IOSR Journal of Engineering (IOSRJEN) **www.iosrjen.org**
ISSN (e): 2250-3021, ISSN (p): 2278-8719
Vol. 08, Issue 8 (August. 2018), ||V (VI) || PP 42-48

Analytical & Comparative Studies of EPS Core Member in Sandwich Slab Using ANSYS

Yash Atulkumar Jani[1],Dhananjay Patel[2]
[1]Post Graduate student, Department of civil engineering, R. K. University, Rajkot, Gujarat, India
[2] Assistant Professor, Department of civil engineering, R. K. University, Rajkot, Gujarat, India

Abstract: This paper presents to evaluate the study of sandwich slab using EPS as a core member. This paper presents a numerical analysis using ANSYS finite element program to simulate the reinforced concrete slab with sandwich slab which subjected to five point load. The finite element models represented by this work can be used to take out parametric study as well as comparative study for the sandwich slab and conventional slab example. The slab is a very important structural member in the building and slab is one of the largest member consuming concrete. When the load acting on the slab is large or clear span between column is more, if the slab thickness is increases. It leads to consume more concrete and steel, due to that self weight of the slab is also increased. To avoid this disadvantage various researches carried out and researchers suggest sandwich slab system to cut the self weight of the slab. EPS Sheet is reduce the weight of the sandwich slab so it is one of the most effective system, which can reduce the weight of concrete and steel. This report examines a two-way, reinforced concrete slab with core member of Polystyrene. The design process for sandwich slab is directly compare to solid slab concrete and to get the comparative results of deformation and stresses by using ANSYS.
Hence the study shows the deformation value & stress value for each of boundary conditions for the conventional slab & also sandwich slab using EPS.

Keyword: Expanded Polystyrene sheet(EPS), ANSYS, Finite Element Method, ICF Technology, Sandwich Slab.

Date of Submission: 16-08-2018 Date of acceptance: 03-09-2018

I. Introduction

The actual architecture more and more is based on modern aspects like environmental footprint and energy efficiency which necessarily requires lightweight constructions in all future buildings. New materials like polymers and carbon fibers which differs clearly in weight to classical materials like stones, clay, burnt brick, wood and glass asks for new competences and grasp of mathematics, engineering and science. There is a great potential in the field of lightweight building design. This minimizes the consumption of raw material for buildings.

Due to change in the environmental conditions temperature difference has increases a lot. This variation is very high on daily basis as well as on yearly basis. Nowadays, it became crucial to maintain ambient temperature within the structures specifically in the areas having extreme weather conditions. Maintaining this temperature control became very costly at present condition. This motivates to think about the system which provides minimal temperature loss and maintains thermal conditions for a longer period. Insulated concrete form (ICF) has become very famous for such application. System is developed such a way that this application is possible in slab and wall both which have maximum surface area in the structure. This option provides very less heat loss. This ICF system are rigid plastic forms which holds concrete in place during construction as well as curing and remains in place afterwards to serve for thermal insulation for structural elements

II. Material And Methods

2.1. Materials Properties

Concrete Idealization: The solid65 element models the nonlinear response of reinforced concrete. The behavior of the concrete material is based on a constitutive model for the tri axial behavior of concrete. Solid 65 is capable of plastic deformation, cracking in three orthogonal directions at each integration point. The cracking is modeled through an adjustment of the material properties that is done by changing the element stiffness matrices. If the concrete at an integration point fails in uniaxial, biaxial, or triaxial compression, the concrete is assumed crushed at that point. Crushing is defined as the complete deterioration of the structural integrity of the concrete. Table-1 lists concrete properties within Solid65 element prior to initial yield surface. The solid65 element is capable of cracking in tension and crushing in compression. The multi linear isotropic concrete model uses the von Misses failure to define the failure of concrete. The concrete element is similar to a 3-D structural solid but with the addition of special cracking and crushing capabilities. The most important aspect of this element is the treatment of nonlinear material properties. The concrete is capable of cracking (in three orthogonal directions), crushing, plastic deformation, and creep. The rebar are capable of tension and compression, but not shear. They are also capable of plastic deformation and creep. See Solid 65 in the Mechanical APDL Theory reference for more details about this element.

