

STRUCTURAL BEHAVIOUR OF GEOPOLYMER SLAB PANELS

S.Veeraraghavan

PG Student, Dept of Civil Engineering
Thiagarajar College of Engineering
Madurai, Tamil Nadu

Mr.R.Sankaranarayanan

Assistant Professor, Dept of Civil Engineering
Thiagarajar College of Engineering
Madurai, Tamil Nadu

ABSTRACT- Geopolymer ferrocement slab panels are light weight, having reinforcement in both the transverse and longitudinal direction is considered to be a high characteristic material. Geopolymer ferrocement is composition of two different material in which fly ash is fully replaced by cement. Fly ash is a waste by-product from thermal power plants. Hence this material is not only effectively recycled but also a suitable replacement material for cement satisfying its mechanical properties. Due to its light weight characteristics it finds application in the construction of prefabricated structures. Hence this material needs more exploration to suit the industrial need. In this work an analytical study was performed virtually using a finite element software ABAQUS. A completed experimental work is considered and it is depicted in the aforementioned software. Initially the models of mortar slab, chicken mesh and reinforcement are created and their corresponding mechanical properties are assigned. All the individual elements are assembled together. A one-way slab of size 1000 X 200 X 30mm is created and it is allowed to act in a simply supported condition. A two point load is applied on the slab and its load- deformation characteristics are plotted. Also the maximum deflection for the applied load is obtained from the results. Finally the results are validated with that of the experimental and theoretical values.

Key Words: Geopolymer, Ferrocement, ABAQUS, Slab, wire mesh.

I. INTRODUCTION

Ferrocement is a debutant material of the reinforced concrete family. Yet Reinforced concrete plays a dominant role in many design aspects of any structure. Unlike reinforced concrete, ferrocement has very small diameter wires mesh closely spaced with the absence of coarse aggregate. It has a thickness in the range of 25 to 50mm. Generally ferrocement accommodates reinforcement in both the transverse and longitudinal direction to establish its high characteristics and performance. It is an environmental friendly, easy to fabricate in field possessing excellent mechanical properties such as good tensile strength, toughness, lightness, resistance to fire and resistance to cracking. Ferrocement has both metallic and non-metallic meshes as recommended by ACI. There are different types of meshes available in the market. They are woven mesh, chicken mesh and expandable mesh. Welded meshes are rectangular in shape welded at every interface.

The term 'Geopolymer' was coined by Davidovits in 1978 to represent a geological element into a link of chain like substance through a polymerization reaction with a help of alkaline solution. The main ingredients in fly ash is silicon(Si) and aluminium(Al) which when dissolved in the alkaline solution produces a hardened binder. Here the alkaline solutions used are mostly sodium or potassium based. Hardjito and Rangan studied the engineering properties of geopolymer through steam curing in order to

accelerate the polymerization reaction. Geopolymerisation is an exothermic reaction takes place in atmospheric pressure at a temperature of 100°C.

A. OBJECTIVE

- To study the behavior the geopolymer ferrocement panels analytically using finite element software ANSYS 16.4.
- To correlate the results obtained from the experimental values with that of the analytical values.

II. ANALYSIS OF GEOPOLYMER FERROCEMENT SLAB PANELS USING ABAQUS

ABAQUS-CAE is an interactive, graphical environment for Abaqus. It allows models to be created easily and quickly by producing or importing geometry of the structure to be analysed and decomposing the geometry into meshable region. Physical and material properties can be assigned together along with the loads and boundary conditions. ABAQUS-CAE contains very power options to mesh the geometry and to verify the resulting analysis model. Once the model is complete, it can submit, monitor and control the analysis model. Creating and analyzing of model has eight main module to obtain the expected results.

A. The Part Module

This module allows the user to create, edit and manage any required geometry using the tools available in the part. ABAQUS stores each part in the form of an ordered list of features. Part allows the user to create rigid, discrete rigid, analytical rigid or Eulerian which is defined in the current tools of it.

