for rebar 1	Real constants for rebar 2	Real constants for rebar 3
1	Solid 65	Material no	0	0	0
2	BEAM 188	Initial strain	113.1	0	0

4.5.1.4 Material properties: Two material models were given: material 1 for concrete and material 2 for steel, under the linear isotropic material definition. Parameters needed to define the material models are shown in Table 4.6. There are multiple parts of the material model for combined model to account for the nonlinearity.

Table 4.6 Material properties

Element	Material Properties	
Solid 65	Linear Isotropic	
	Young's Modulus EX	25000 N/mm^2
	Poisson's Ratio PRXY	0.2
BEAM 188	Linear Isotropic	
	Young's Modulus EX	2.1x10^5 N/mm^2
	Poisson's Ratio PRXY	0.3
	Bilinear isotropic	
	Tangent modulus	2x10^{5}N/mm^2
	Yield stress	415N/mm^2

4.5.1.5 Cross-Section of the Beam and Elements: A cross section defines the geometry of the beam in a plane perpendicular to the beam axial direction. ANSYS supplies a library of eleven commonly-used beam cross section shapes, and permits user-defined cross section shapes. When a cross section is defined, ANSYS builds a numeric model using a nine node cell for determining the properties of the section.

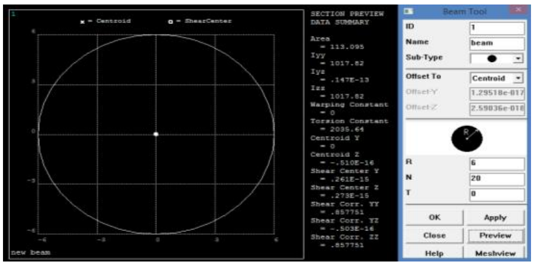


Fig 4.7 showing the sectional properties of the reinforcement

4.5.1.6 Modeling of Beam: The beam was modeled as volume and model was 1000mm long with a cross section of 230mx300mm the finite element model was shown in Fig.4.5.and the dimensions for concrete model was shown in Table 4.7.

Table 4.7 Dimensions for concrete beam

ANSYS	Concrete	
X_1, X_2, X co-ordinate	0	1000
Y_1,Y_2, Y co-ordinate	0	230
Z_1, Z_2, Z co-ordinate	0	300

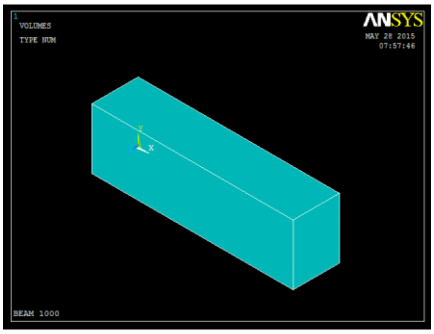


Fig 4.8 Finite Element Model of Beam

4.5.1.7 Meshing of the Model Beam and Reinforcement: For Solid65 element the mesh was set up such that square or rectangular elements were created. This properly sets the width and length of elements in the beam to be consistent with the elements and nodes in the concrete portions of the model. The necessary element divisions are noted. The meshing of the reinforcement is a special case compared to the volumes. The reinforcement model was meshed using line elements so that the nodes of the line elements come exactly over the node of the solid elements which are later merged so that both rebar elements and the concrete elements share the same nodes. To obtain accurate results from the solid65 element the use of rectangular mesh was recommended. The different mesh sizes that are made for the model are shown in Figs.4.9, 4.10, 4.11.

