Finite Element Modelling of Geopolymer Concrete Beams Using Ansys

G.Jayarajan[1], Dr.S.Arivalagan[2]
Ph.D Research Scholar, Department of Civil Engineering, Dr.M.G.R Educational and Research Institute Maduravoyal, Chennai -600095, Tamilnadu, India
Professor and Dean, Department of Civil Engineering, Dr.M.G.R Educational and Research Institute University, Maduravoyal, Chennai-600095, India

***Abstract*- Geopolymer concrete is the concrete made without using any quantity of cement. Instead the waste material from the thermal power plant called Fly Ash is used as the binding material. This fly ash reacts chemically with alkaline solution like Sodium Hydroxide (NaOH) and Sodium Silicate (Na2SiO3) and forms a gel which binds the fine and coarse aggregates. Similarly another Artificial material called Quarry sand is also used as the fine aggregate and 10mm aggregate is also used as the coarse aggregate. Concrete beams of size 500mm x 100mm x 100 mm were prepared This paper focus the ANSYS software was utilized to simulate the behavior of Concrete beams.**

***Index terms*- ANSYS, Beam section, Geopolymer concrete, Meshing, FEM**

I.INTRODUCTION

Cement Concrete is the world‘s most versatile, durable, reliable construction material and the most used material next to water. Portland cement production is the second only to the automobile as the major generator of carbon dioxide. One ton of Cement production approximately liberates one ton of carbon di oxide to the atmosphere. Hence, it is inevitable to find an alternative material to the most expensive, most resource consuming cement concrete. Geopolymer concrete is the one, which is produced without using any quantity of cement. ANSYS Workbench capabilities include a unique and extensive materials and sections for concrete and steel structures.. A user- friendly beam and shell postprocessor included listing and plotting section geometry, reinforcements, beam stresses and strains inside the cross section. The skilled combination module, selects loads and coefficients for logic code combinations. Results embrace concomitance. The analysis is carried out in three stages such as. 1.Preprocessor, 2.Solution, 3.Post processor.

II. LITERETURE REVIEW

Sunilaa George et al (2008) suggested that river sand is widely used for concrete as fine aggregate. The increased cost of river sand and depletion in ground water table due to illegal sand mining leads to finding an alternative for fine aggregate without compromising the strength. The authors proposed a mix design for mix M30 grade of concrete using quarry dust and manufactured sand by replacing the river sand. Four mix proportions were made to test the effect of inclusion of quarry dust and manufactured sand in concrete and the results were compared with the control specimens. It was found that the strength of the concrete is enhanced in both the types of replacements.

Gui Feng Liu et al (2011) reported that experimental studies on C30 concrete with manufactured-sand were carried out under conditions of freeze-thaw cycle, which is based on the testing of raw material performance and concrete mix ratio; The authors reported the comparative studies on the changing laws of the mass, strength and the relative dynamic Elastic Modulus of concrete were developed in three cases which were freeze-thaw cycle, freeze-thaw cycle and acid corrosion, and freeze-thaw cycle and alkali corrosion. The test results showed that the mass, strength and the relative dynamic Elastic Modulus of concrete with manufactured-sand decreased evidently with the increasing of times of freeze-thaw cycle.

Mahendra Chitlange and Prakash S. Pajgade (2010) studied the steel fiber reinforced concrete with

artificial sand as fine aggregate. Three matrices with compressive strength of 20, 30 and 40 MPa were designed and reinforced with crimpled steel fibers at dosage rate of volume fraction 0, 0.5, 1.0, 1.5 and 2.0 percent. The specimens were prepared, cured and tested for compressive strength, flexural strength and split tensile strength. The strength of steel fiber reinforced natural sand concrete (SFRNSC) and steel fiber reinforced artificial sand concrete (SFRASC) have been compared with the test data from the present study. The promotional use of artificial sand will conserve the natural resources for the sustainable development of the concrete in construction industry. Fernández-Jim nez and Palomo (2003) studied the suitability of various types of fly ash to be Geopolymer source material. These researchers proved that to produce optimal binding properties, the low-calcium fly ash should have the percentage of unburned material (LOI) less than 5%, Fe2O3 content should not exceed 10%, and low CaO content, the content of reactive silica should be between 40-50%, and 80-90% of particles should be smaller than 45 m.

Earlier Balaguru et al (1999) suggested Geopolymer coating to protect the transportation infrastructures as well as the use of Geopolymer composites to strengthen concrete structures. They reported that to strengthen reinforced concrete beams, Geopolymer composites have been successfully applied. In terms of fire resistance, durability under ultra violet light, the performance

III.DESCRIPTION ABOUT THE SOFTWARE

ANSYS is analysis software across a range of disciples including finite element analysis, structural analysis, computational fluid dynamics, explicit and implicit methods and heat transfer. The ANSYS MECHANICAL is a finite element analysis tool for structural analysis including linear, non-linear and dynamic studies. This computer simulation provides finite elements to model behavior, and support material models and equation solvers for a wide range of mechanical design problems. ANSYS Mechanical also includes thermal analysis and coupled-physics capabilities involving acoustics, piezoelectric, thermal–structural and thermo- electric analysis. ANSYS workbench (Mechanical) is more graphics focused and geometry focused. There is less direct connection to the FEM. Workbench provides easy to learn, easy to use environment. The ANSYS Workbench environment is an intuitive up-front finite element analysis tool that is used in conjunction with CAD systems and/or design modeler.

