International Research Journal in Global Engineering and Sciences. (IRJGES)
ISSN : 2456-172X | Vol. 8 Issue 1 - March 2023 to May 2023| Pages 1-6

Modelling and Experimental Validation of High-Performance Concrete Beams Using Ansys

Dr. G. Dineshkumar[1], Dr. K Thirupathi Rao[2], Dr. H Vijay Kumar[3], A Ashok[4], D Santhosh[5]
[1,3]Associate Professor, [2]Professor, [4]Assistant Professor, [5]B.Tech Student
Department of Civil Engineering, Vaagdevi College of Engineering
Warangal, Telangana-506005

ABSTRACT

A three dimensional Finite Element Method (FEM) models were developed to simulate the behavior of high performance reinforced concrete beams, from linear through non linear response up to failure, using ANSYS 10.0. The solid 3D hexahedral element represent concrete, the solid 3D link element is used for discrete reinforcing steel bars, based on each component actual characteristics, linear material properties are defined for both elements. The high-performance concrete beams with reinforcing steel modeled discretely will be developed with results compared with experimentally tested beams. The load deflection response of the experimental beams will be compared to analytical prediction to calibrate the FE model for further use. The high-performance concrete beams were modeled with two parameters like partial replacement of silica fume of 5, 7.5, 10% alone and silica fume with fly ash of above mentioned percentage of replacement along with 10% replacement of fly ash. The same percentage of replacement is repeated for metakoalin. The shear behavior of beam is studied with above modeled beams. The model for flexural and shear behavior of beams was created using partial replacement of silica fume and metakaolin alone for cement . The different stages of response of HPRC beams are computed using FEA and compared the results with the experimental results. Comparisons were made for Load Deflection plots at failure. Modeling simplifications and assumptions developed during this research are presented.

Keywords: *Finite Element Method, metakoalin, silica fume, fly ash*

INTRODUCTION

The need to increase the capacity of the existing structures has recently become one of the most important criteria in Civil Engineering. Finite element method (FEM) models are developed to simulate the behavior of Full-size beams through nonlinear response and up to failure, using the ANSYS Program. Comparisons are going to be made for load-deflection plots of mid span on the beams, first cracking loads, loads at failure, and crack patterns at failure. The main obstacle to finite element analysis of reinforced concrete structures is the difficulty in characterizing the material properties. Much effort has been spent in search of a realistic model to predict the behavior of reinforced concrete structures. Due to the complexity of the composite nature of the material, proper modeling of such structures is a challenging task. This paper presents the numerical study to simulate the behavior of beams and columns strengthened by adding mineral admixture as Metakaolin and Silica Fume. The load deflection plots for the above cases obtained from numerical study are going to be plotted. The results from the finite element models will show good agreement.

OBJECTIVE OF THIS STUDY

The objective to this study is to verify

1. To generate the finite element modeling and to study the behavior of reinforced high performance concrete beams in shear and flexure behavior.
2. Validation of results obtained from ANSYS 10.0 model with the experimental results.

MATERIAL PROPERTIES

Table 1: Material properties

MATERIAL	PARTICULARS
CONCRETE	Poisson's ratio=0.2 Grade of concrete=60 MPa Modulus Of Elasticity E(according to replacements of additives added)
STEEL	Young's Modulus E=2*10^5 MPa Poisson's ratio=0.3, Yield strength=415 MPa

Table 2: summary of ANSYS 10.0 model of the specimen

CATEGORIES	ANSYS 10.0 MODEL DETAILS
Material properties Concrete Steel	As listed in Table 1
Model descriptions Length of beams Size of beams	Full scale model 2000mm 100x200x2000
Boundary conditions For beams	Simply supported beams
Loading pattern For beam	Two point loading

SOLUTION PROCEDURE

These are no. of steps in the solutions procedure using finite element packages require the user to go through steps in one form or another.

Discretization of structure

1. Specifying geometry

First the geometry of the structure to be analysis is defined. This can be done either by entering the geometric information in the first element package through the key board or mouse, or by importing the model from a solid modular like structural engineer.

2. Specifying element and material properties

Next the materials properties are defined. in an elastic analysis of an isotropic solid these consists of modules of elasticity of concrete, young's modulus of steel and poison's ratio of steel and concrete.

3. Meshing

The structure is meshed into small elements. this involve defining the types of elements into which the structure will be broken as well as specifying how the structure will be subdivided into elements(how it will be mashed). This subdivided into elements into elements can either be input by the user or with some finite element programs by the user or with some finite element programs can be chosen automatically by the computer based on the geometry of the structure.

4. Application of the external load and boundary condition

Apply boundary conditions and external loads next the, boundary conditions, (e.g., location of supports) and the external nodes are specified. The support conditions are specified on key points and the loads in nodes.

5. Generate a solution

Then the solution is generated based on the previous input parameters.

6. Post processing:

Based on the initial conditions after a solution is processed, data is returned viewed in a variety of graphs and displays.