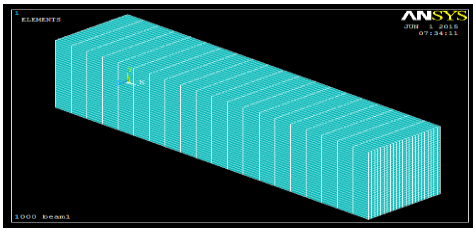


Fig 4.9 Beam model with 15624 mesh elements

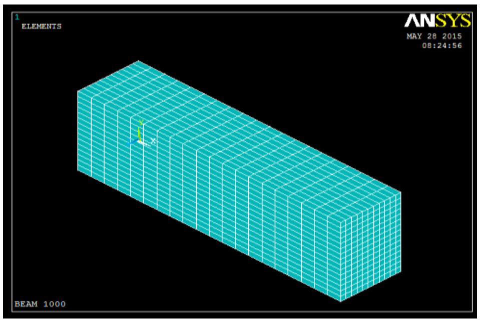


Fig 4.10 Beam model with 4032 mesh elements

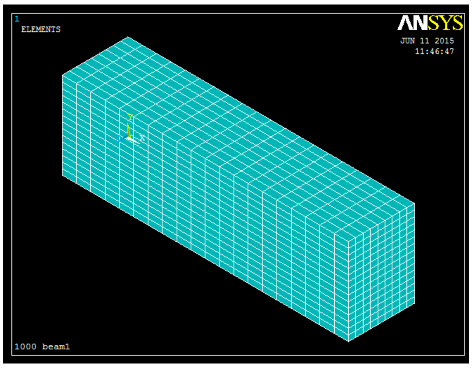


Fig 4.11 Beam model with 3360 mesh elements

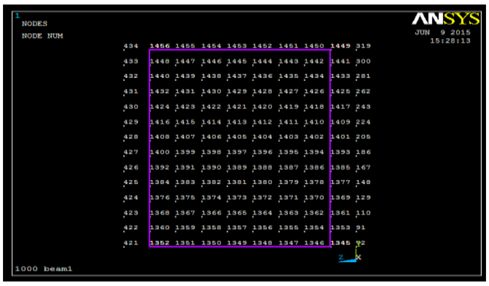


Fig 4.12 Meshing of transverse reinforcement

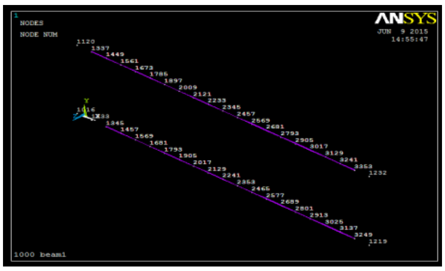


Fig 4.12(a) Meshing of longitudinal reinforcement

4.5.1.8 Boundary conditions: Displacement boundary conditions are needed to constrain the model to get a unique solution. To ensure that the model acts the same way as the experimental beam, boundary conditions need to be applied at points of symmetry and where the supports and loadings exist. The symmetry boundary conditions were set first. The model being used is symmetric about one plane. The support was modeled as a simply supported beam. Nodes on the plate were given constraint in all directions, applied as constant values of zero. The boundary conditions for both planes of symmetry are shown in Fig.4.13.

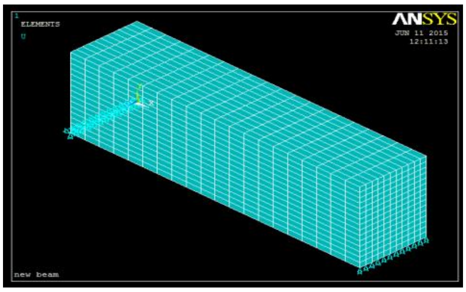
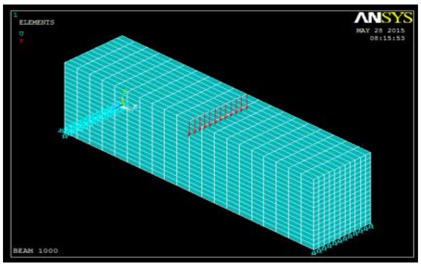
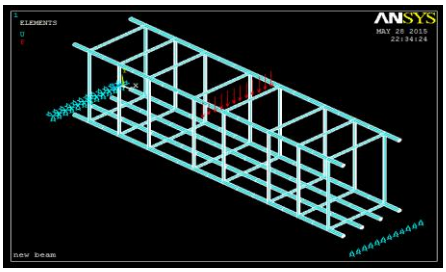


Fig 4.13 Beam model with boundary conditions

4.5.1.9 Loads on the Beam Model: Loads are to be applied on the model so as obtain the unique solution as in the case of the boundary of conditions. Loads are used to represent inputs to the system. They can be in the forms of forces, moments (torque), pressures, temperature, or accelerations. The loads are applied to the beam model on the nodes as shown in Fig.4.14.



Figs 4.14 showing the loading conditions of beam model



Figs 4.14(a) showing the loading conditions of beam model with reinforcement

4.5.2 SOLUTION

Once the mesh is complete, and the properties and boundary conditions have been applied, it is time to solve the model. In most cases submitting a run with multiple load cases will be faster than running sequential, complete solutions for each load case. The following steps are involved in obtaining the solution of the problem.

4.5.2.1 Define Analysis Type

4.5.2. 2 Solution Control

In the solution control the type of analysis is to be chosen.

Ensure the following selections are made:

- Ensure Large Static Displacements are permitted
- Ensure Automatic time stepping is on. Automatic time stepping allows ANSYS to determine appropriate sizes to break the load steps into. Decreasing the step size usually ensures better accuracy, however, this takes time.
- Enter any number of sub steps. This will set the initial sub step to $1/^{5th}$ of the total load.
- Enter a maximum number of sub steps of 1000. This stops the program if the solution does not converge after 1000 steps.
- Enter a minimum number of sub steps of 1.
- Ensure all solution items are written to a results file.

4.5.2.3 Apply Constraints

In this solution is selected and in defining the loads boundary conditions are to be applied.

4.5.2.4 Apply Loads

In defining the loads the loads are applied on the structure in the form of force or moment over the structure.

4.5.2.5 Solve the System

In solving the problem the menu control solve is selected and current solve is to be chosen to obtain the solution.

