International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 11, November-2014

Finite Element Analysis of Composite Precast Roof Panel under Static Flexure

S. Nalini
Assistant Professor,
Department Of Civil Engineering,
Vel Tech Dr.RR Dr.SR Technical University,
Avadi,

E. Ramya
Assistant Professor,
Department Of Civil Engineering,
Vel Tech Dr.RR Dr.SR Technical University,
Avadi,

R. M. Saravanakumar
Assistant Professor,
Department Of Civil Engineering,
Vel Tech Dr.RR Dr.SR Technical University,
Avadi,

B. Karthik Hari
Assistant Professor,
Department Of Civil Engineering,
Vel Tech Dr.RR Dr.SR Technical University,
Avadi,

***Abstract*- Prefabricated composite roof panels offer a variety of possibilities to be used in many locations where economy, ease of construction and speed are of prime importance. High strength to weight ratio, reduced weight and thereby attraction of lesser seismic forces and good thermal insulation are some of the important characteristics of the panels. Numerical study is essential to evaluate the performance of the innovative composite panels.Estabilishing a FEM analysis will be useful to have a better understanding of the performance of composite lightweight large panel roof panel. A numerical study to develop composite lightweight panels for use as roof element in multistoried building was taken up. The composite panel is three layered, with two thin structural ferro cement outer layers and inner layer is made of Expanded Polystyrene (EPS).The inner EPS is 80 mm thick whereas the outer layers are of 25 mm thick. Reinforced concrete ribs are also provided along the periphery of the panels. The length of the panel is 2.8 m long, 1.2 m wide and total thickness of the panel is 0.13 m.**

This paper deals with the analysis of individual panel under flexure load using the Finite Element Analysis (FEA) software ANSYS. A three dimensional Finite Element Model is developed to Stimulate the static flexure behavior. The load- deflection response of the composite panel under different flexural loading conditions was simulated.

Element Analysis (FEA) software ANSYS. A three dimensional Finite Element Model is developed to stimulate the static flexure behavior. The load- deflection response of the composite panel under different flexural loading conditions was simulated.

In this paper, with the given linear material properties displacements, stresses, strains captured are discussed under static conditions.

Keywords: ***Finite Element Analysis (FEA), Ferro cement, Expanded Polystyrene (EPS), Flexure***

IJERT

I. INTRODUCTION:

Prefabrication is the practice of manufacturing components of a structure in a factory or other manufacturing site, transporting the complete assemblies or sub-assemblies and assembling on the construction site where the structure is to be located. The method controls construction costs by economizing on time, wages and materials besides assuring high quality Prefabricated units may include wall panels, floor panels, columns, beams, slabs, piles, footings, door frames, stairs, roof trusses, room-sized components, and even entire buildings. This type of construction requires a restricting of entire conventional construction process to enable interaction between design phase and production planning in order to improve and speed up construction there is exists a close relationship between design, construction, detailing execution and manufacturing of components. A typical insulated sandwich panel is shown in fig**.** 1

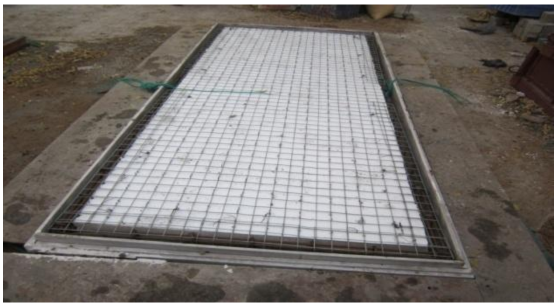

Fig . 1 Lightweight Sandwich Panel

ANSYS is used in the structural analysis. ANSYS is a general purpose FEA package for pre-processing, solution and post-processing of linear or non-linear, structural and thermal model. The flexural behavior of a roof panel is evaluated using ANSYS. As an initial step, a finite element analysis requires meshing of the model. In other words, the model is divided into a number of small elements, and after loading, stress and strain are calculated at integration points of these small elements. An important step in finite element modeling is the selection of the mesh density. A convergence of results is obtained when an adequate number of elements is used in a model. With FEM, the accuracy of the results depends on the selection of suitable