Table 1 Specification of elements of slab panel

| Description | Specification |
|---------------------------|-----------------------------------|
| Slab panel size | 1000 X 200 X 30mm |
| Diameter of Reinforcement | 10mm |
| Mesh size | 12mm spacing of hexagonal opening |

A solid element is chosen for the modelling of slab member. Initially the cross-sectional area is drawn and then extrude option is used to magnify it into the required slab. Similarly a wire member is incorporated for the creation of mesh and reinforcement part. The figure 3.1 shows the created part.

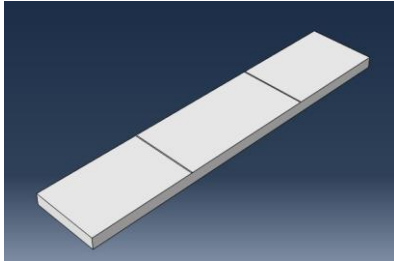


Fig 1-Slab Panels

B. The Property Module

A material definition specifies all property data relevant to a material. Each material created is assigned its own name and is independent of particular section. ABAQUS-CAE assigns the properties of a material to a region of part when a section is assigned referring to that material to the region. In this step the mechanical properties such as Young's modulus and Poisson's ratio are assigned to the corresponding elements from the experimental work. Also the general properties such as density are also assigned apart from the above mentioned properties. The table shows the properties of all individual elements.

Table 2 Material properties of elements of slab panels.

| Elements | Young's Modulus in N/mm ² | Poisson's ratio |
|---------------|--------------------------------------|-----------------|
| Slab panel | 9100 | 0.2 |
| Reinforcement | 2 X 10 ⁵ | 0.3 |
| Mesh | 2 X 10 ⁵ | 0.3 |

C. The Assembly Module

This step involves in the assembly of all individual parts in their respective position in the slab panel. Here the mesh and the reinforcement are assembled inside the slab with a clear cover maintained on all sides of the slab. The figure 3. shows the assembled part of the entire element.

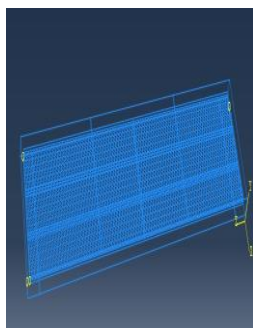


Fig.2 Assembled Slab Parts

D. The Step Module

The step sequence provides a convenient way to capture changes in loading and boundary conditions of the model, changes in the way parts of the model interact with each other, the removal or addition of parts, and any other changes that occur that may occur in the model during the course of analysis. In addition step allows to change the

analysis procedure, the data output and various control. Here the static approach is performed over the slab.

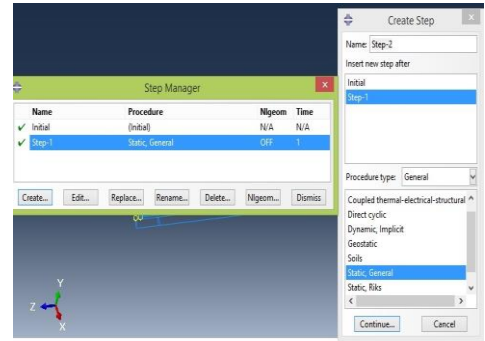


Fig.3 Step module

E. The Interaction Module

ABAQUS-CAE does not recognize mechanical contact between part instances or regions of assembly unless that contact is specified in the interaction module. The set and surface tools allows the user to define and name regions of the model for which the interactions and boundary conditions are to be applied. Interaction module can be used to define the following types of interaction. Here tie constraints and embedded region constraints are the two major ones adopted to ascertain the interaction between the parts. The table briefly explains about the interaction part.

Table 3 Interaction properties

| MEMBERS | CONNECTIONS |
|------------------------|-------------|
| Mesh and reinforcement | Tie |
| Slab and mesh | Embedded |

The slab component which accommodates both reinforcement and mesh acts as a master element and the mesh acts as slave element. The embedded region constraint allows embedding a region of model within a host region of model or within the whole model.