4.5.3 Pot-Processing

Unexpectedly high or low displacements (by order of magnitude) could be caused by an improper definition of load and/or elemental properties.

4.5.3.1 View the deformed shape

In order to view the deformed shape of the model, post processing option is to be selected for plotting results for deformed model.

4.5.3.2 View the deflection contour plot

To obtain the deflection contour plot in general post processing the nodal solution for displacement vector sum for UY direction is to be selected.

4.6 Validation of Model

The beam was modeled using ANSYS as volume and is meshed with 3360 number of elements as trial model. The load values obtained from experiment are applied on the model as boundary condition. The deflections and stresses observed in the analytical model are closer to the experimental values. This indicates the analytical model is validated. Hence in the same Chapter beam models were modeled for the required problem with and different numbers of elements are meshed to obtain accurate results.

4.7 Experimental work

The experimental program consists of casting and testing of beams of 1000mm length reinforced concrete beams. All the beams were tested over simply supported conditions. The beams are designed as under reinforced section to sustain loads. Tests are carried out as per Indian standards in Strength of Materials laboratory. The testing arrangement was shown in Fig.4.3. Single point load was applied on reinforced concrete beam of span 1000mm through hydraulic jack of capacity 1000kN. The beams were suitably instrumented for measuring the mid span deflection with dial gauges. The test results were shown in the Table 4.8

4.8 Finite element analysis (FEA) using ANSYS

The concrete beams were modeled using ANSYS 14.0 with different number of mesh elements with and without reinforcement. The deflection and Von Mises Stresses were observed after application of loads on the beam. The deflection values of concrete beam with and without reinforcement using ANSYS 14.0 were shown in Fig.4.15, 4.16, 4.17. Similarly von mises stresses of beams with and without reinforcement are shown Fig.4.18, 4.19, 4.20. Table 4.8 shows the mid span deflection at failure loads on the basis of Finite Element Analysis and as well as experimental analysis.

Table 4.8 Deflection values of beam observed from FEA model and experimental

Model		Deflection	Max. total deflection	Permissible Deflection(mm)
Experimental (With-out reinforcement)		0.68	1.2	4
FEA (With-out reinforcement)	Mesh size	0.93	1.1	4
	15624			
	4032	0.77	1.42	
	3360	0.79	1.68	
Experimental (With reinforcement)		2.5	3.68	4
FEA (With reinforcement)	Mesh size	2.44	3.62	4
	15624			
	4032	1.62	3.47	
	3360	2.02	3.66	

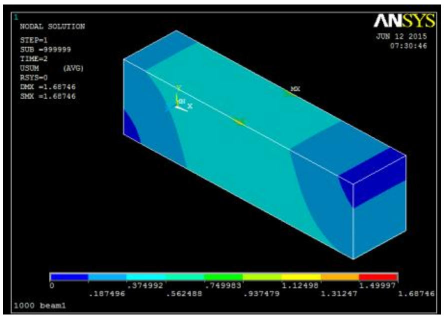


Fig. 4.15(a) Deflection contour plot of beam without reinforcement with 3360 number of elements

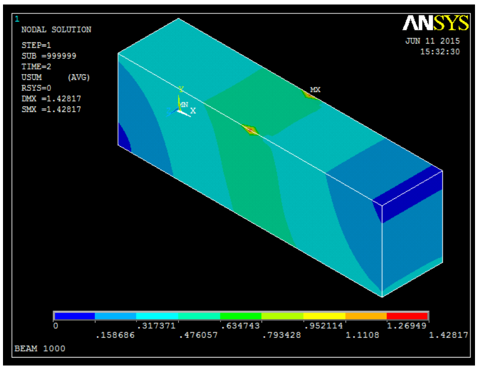


Fig. 4.16(a) Deflection contour plot of beam without reinforcement with 4032 number of elements

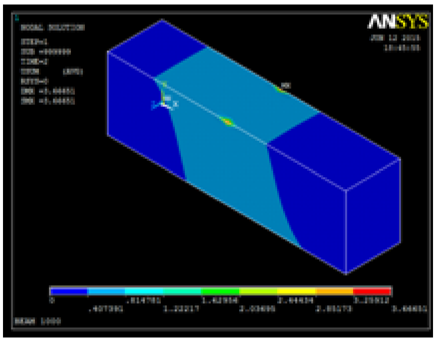


Fig. 4.15(b) Deflection contour plot of beam with reinforcement with 3360 number of elements

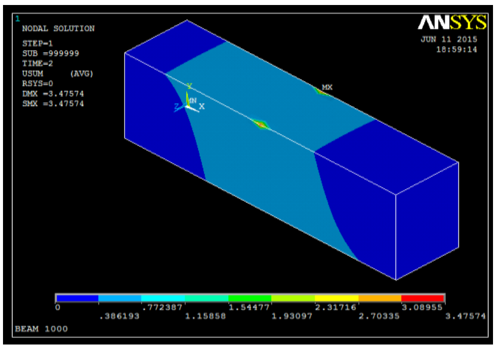


Fig. 4.16(b) Deflection contour plot of beam with reinforcement with 4032 number of elements