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

elements with the appropriate material characteristic. The elastic properties, poisons ratio and density of materials were used to stimulate the linear behavior of panel under static load. The size of the insulated sandwich roof panel is 2.8 m x 1.2 m x 0.13 m.The finite element model of the panel using ANSYS is shown in fig.2

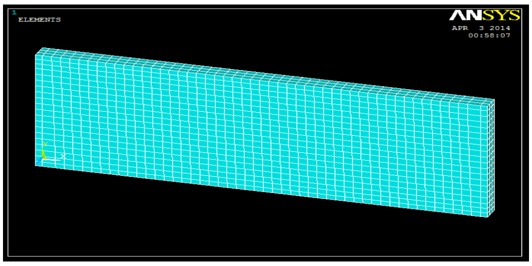

Fig 2 Sandwich Panel Using Finite Element Model (Ansys)

II. SIZE AND REINFORCEMENT DETAILS OF THE PANEL:

The composite lightweight panel consists of 25 mm thick outer layer and using M50 grade of self compacting concrete with square welded wire mesh of size 50 mm x50 mm x2.5 mm at the centre. The two welded mesh are connected together with shear connectors of 2.5 mm thickness. The EPS is 80 mm thick. The size of the main reinforcement is 80 mm x 40 mm, in this panel 8 mm dia rebars and 6 mm dia stirrups are used.

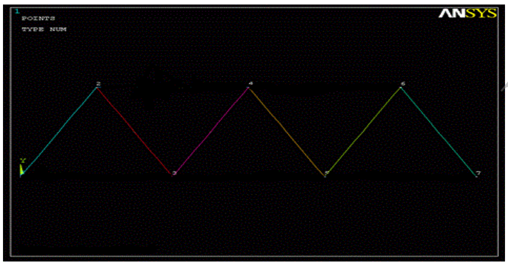

Fig.3 Shear Connectors

III. MATERIAL PROPERTIES USED FOR MODELING:

ANSYS requires input data for material properties are as follows

Table.1 Material Properties

Sl N	Material	Material model	Youngs modulus of Elasticity (MPa)	Poisson's Ratio
1	Concrete	Solid65	35355	0.18
2	Expanded Polystyrene	Solid65	10	0.12
3	Reinforcement bars, stirrups	Link8	2E5	0.3

IV. ELEMENT TYPES:

A. Link8 3-D Spar:

LINK8 is a spar which may be used in a variety of engineering applications. Depending upon the application, the element may be thought of as a truss element, a cable element, a link element, a spring element, etc. The three-dimensional spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, swelling, stress stiffening, and large deflection capabilities are included.

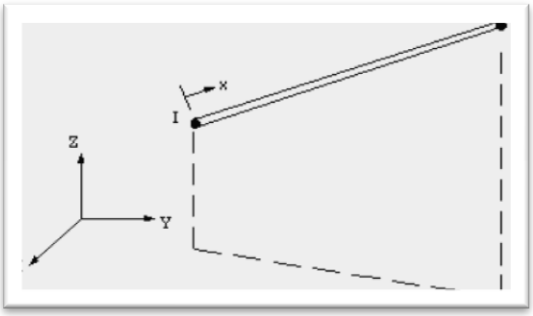

Fig.4 Link8

B. Solid65 3-d reinforced concrete solid:

SOLID65 is used for the three-dimensional modeling of solids with or without reinforcing bars (rebars). The solid is capable of cracking in tension and crushing in compression. In concrete applications, for example, the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions.