IV ANALYTICAL STUDY OF BEAMS

The above mentioned steps were explained in a detailed manner below with the data obtained from the experimental values and the same is validated with that of the analytical part with comparison of effective cover of 20mm and 25mm normally experimental analysis is done to study the individual component response and the strength of concrete under various loading conditions. This method shows the actual response of the structure but it is expensive and time consuming. To minimize that difficulty, the finite element analysis is an effective tool for the evaluation of structures and to find the response of structures by giving almost accurate results under various loading conditions. By taking the advantage of the symmetry of the beam and loading, one quarter of the full beam was used for finite element modeling. This approach reduces computational time and computer disk space requirements significantly.

V. NUMERICAL INVESTIGATION

Finite element analysis package ANSYS 14 was used for the modeling and analysis. A non-linear analysis was performed and the materials are assumed to behave as an isotropic hardening material. The element type used to model the test specimens is SHELL 63. It is a 4-noded 3 dimensional quadratic shell element. This element has six degrees of freedom at each node. Finite element mesh of size 2x2mm was implied and used in all the simulations. The friction or contact between connected leg of the specimen and the gusset plate was ignored. ANSYS workbench capabilities include a unique and extensive materials and sections liberary for steel structures. In addition, the user could introduce ant shape or material into the corresponding ANSYS workbench liberaties.

ANSYS is a general purpose software, used to simulate interactions of all the disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers. ANSYS can carry out

advanced engineering analysis quickly, safely and practically by its variety of contact algorithms, time based loading features and nonlinear material models. SOLID187 element, a higher order 3-D, 10-node element was used to model the concrete material. SOLID187 has a quadratic displacement behavior and is well suited to modeling irregular meshes. The element was defined by 10 nodes, with three degrees of freedom at each node: translations in the nodal x, y, and z directions. The SOLID 187 element is shown in the Figure 2

Object Name	*Global Coordinate System*
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. mm
Origin Y	0. mm
Origin Z	0. mm
Directional Vectors	
X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

Fig 1 Coordinate system

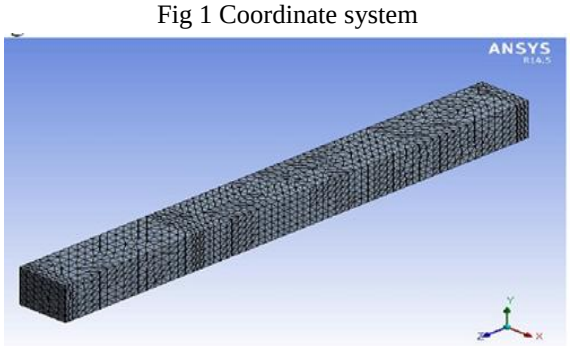

Fig 2 Modeling of structures

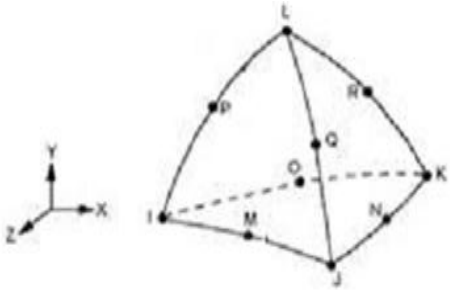

Fig 3 Solid 187 element

VI FINITE ELEMENT ANALYSIS

In this section, finite element modelling of the experimental Angle specimens is described. FEA as applied in engineering is a computational tool for performing engineering analysis. The FEA is performed using 3D structural solid elements that are capable of representing large deformation geometric and material non-linearities. In the current study, each of the Angle specimens is analysed. SOLID 187 is used for the 3-D modelling
of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials. An elastoplastic von Mises yield criterion is adopted to represent the material non- linear effects.

VI MODELING OF STRUCTURES USING ANSYS

Modeling is one of the most important aspects for the FEM analysis. Accuracy in the modeling of element type and size, geometry, material properties, boundary conditions and loads are absolutely necessary for close numerical idealization of the actual member. Modeling the complex behavior of reinforced concrete, which is anisotropic and non homogeneous, is a difficult challenge in the finite element analysis of Civil Engineering structures. The angle sections were modeled using shell element. The numerical models were discredited with the reduced integration four-noded doubly curved shell element and it has five degrees of freedom per node.

VIII MESHING

To obtain good results from the solid 3D, concrete 65 elements, and the use of hexagonal mapped mesh is recommended. Therefore, mesh was set up 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 by means of which results are

obtained are too accurate. As the elemental number increases, the time consuming for solving a problem for the particular load increases thereby requiring more memory space in the computer. The meshing of reinforcing bar was done in the procedure mentioned above from which the size of the element for bars should be reduced very low because the bar diameter is very less.6.5 Analysis. Initially linear analysis was carried out. Having confirmed these results in the linear range, nonlinear analysis was performed.