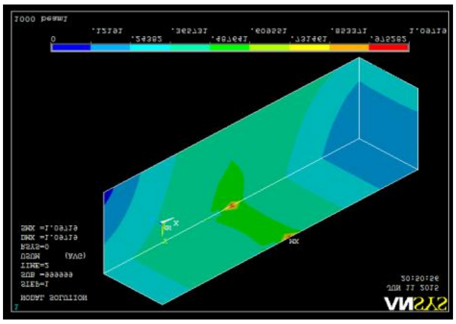


Fig.4.17 (a) Deflection contour plot of beam without reinforcement with 15624 number of elements

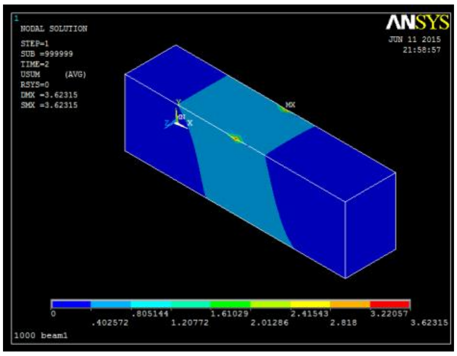


Fig.4.17 (b) Deflection contour plot of beam with reinforcement with 15624 number of elements

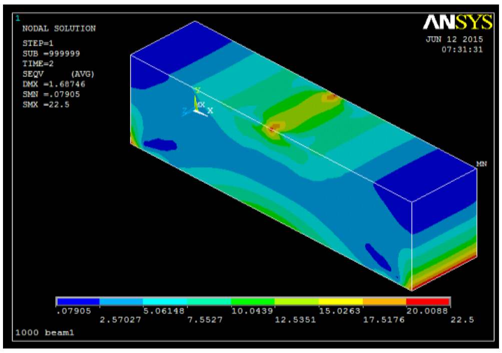


Fig. 4.18(a) Von mises stress contour plot of beam without reinforcement with 3360 number of elements

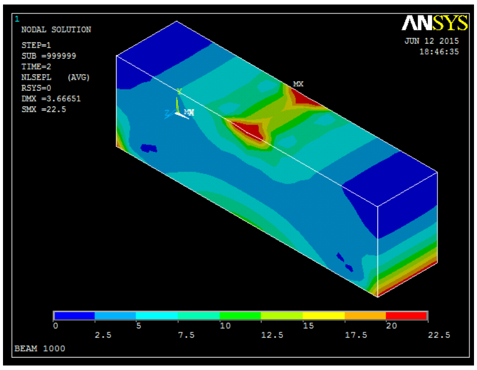


Fig. 4.18(b) Von mises stress contour plot of beam with reinforcement of 3360 number of elements

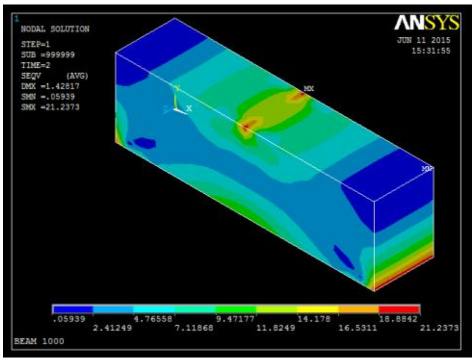


Fig. 4.19(a) Von mises stress contour plot of beam without reinforcement with 4032 number of elements

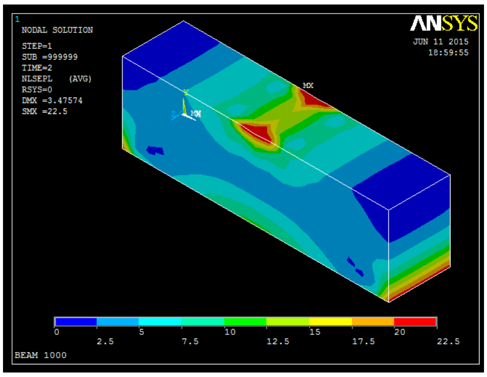


Fig 4.19(b) Von mises stress contour plot of beam with reinforcement with 4032 number of elements

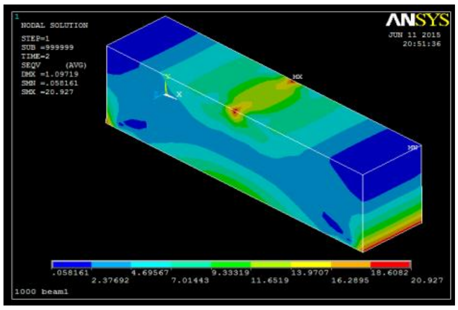


Fig.4.20 (a) Von mises stress contour plot of beam without reinforcement with 15624 number of elements