IJERT

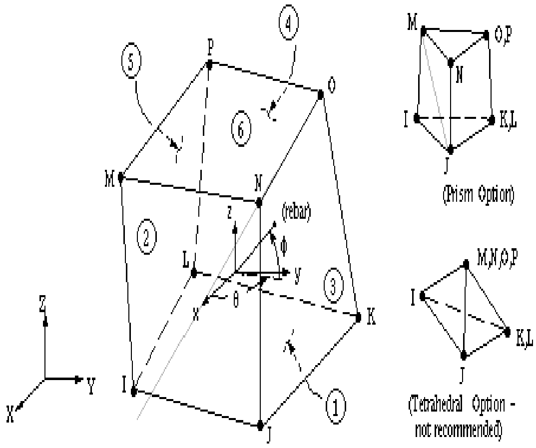

Fig. 5 solid

V. NUMERICAL MODELING AND FINITE ELEMENT ANALYSIS:

In numerical analysis, the roof panel is exhibited and analyzed with reinforcement. The objective of this paper is to understand the mechanical behavior of the roof under flexural. Specimens were modeled with linear finite element models. The properties of the sandwich structures

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

differ according to its material models of the structures, therefore characteristics of the sandwich panel are needed to be considered. The evaluation is done interactively using the visualization module of ANSYS. The support conditions are given in the end of the panel and loading is applied at the distance of 700mm from the ends. The linear solution is done and the panel solution is obtained both for its nodes and elements. Reinforced section with LINK8, Mesh with reinforcement section and supporting conditions are shown in fig.6, 7, 8

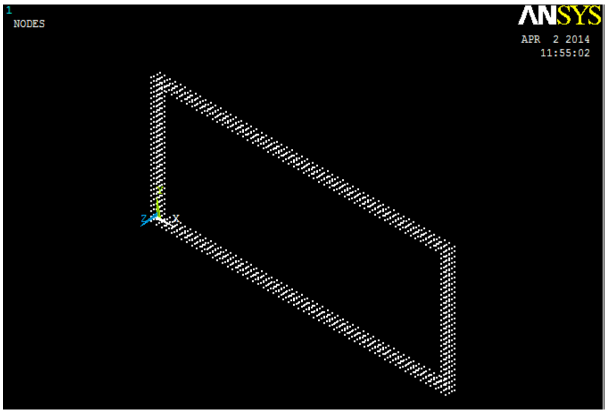

Fig .6 Reinforced Section

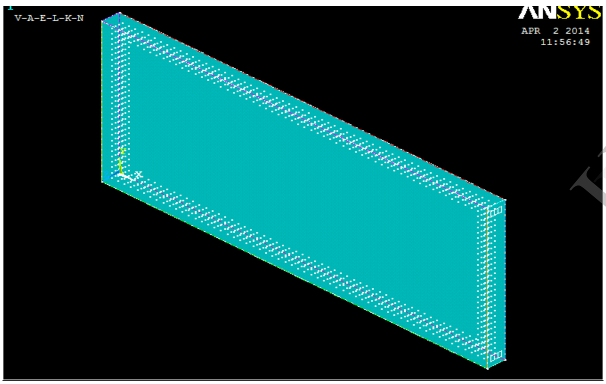

Fig.7 mesh with reinforced section

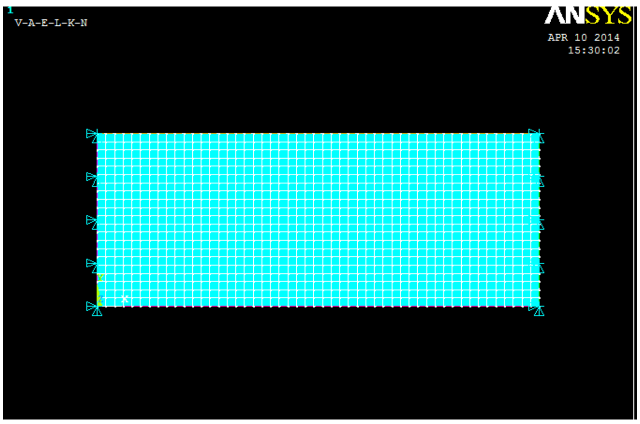

Fig 8 support conditions

VI. ANALYSIS RESULTS AND DISCUSSIONS:

Flexural load is applied on the panel by increasing the load from 0, 10, 20, 30, 40, 50, 60, 70, and 80 KN. The results are obtained and it can be visually viewed in the form of I, II, III, principal stresses, von-misses stress, elastic strain intensity, deflection at x, y, z, xy, yz & xz, etc. The behavior of the panel under a load of 30 KN is shown in fig .9. The ultimate load carrying capacity of the panel is 80 KN. The stress intensity of the panel is also verified. The strain at both the ends of the reinforcement remains same at its opposite nodes. The vector mode deflections of the panel are shown in figure 10,11 The details of maximum and minimum stress and strain are shown in table 2 and the load deflection curve for the above analysis is shown in Fig .12