Steel Reinforcement: LINK8 is a spar which may be used in a variety of engineering applications and the 3-D spar element LINK8 available in the elements library of the ANSYS program was used. This element can be used to model trusses, sagging cables, links, springs, etc. The 3-D 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.

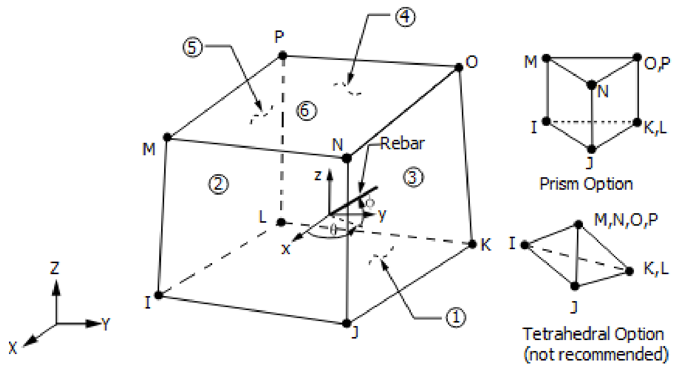


Figure 1: SOLID65 Geometry

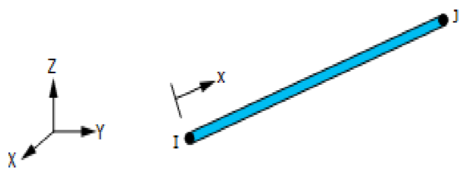


Figure 2: LINK8 Geometry

2.2. Modeling of Material Properties Parameters

Concrete Properties: Modeling an element for the behavior of concrete is a challenging task. Concrete is aquasi brittle material and has show different behavior in compression and tension. In compression, the stress-strain curve for concrete is linearly elastic up to about 30 percent of the maximum compressive strength. Above this point, the stress increases gradually up to the maximum compressive strength. After it reaches the maximum compressive strength, the curve descends into a softening region, and eventually crushing failure occurs at an ultimate strain. In tension, the stress-strain curve for concrete is approximately linearly elastic up to the maximum tensile strength. After this point, the concrete cracks and the strength decreases gradually to zero. The present study assumed that the concrete is a homogeneous and initially isotropic.

Es= 2716, 30MPa, Poisson's ratio γ= 0.2

Steel: Steel in RC slab is of grade Fy550. The steel for the finite element models has been assumed to be an elastic-perfectly plastic material and identical in tension and compression. Poisson's ratio of γ = 0.3 has been used for the steel in this study. Elastic modulus Es = 200,000 MPa and Poisson's ratio of 0.3 has been used for all the steel bars. Steel plates were provided at support locations and at the loading point in the finite element models to provide a more even stress distribution over the support and loading areas. Same elastic modulus equal to 200,000 MPa and Poisson's ratio of 0.3 was used for the plates. The steel plates were assumed to be linear elastic materials.

Table 1: Material Properties Parameters for Slab Specimens

Material Type	Element Type	Material Properties	
Concrete	Solid 65	Linear Isotropic	
		Es	27163 Mpa
		Poisson's ratio	0.2
		Multi linear Isotropic	
		Strain	Stress(MPa)
		0	0

		0.00036	10.02
		0.0006	15.28
		0.0009	21.55
		0.0013	27.59
		0.0019	32.31
		0.0024	33.40
		0.003	33.40
Reinforcement	Link 8	Linear Isotropic	
		Es	200000 MPa
		Poisson's ratio	0.3
		Yield stress	550 MPa
plate	Solid 185	Linear Isotropic	
		Es	2,00,000 Mpa
		Poisson's ratio	0.3