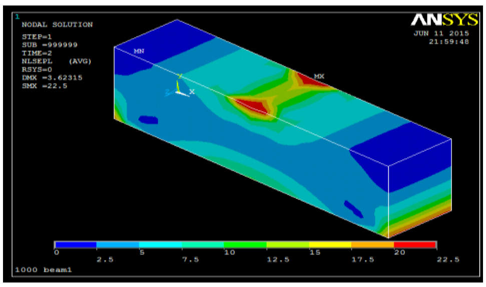


Fig. 4.20(b) Von mises stress contour plot of beam with reinforcement with 15624 number of elements

International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Email:editorijoer@gmail.com http://www.ijoer.in
Vol.4., Issue.4., 2016
(July-August)

Table 4.9 values of Deflection for the beam without/with reinforcement having 15624 number of elements

S.no	Load (kN)	Deflection (Without)	Deflection (With)
1	0	0	0
2	25	0.14	0.13
3	50	0.29	0.27
4	75	0.49	0.46
5	100	0.62	0.65
6	125	0.7	0.73
7	150	0.66	0.92
8	175	0.82	1.17
9	200	0.97	1.59
10	225	1.05	2.17
11	250	1.1	2.44

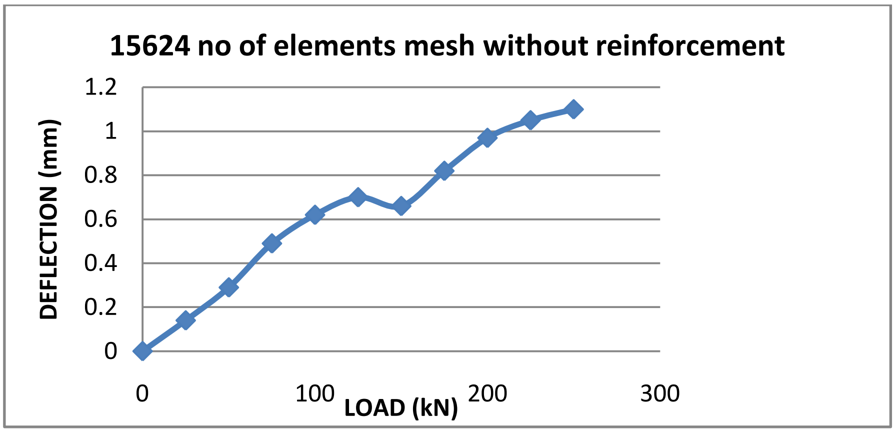


Fig.4.21 variation of deflection with load for the beam without reinforcement having 15624 number of elements

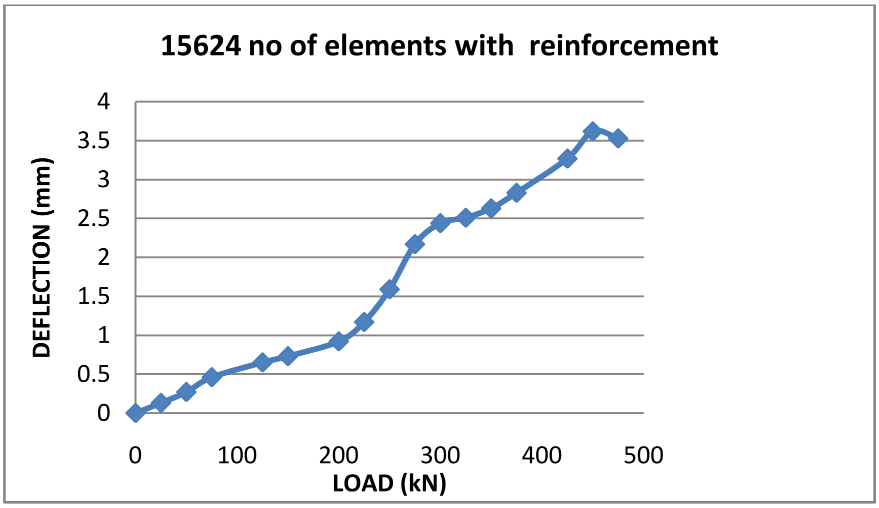


Fig 4.22 variation of deflection with load for the beam with reinforcement having 15624 number of elements

Table 4.11 values of Deflection for the beam without reinforcement having 4032 number of elements

S.no.	Load (kN)	Deflection (mm)
1	0	0
2	25	0.17
3	50	0.36
4	75	0.48
5	100	0.62
6	125	0.64
7	150	0.68
8	175	0.73
9	200	0.9
10	225	1.06
11	250	1.42
13	300	1.35