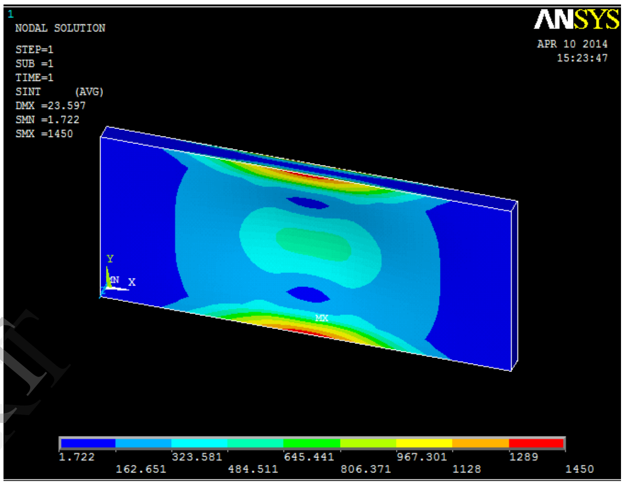

Fig .9 distribution stresses on loaded panel for 30kn

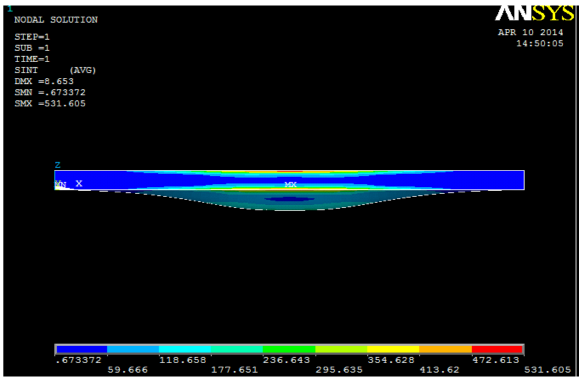

Fig. 10 vector node diagram for 30kn

IJERT

(This work is licensed under a Creative Commons Attribution 4.0 International License.)

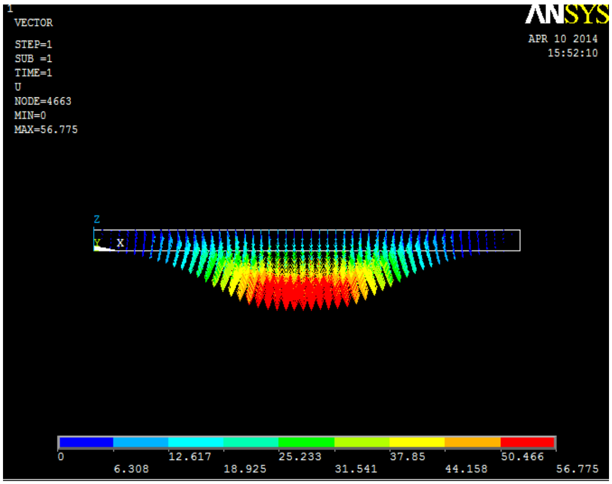

Fig .11 vector node deflection for 80 kn

Table.2 Maximum nodal stress and strain values for given loads in FEM

The maximum and minimum stress and strain values are observed in various nodes.