7. Refine the mesh:

Finite element methods are approximate methods and in general the accuracy of the approximation increases with the number of elements used. the number of elements needed for an accurate model depends on the problem and the specific results to be extracted from it. thus in order to judge the accuracy of result from a single finite element run, one needed to increase the number of elements in the object and see if the results changes.

8. Interpreting results

The most critical step in the entire analysis because it requires that modular use fundamental knowledge of mechanics to interpret and understand the output of the model. This is the critical for applying concrete result to solve real engineering problems and in identifying when modeling mistakes have been made (can easily occur).

The eight steps mentioned above have to be carried out then only meaningful information can be obtained regardless of size and complexity of the problem to be solved. However, the specific commands and procedures that must be listed for each of the steps will vary from one finite element package to another.

BEAMS

Beams are members that are subjected to bending. bending causes compressive as well as tensile stress in the same cross section depending upon the position of particles and the type of end supports beam is said to be statically determined, if its reaction components can be determined using equations of static equilibrium only. Commonly encountered statically determined beams are,

a. Cantilever beams

b. Simply supported beams and **c.** Over hanging beams.

The calibration of the finite element model using experimental load deflection behavior of HPRC beam. The use of ANSYS 10.0 to create the finite element model is discussed. All the necessary steps to create the calibrated model are explained in detail and the steps taken to generate the analytical load deflection responses of the members are discussed. The shear and flexure behavior of beam is modeled by changing the material properties with various percentages of replacements such as 5, 7.5, 10% of cement with two additives like Metakaolin and Silica Fume and Fly Ash constant replacement of 10%.

MESHING

To obtain good results from the solid 3D, concrete 65 elements, and the use of hexagonal mapped mesh is recommended. Therefore, mesh was setup such that hexahedral elements were created. The meshing is done with mesh tool menu which has global set containing the size of the element divisions which defines the size of the element which is formed. As the size of the elements decreases the elements are increased in number which the results are obtain are too accurate. As the elemental number increases the time consuming for solving a problem for the particular load increases thereby requires more memory space in the computer. The meshing of reinforcing bar is done is same as the procedure mentioned above from which the size of the element for bars should be reduced very low as the bar diameter being very less.

LOADS AND BOUNDARY CONDITIONS

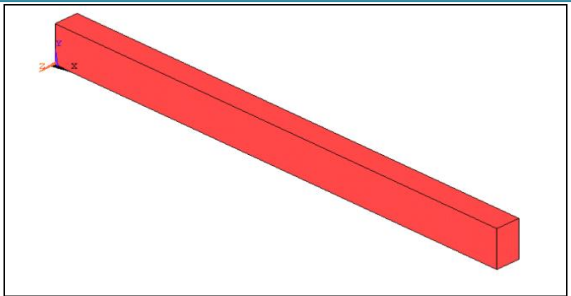
Displacement boundaries are needed to constraint the model to get a unique solution. To ensure that the model acts the same way as the experimental beam boundary conditions need to be applied at the points of symmetry, and where the supports and loading exist. The support was modeled as a hinged support on left side of the beam and the right side as the roller, maintaining the support specification is fixed with effecting length as 1500 mm and 500 mm being left as overhanging equally on both side of beam. The force p, applied at the top of beam as a two point loading with increments of 5kN at each step.

ANALYSIS PROCESS FOR THE FINITE ELEMENT MODEL

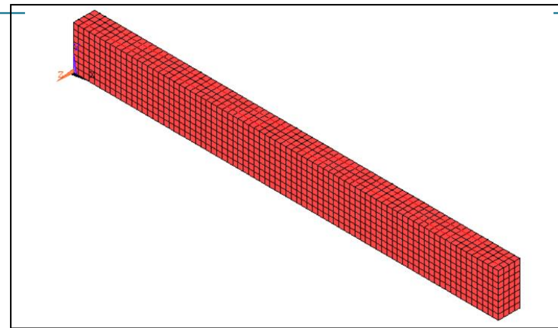
The finite element analysis of the model was set up to examine different behaviors. Here the analysis was done linearly to find deflection, stress, strain plots and to validate with experimental values.

POST PROCESSING

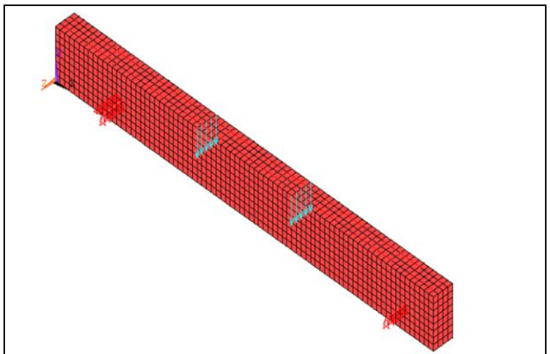
There are two types of post processing in ANSYS 10.0 program; general and time history. The later provides a step by step variation of any desired variable such as stress strain at various nodes or within any element in the model. The former provides and listing capabilities for the ultimate results (last time step) such as deformations, contour plots of stress and strains allow an automatic output of time history.