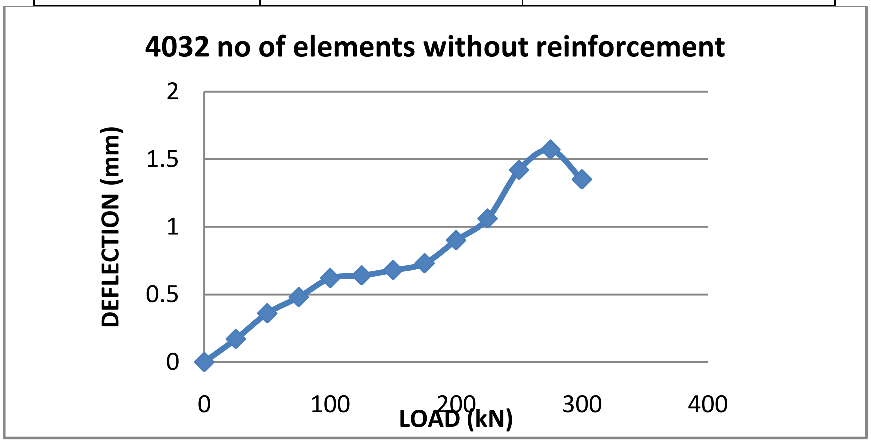


Fig 4.23 variation of deflection with load for the beam without reinforcement having 4032 number of elements

Table 4.12 values of Deflection for the beam with reinforcement having 4032 number of elements

S.no	Load (kN)	Deflection (mm)
1	0	0
2	25	0.11
3	50	0.24
4	75	0.33
5	100	0.49
6	125	0.47
78	150	0.65
9	175	0.73
10	200	0.86
11	225	1.15
12	250	1.43
13	275	1.53
14	300	1.62

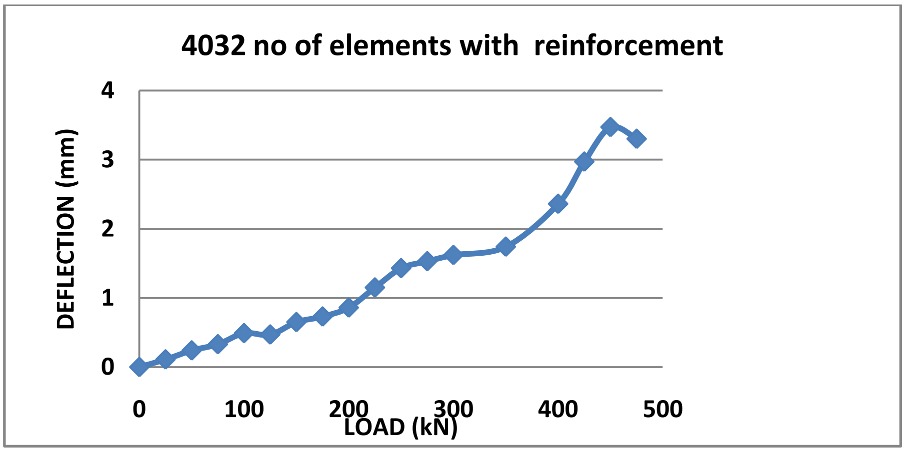


Fig 4.24 variation of deflection with load for the beam with reinforcement having 4032 number of elements

Table 4.13 values of Deflection for the beam without reinforcement having 3360 number of elements

S.no	Load (kN)	Deflection (mm)
1	0	0
2	25	0.26
3	50	0.3
4	100	0.54
5	150	0.71
6	175	0.89
7	200	1.07
8	225	1.35
9	250	1.68
10	275	1.59

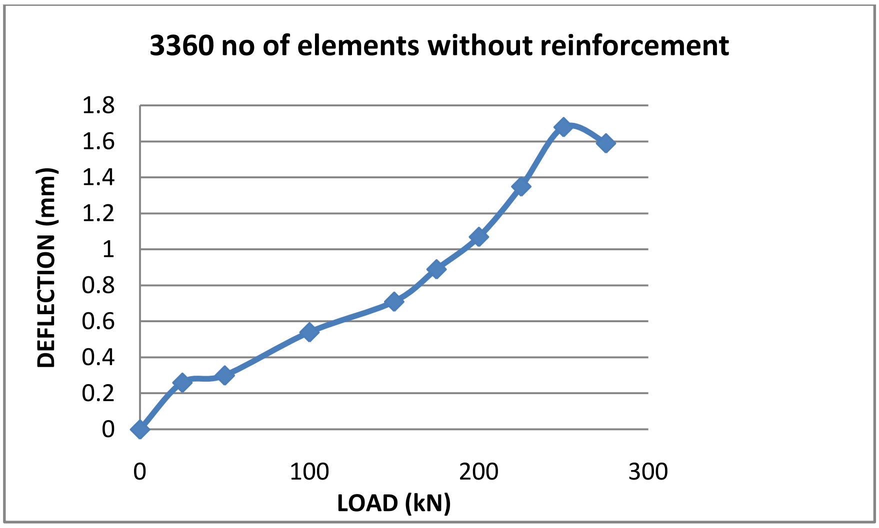


Fig 4.25 variation of deflection with load for the beam without reinforcement having 3360 number of elements