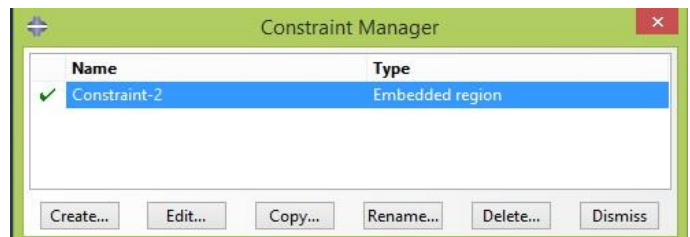


Fig.4-Constraint manager

F. The Load And Boundary Condition Module

The load module allows the user to give different types of loads to act on the slab panel. Here the two point load is applied on the top surface of the slab with a specified distance from the support. The supporting condition is simply supported and hence Displacement and rotation option is used to depict this condition. The following figure shows the loading and boundary conditions.

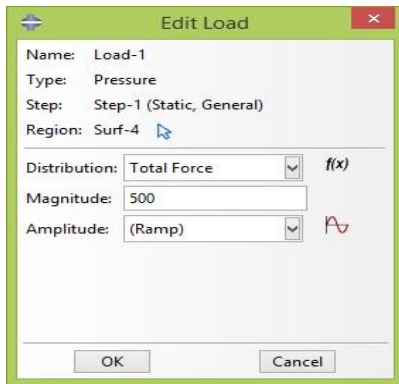


Fig.5 Load Manager

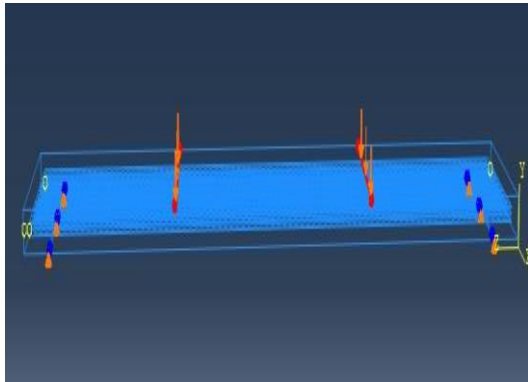


Fig.6 Loading Diagram

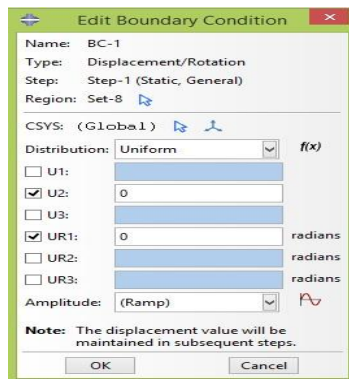


Fig.7 Boundary Condition

G. The Meshing Module

The mesh module allows to generate mesh on parts and assemblies created within ABAQUS-CAE. The mesh module provides the following features.

1. Tools for prescribing mesh density at local and global level.
2. A tool for verifying mesh quality.
3. A tool for refining mesh and for improving mesh quality.
4. A tool for saving meshed assembly or selected part instances as mesh part.

Meshing can be done both dependent and independent instance manner.

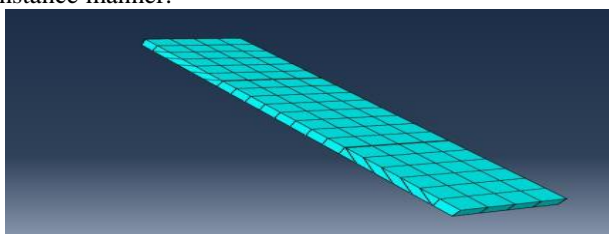


Fig.8 Meshing of Steel mesh

H. The Job Module

After the completion of all modules, the job module allows to create a job, to submit it for analysis

and to monitor its progress. This is done using a job manager. It does the following activities.

1. Write an input file for a model based job without submitting it for analysis.
2. Perform a data check on model.
3. Submit a job for analysis.
4. Continue an analysis to completion after performing a data check.
5. Monitor the analysis as it progresses.
6. View the results from the job.
7. Kill a job that is running.