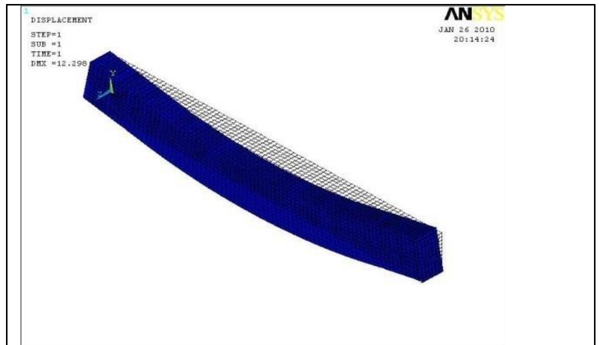
Model of Beam



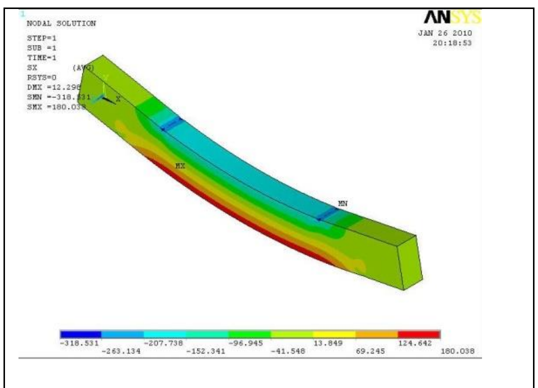
Meshing of Beam



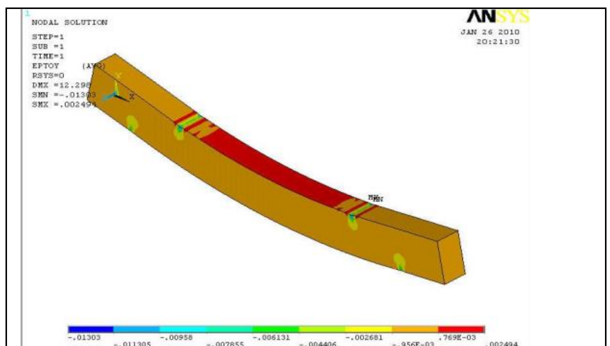
Loading of Beam



Deflection For M1



Stress For M1



Strain For M1

Table 3:MIX PROPORTIONS OF ADMIXTURES IN PERCENTAGE

MIX	% OF SILICA FUME	% OF GLASS FIBER	**Ultimate load - Experimental (kN)**	**Deflection as per experimental**	**Deflection as per ANSYS**
M1	**0**	**0**	**46**	**12.46**	**12.29**
M2	5	0	48	14.65	14.35
M3	7.5	0	54	17.75	17.50
M4	10	0	52	16.26	16.10
M5	5	0.2	52	17.89	17.89
M6	**7.5**	**0.2**	**58**	**21.77**	**21.34**
M7	10	0.2	54	19.08	19.08

CONCLUSIONS

In this project work to validate the experimental results obtained from the investigation of behavior of high performance reinforced concrete beams, column, wrapped beam& wrapped column element was performed with ANSYS 10.0. In general the specimens modeled with ANSYS 10.0 showed higher values of ultimate load and deflections when compared to the results of experimental work.

The following important conclusions are drawn from this study,

1. The ultimate loads obtained from the ANSYS modeling for the test specimens were higher than the corresponding specimens tested in laboratory in the range of 13% to 20%.
2. The FE modeling results have a good agreement with the experimentally tested specimen.
3. Auto refining is not possible in ANSYS 10.0 hence fine meshing should be done suitably.
4. Results obtained experimentally are conservative than the ANSYS 10.0 software. It is due to more stiffness in ANSYS 10.0.
5. The accuracy of the result depends up on meshing of FE model.

REFERENCES

1. **Bathe, K.J.,** "Finite element procedures", prentice-hall, Inc., upper saddle river, New Jersey, 1996.
2. **Chansawat, K.**, "Nonlinear Finite Element Analysis Of Reinforced Concrete Structure Strengthened With FRP Laminates," Personal Communications, Corvallis, Oregon, 2000.
3. **Chen, A.C.T. and Chen, W.F.,** "Constitutive relations for concrete," journal of the engineering mechanics division, ASCE, vol.101, no.EM4, pp.465-481, august 1975.
4. **Desayi,P. and Krishnan,S.,** "Equation for the stress-strain curve of concrete," journal of the American concrete institute,61,pp.345-350,march 1964
5. **Hope-Gill, M. C., and Johnson, R. P. (1976).** "Tests on three three-span continuous composite beam".Proc. Inst.Civv.Eng. 61,367-381.
6. **Jing Liu.Stephen J.Foster (2000)**, "A three dimensional Finite element model for confined concrete structures"- computers and structures Vol.77,pp 441-451.
7. **Kachlakev, D.I and McCurry, D., Jr**., "Simulated full scale testing of reinforced concrete beams strengthened with FRP copmposites": Experimental results and design model verification, Oregon Department of transportation, Salem, Oregon, June 2000.
8. **Mindess, S**. **and Young, J.F**., Concrete, prentice-hall, Inc., Englewood cliffs, New Jersey, 1981.