Table 4.14 Values of Deflection for the beam with reinforcement having 3360 number of elements

Sno	Load(kN)	Deflection(mm)
1	0	0
2	25	0.23
3	50	0.5
4	75	0.65
5	100	0.81
6	150	1
7	175	0.98
8	200	1.14
9	250	1.54
10	275	1.86
11	300	2.02
12	350	2.54
13	400	3.24
14	450	3.66

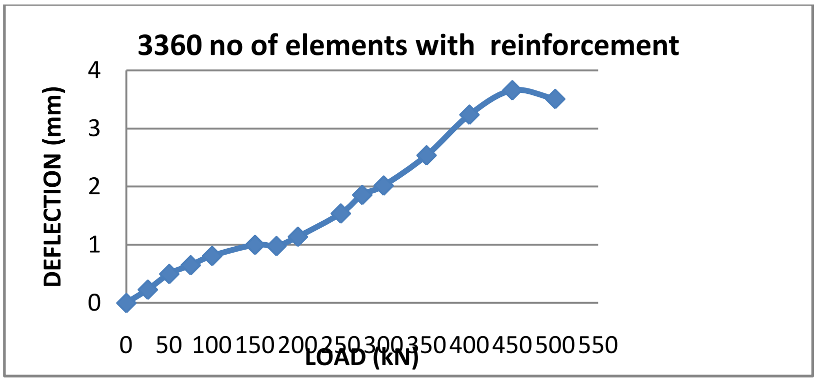


Fig 4.26 variation of deflection with load for the beam with reinforcement having 3360 number of elements

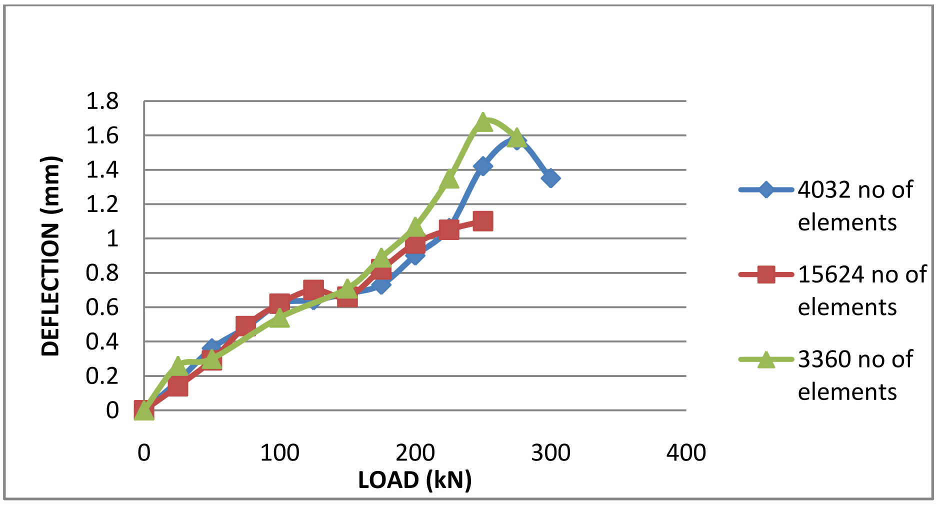


Fig.4.27 variation of deflection with load for the beams without reinforcement having different number of elements

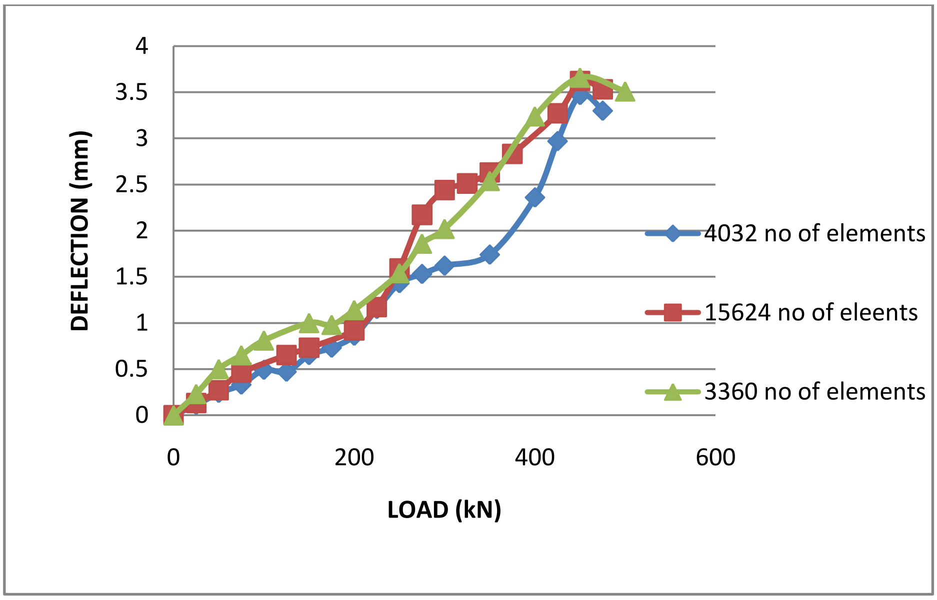


Fig.4.28 variation of deflection with load for the beams with reinforcement having different number of elements