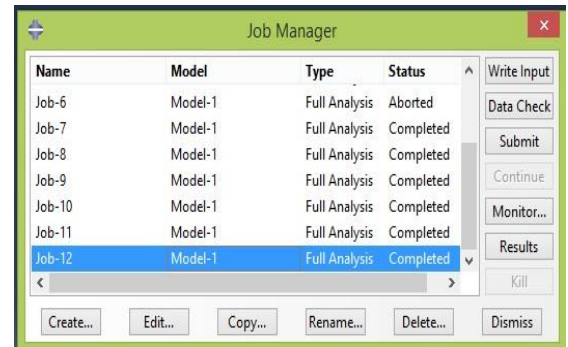


Fig.9 Job Manager

The deformed shape of the slab is finally obtained from the job part. The corresponding deflection is seen from the visualization part. The various colour coding are represented for the maximum deflection to the minimum deflection in the slab panels.

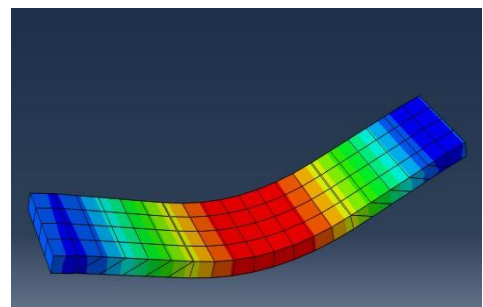


Fig.10 Deformed shape of slab panel

III. RESULTS AND DISCUSSIONS

After successful running of the analysis in ABAQUS, the load deflection curve can be obtained by varying the loads and retrieving the corresponding deflection. The maximum deflection obtained for the ultimate load of 6.25kN is 10.38mm. Here the slab is tested with two layers of mesh. The load and their corresponding deflection are tabulated below.

Table 4 Applied Load and obtained deflection for Double layer of mesh

| LOAD in kN | DEFLECTION in mm |
|------------|------------------|
| 0.5 | 0.83 |
| 1 | 1.661 |
| 3.8 | 6.31 |
| 4.3 | 7.14 |
| 4.78 | 7.93 |
| 5.23 | 8.68 |
| 5.7 | 9.465 |
| 6.25 | 10.38 |

A graph is plotted between load in the Y-axis and deflection in the X-axis. The line obtained is neither steeper nor flatter. Hence it can store minimum amount of strain energy and this indirectly indicates the low stiffness value. The graph is shown below. The maximum deflection from theoretical value is 6.45 mm for the same amount of load.

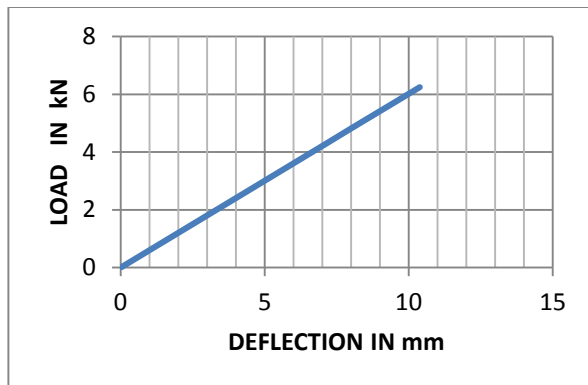


Fig.11 Load –Deflection Graph

IV. CONCLUSION

The geopolymer ferrocement slab panels with their properties and dimensions assigned to them are analysed successfully in the finite element software ABAQUS 16.4. The following conclusions are arrived from the analytical work. They are as follows,

- The maximum deflection is found to be 10.38mm for a specified load of 6.25 kN for a double mesh layers.
- From theoretical formula, the maximum deflection is obtained as 6.45mm which does not vary much with the analytical value.
- This may be due to the assumptions made in the analytical part that, slab panels is considered as a homogeneous and isotropic, whereas in practical the slab is a heterogeneous material.

Acknowledgment

I sincerely express my profound to Dr. B. SIVAGURUNATHAN, Associate Professor and Head, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai for providing all necessary facilities for this project work.