Sl. No	Load (KN)	Maximum stress	Maximum strain
1	10	568.78	0.15266E-01
		116.07	0.23425E-05
		78.846	-0.63073E-05
		531.61	0.17743E-01
		498.59	0.14102E-01
2	20	1137.6	0.30531E-01
		232.14	0.46850E-02
		157.69	-0.12615E-04
		1063.2	0.35485E-01
		997.17	0.28205E-01
3	30	1157.5	0.40572E-01
		243.15	0.5680E-02
		167.79	-0.22615E-04
		1073.3	0.52145E-01
		999.18	0.38205E-01
4	40	2103.0	0.56267E-01
		435.32	0.88100E-02
		303.32	-0.24570E-04
		1967.1	0.65654E-04
		1838.6	0.52005E-01
5	50	2568.0	0.68522E-01
		561.86	0.11570E-01
		384.19	-0.27646E-04
		2365.9	0.78963E-01
		2216.4	0.62690E-01
6	60	2935.6	0.78449E-01
		633.27	0.12972E-01
		441.59	-0.36350E-04
		2745.1	0.91620E-01
		2562.8	0.72487E-01
7	70	3287.3	0.87745E-01
		723.71	0.14878E-01
		506.86	-0.41359E-04
		3050.1	0.10180E-01
		2847.7	0.80548E-01
8	80	3735.6	0.99598E-01
		829.14	0.17116E-01
		580.37	-0.45525E-04
		3466.5	0.11570-01
		3236.0	0.91530-01

VII. GRAPH OBTAINED FROM ANALYSIS:

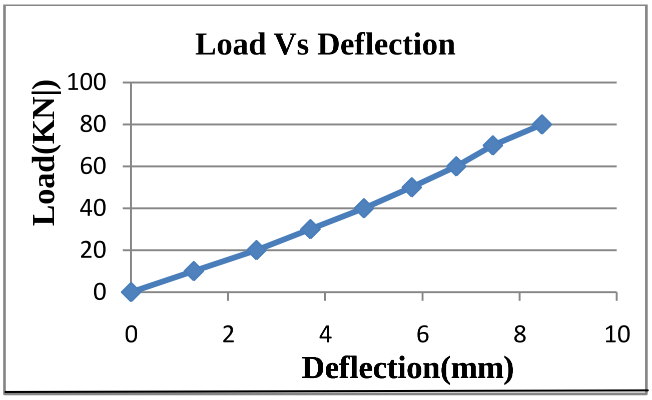

Fig.12 load vs deflection

VIII. CONCLUSIONS:

The following conclusion is drawn based on the Finite element analysis of innovative precast roof panel.

The maximum uniformly distributed load carrying capacity of the composite lightweight roof panel under flexure with the linear material properties is 80 KN.

REFERENCES:

1. Fabriziogara, Laura ragni, Davideroia, Luiginodezi, 'Experimental behavior of floor sandwich panels', Engineering structures 36 (2012), pg. 258-269.
2. Ezzat H Fahmy, yousry B shaheen, et al, "Ferro cement sandwich and cored panels for floor and wall construction'', Concrete & structure (2004), pg. 25-26
3. Nagesh M. Kulkarni, D.G.Gaidhankar, "Analysis and design of Ferro cement panels an experimental study'', International Journal of Inventive Engineering and science April (2013)
4. G.Carbonari, S.H.P.Cavalaro, M.M.Cansario, "Flexural behavior of lightweight sandwich panels composed by concrete and EPS'', Construction and Building Materials 35 (2012), pg. 792-799
5. R.Sriravindrarajah, et al, "Flexural creep of Ferro cement Polystyrene concrete composite", Second International Conference on Adavances in Composites December (1996), pg.18-20
6. A.Benayoune, A.A.AbdulSamad, et al, "Flexural behavior of precast concrete sandwich composite panel –Experimental and Theoretical Investigations", Construction and Building Material (2008), pg. 580-592
7. J.L. Wilson, A.J. Robinson, T. Balendra, "Performance of precast concrete load bearing panel structures in region of low to moderate seismicity", Engineering Structures 30 (2008),pg.1831-1841.
8. M.Giglio, A. Manes, A. Gilioli, "Investigation on sandwich core properties through an experimental numerical approach", Composites 43 (2012), pg.301-374

IJERT

(This work is licensed under a Creative Commons Attribution 4.0 International License.)