IX STATIC STRUCTURAL ANALYSIS

A static structural analysis determines the displacements, stresses, strains and forces in structures or components caused by loads that do not induce substantial inertia and damping. The ANSYS 14 was used to perform a static structural load.The static material property of the tension tests and the measured cross- sectional dimensions were used to model the angle specimens .Structural analyzes can be performed using linear and non- linear. Linear models use simple parameters and suppose the material is not deformed plastically. Nonlinear models involve stressing the material beyond its elastic capacity.A non- linear analysis was carried out and the materials are assumed to be an isotropic material for hardening. In this chapter, non linear static structural analyzes are performed in simulation.

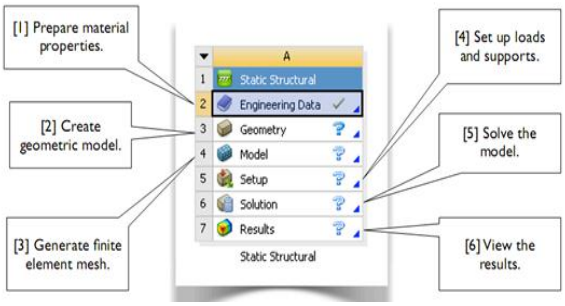

Fig 4 Static Structural Simulations

An easy way to comply with the conference paper formatting requirements is to use this document as a template and simply type your text into it.

X ANSYS WORKBENCH

ANSYS (Workbench) 14 makes it easy to evaluate the quality of the elements during the development of meshing controls and the results of post- processing. Well- formed elements deliver superior results and help reduce element shape errors during large- scale displacement analysis, such as when using hyper elastic materials with a significant strain. This article briefly reviews the features in the Workbench Mechanical Outline for displaying the quality of the elements(1) in color in the mesh branch,(2) using element quality bar charts and(3) plotting the quality of the elements in post- processing after element shapes have been distorted by strain in a model. The ANSYS workbench capabilities include unique and extensive steel structure materials and sections. The user could also add the various shapes or materials to the corresponding ANSYS workbench library.

A user- friendly beam and shell post processor lists the geometry and stresses of the plotting section and the strain inside the cross section. The skilled combination module selects loads and coefficients for element and global logic code combinations as well as the worst shell and solid element load arrangement.

XI ELEMENT TYPES USED FOR MODELING

The following were the element types used in the simulation.

LINK 8 for steel

LINK 8 is a 3D spar element. It is a uniaxial tension-compression element with three degrees of freedom at each node. Plasticity, creep, swelling and stress stiffening capabilities were included. A Link8 element was used to model the links. Two nodes were required for this element. Each node had three degrees of freedom, – translations in the nodal x, y, and z directions. The element was also capable of plastic deformation. The geometry and node locations for this element type are shown in Figure.5

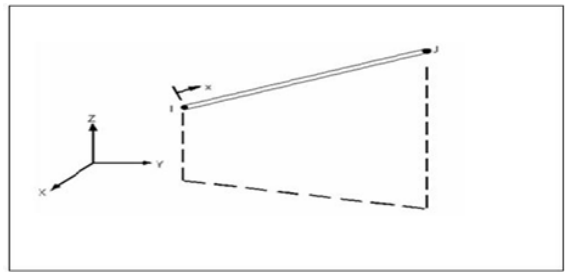

Fig 5 Link 8 element type

XII.CONCLUSIONS

There are two types of post processing in ANSYS 14.0 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 lists capabilities for the ultimate results (last time step) such as deformations, contour plots of stress and strains, and allows an automatic output of history.

REFERENCES

[1] Karthik M. P, Arul Gnanapragasam A., Sree Vidya V., Ragul K.,Robert Ravi S.,Rex J."Analytical Study on Fibre Reinforced Geopolymer Concrete".

[2] Darmansyah Tjitradi, Eliatun Eliatun, Syahril Taufik"3D ANSYS Numerical Modeling of Reinforced Concrete Beam Behavior under Different Collapsed Mechanisms".

[3] Shaishav R. Viradiya, Tarak P. Vora "Comparative Study of Experimental And Analytical Results of Frp Strengthened Beams In Flexure"

[4] B. Sri Kalyan, Dr.J. Guru Jawahar, C. Sreenivasulu "Analytical Study on Fly Ash And GGBS Blended Reinforced Geopolymer Concrete Beams By Using Ansys"

[5] Experimental Validation of Geopolymer Concrete Beams Using Finite Element Analysis

[6] A.M. Malek, H. Saadatmanesh "Analytical Study of Reinforced Concrete Beams Strengthened With Web-Bonded Fibre Reinforced Plates or Fabrics"

[7] C.C. Spyrakos, I.G. Raftoyiannis , L. Credali And J. Ussia "Experimental And Analytical Study on Reinforced Concrete Beams In Bending Strengthened With FRP"

[8] Malek, Amir Masoud "Analytical Study of Reinforced concrete Beams Strengthened With Fiber Reinforced Plastic Plates (Fabrics)"