I would like to thank my guide Mr. R. Sankaranarayanan, Assistant Professor, Department of Civil Engineering, Thiagarajar College of Engineering, Madurai for his cordial and friendly support during this work. I thank him for sharing ideas with me which played a vital role in carrying out this investigation.

I thank all the faculties in Civil Engineering Department, Thiagarajar College of Engineering, Madurai for their valuable advices. I also express my sincere thanks to all who have directly and indirectly assisted me in completing the work.

REFERENCES

1. G.B.Maranan (2015) "Evaluation Of The Flexural Strength and Serviceability Of Geopolymer Concrete Reinforced With GFRP"- *Engineering Structures*, Pg.529-5441.

2. T.Kiran, Sadath Ali Khan Azath,(2015)"Impact Test on Geopolymer Concrete Slab", *International Journal of Engineering and Technology*,Vol.4,Issue.12,PP-110-116.
3. Smiriti Raj,V.Ramesh Kumar, B.H.Bharath Kumar, Smitha Gopinath, Nagesh R. Iyer(2015)"Flexural Studeies on Basalt Fibre Reinforced Composite Sandwich Panel with Profile Sheet as Core" *Construction and Building Materials*, Vol.82,PP.391-400.
4. S.Nagan, S.Dharmar, C.Manikandan (2015)"Experimental Study On Flexural Behaviour Of Fiber Reinforced Geopolymer Ferrocement Folded Panel", *International Journal Of Advanced Research In Science And Engineering* Vol.No.4,Special issue(01)Pg:1761-1768.
5. M.Mastali, M.B.Valentene,(2015)"Development of Innovative Hybrid Sandwich Panel:Experimental Results" *Composite Structures*, pp-476-498.
6. V. Sreevidya, R. Anuradha, R.Venkatasubramani and S. Yuvaraj, (2014) "Flexural Beaviour of Geopolymer Ferrocement Element", *Asian Journal of Civil Engineering* vol. 15, NO. 4 pg: 563-574.
7. R.Vicent prabakar, S.Dharmar.(2014), "Experimental Investigation on Flexural Behaviour of Folded Ferrocement Panels", *International Journal of Innovative Research in Science, Engineering and Technology*, vol3.
8. N.Ganaesn, Ruby Abraham,(2014)"Stress-Strain Behaviour of Confined Geopolymer Concrete" *Construction and Building Materials*,pp-326-331.
9. Burak Isikdag,(2013) "Characterisation Lightweight Ferrocement Panels Containing Expanded Perlite Based Mortar", *Construction and Building Material*, pp-15-23.
10. Nagesh M. Kulkarni (2013), "Analysis and Design of ferrocement Panels an Experimental Study", *International Journal of Engineering Science and Technology*, Vol. 1, No.5.
11. Nagan Soundarapandian and Mohana Rajendran (2013) "An Experimental Investigation on the Flexural Behaviour of Geopolymer Ferrocement Slabs"- *Journal of Engineering and Technology*, Pg.97-104.
12. V. Sreevidya, R. Anuradha, D. Dinakar, Dr. R. Venkatasubramani (2012) "Acid Resistance of Flyash Based Geopolymer Mortar under Ambient Curing And Heat curing" *International Journal of Engineering Science and Technology*, Vol. 4, No.02 pp 681-684.
13. Davidovits, J. (2008). Geopolymer chemistry and applications. Institute Geopolymer, Morrisville, pp. 585.
14. Ramakrishna, G. & Sundararajan, T. (2005), "Impact strength of a few natural fibre reinforced cement mortar slabs: a comparative study", *Cement & Concrete Composites*, Vol. 27, pp. 547-553.
15. ACI Committee 549, state-of-the-art report on ferrocement. Reported by ACI Committee 549, 1988.
16. Shuxing Wang, A.E.Namman, Victor C.Li(2004),"Bending Response of Hybrid Ferrocement Plates with Meshes and Fibres", *Jouranal of Ferrocement*, Vol.34.No.1,pp.275-289.