4.9 Results and Discussion

The average initial cracking load for the beam with reinforcement was found to be at 300kN. Minor cracks were observed in shear region in the direction of the line joining the loading point and support at this load. Also minor flexural cracks were observed at mid span of the beam. Loading was continued beyond this point and the failure load observed was at 450kN. Whereas for beams without reinforcement the initial crack and failure loads were observed at 178kN and 250kN respectively and the deflections obtained are to be 0.68mm and 1.20mm respectively. The FE analysis has been carried with different number of mesh elements and the percentage error in deflection and stresses of FE analysis compared with that of experimental solution and are shown in Table 4.15, 4.16.

Table 4.15 values of deflection for both types of beams with and without reinforcement

Type of Beam	Load at Centre	Mesh Size(Number of Elements)	Deflection At Centre Point	Analytical Deflection At Centre	% difference in values
Beam without reinforcement	250	15264	1.1	1.2	8.3
Beam with reinforcement	450		3.62	3.68	1.63
Beam without reinforcement	250	4032	1.42	1.2	18.3
Beam with reinforcement	450		3.475	3.68	5.57
Beam without reinforcement	250	3360	1.68	1.2	28.5
Beam with reinforcement	450		3.66	3.68	0.55

4.10 Variation of Deflection: The experimental deflections were recorded and variations were drawn at mid-section of beam for experimental results. The variation of deflection was observed to be linear to some point and nonlinear beyond that point.

Table 4.16 values of stresses of beams by both experimental and analytical methods

Type of Beam	Load at Centre	Mesh Size(Number of Elements)	Stress	Analytical stress	% difference
Beam without reinforcement	250	15264	22.5	18.1	243
Beam with reinforcement	450		22.5	31.95	30
Beam without reinforcement	250	4032	21.23	18.1	17
Beam with reinforcement	450		22.5	31.95	30
Beam without reinforcement	250	3360	20.92	18.1	15.4
Beam with reinforcement	450		22.5	31.95	30

V CONCLUSIONS

The stresses of the finite element model are compared to the analytical stresses obtained from the experimental data.

- The stress observed in beam model without reinforcement with 15624 elements is less by 15% when compared to experimental.
- Finite element model with reinforcement with different number of elements have shown a difference of 30% when compared to experimental.
- Similarly beam model without reinforcement with 4032 and 3360 number of elements have shown 17% and 24.3% error when compared with the experimental results.

Compared to a beam without reinforcement, beams with reinforcement have adequate deformation capacity, in spite of their brittle mode of failure.It can be concluded that beam model with greater number of finite elements have obtained closer results to that of experimental. So, Beam with more number of finite elements can be modeled to study more precisely the behavior of RCC beam and to obtain accurate results using FEA (ANSYS).

REFERENCE

[1]. Anthony J. Wolanski, B.S. (2004) “Flexural Behavior of Reinforced and Prestressed Concrete Beams Using Finite Element Analysis”, Master’s Thesis, Marquette University, Milwaukee, Wisconsin.

[2]. Elavenil, S., Chandrasekar, V. (2007), “Analysis of reinforced concrete beams strengthened with ferrocement”, International Journal of Applied Engineering Research, Research India Publication, 2(3), pp 431440.

[3]. Franklin, H.A. (1970). "Non-Linear Analysis of Reinforced Concrete Frames and Panels". Ph.D. Dissertation, Division of Structural Engineering and Structural Mechanics, University of California, Berkeley, SEMM 70-5.

[4]. GargiMajumder; Kaushik Kumar, “Deflection and Stress Analysis of a Cantilever and its Validation Using ANSYS”.

International Journal of Mechanical Engineering and Research, 2013, ISSN 2249-0019, Volume 3, Number 6, pp. 655-662.

[5]. Mohammed Sh. Mahmood and Amer M. Ibrahim, "Finite Element Modeling of Reinforced Concrete Beams Strengthened with FRP Laminates", European Journal of Scientific Research ISSN 1450-216X Vol.30 No.4 (2009), pp.526-541.

[6]. Patil S S, Shaikh A N and Niranjan B R 2012 "Nonlinear finite element method of analysis of Reinforced Concrete Deep Beam". Int. J. Modern Eng. Res. 2(6): 4622–4628

[7]. Revathi, P., DevdasMenon. (2005), "Nonlinear finite element analysis of reinforced concrete beams, Journal of Structural Engineering, Structural Engineering Research Centre, Chennai, 32(2), pp 135137.