2.3. Test Specimens and Modeling

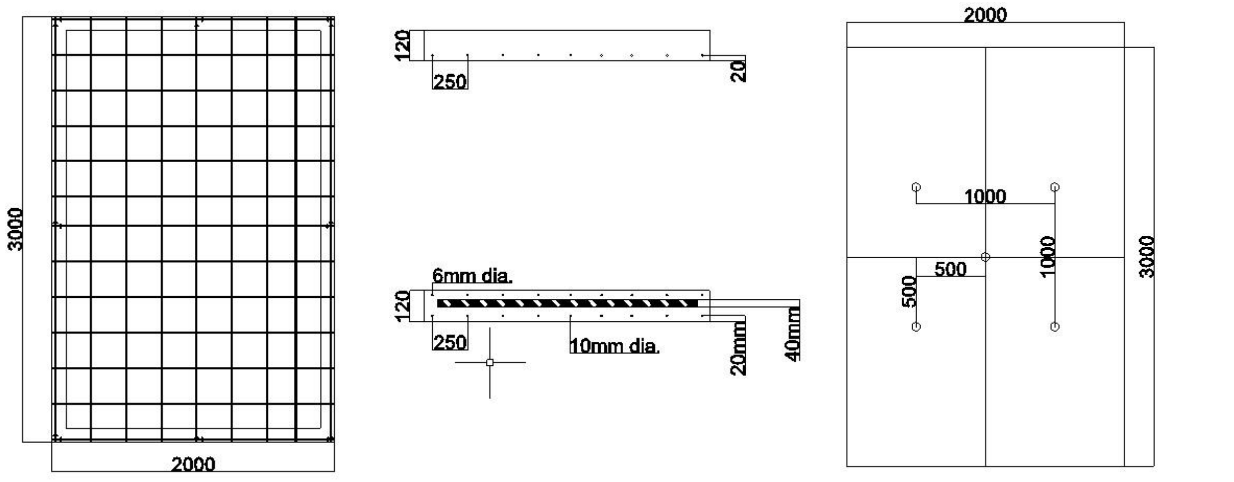
The geometry of the full size slab is 3000mm x 2000mm x 120mm. Slab is bounded with different boundary conditions from IS 456:2000 & shown in Table: 4. Applied factored load 10 kN, Force are applied at the top surface with five force point which are 500mm from centerline of the slab. The details of the RC slab are as shown in Table: 2 & 3.

Table 2: Slab Dimension

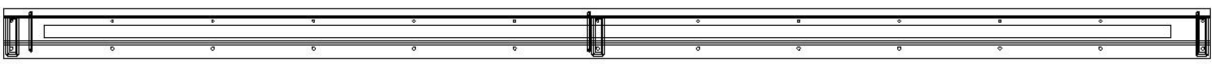
No.	Name	Length (mm)	Width (mm)	Thickness (mm)	Wight (Kg)
1	Conventional Slab	3000	2000	120	1670
2	EPS sandwich Slab				1411
3	EPS Sheet	2400	1400	40	20.9

Table 3: Slab Details

No.	Name		Thickness (mm)	Reinforcement (mm)	Clear Cover (mm)
1	Conventional Slab		120	10	20
2	EPS sandwich Slab	Upper		6	20
		Lower		10	20



Detail dimension & Loading dimension of slab



C/S of both slab

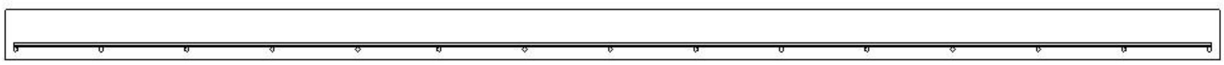


Figure 3 – Detail Cross section of Model

Table 4: Different boundary conditions for Slab Specimens as per IS456:2000

No.	Boundary Condition	Explanation
1.	Interior Panels :	All Edge Fixed
2.	One Short Edge Continuous:	One Short edge S.S, Others Fixed.
3.	One long Edge Discontinuous:	One Long edge S.S, Others Fixed.
4.	Two Adjacent Edges Discontinuous:	Two (1 Short,1 Long edge S.S),Others Fixed.
5.	Two Short Edges Discontinuous:	Two Short edge S.S, Others Fixed.
6.	Two Long Edges Discontinuous:	Two Long edge S.S, Others Fixed.
7.	Three Edges Discontinuous	One Long edge Fixed, Others S.S
8.	Three Edges Discontinuous	One Short edge Fixed, Others S.S
9.	Four-Edges Discontinuous:	All edge S.S

III. Result & Discussion

3.1. Test Specimens and Modeling

For finding out the parameters of Sandwich slab & Solid slab, they were compared and analyzed each other. As a result, here what shows the deformation & stress stability of the slab with workbench ANSYS 16.1 However, solution of slabs are compared on support conditions base, which are 9 cases to compare with each other, Solid slab & Sandwich slab are subjected to apply same load & support condition & compare with Deflection & Stress. This deflection was measured at the bottom of the slab & stress that measured to the bottom as well as at support.

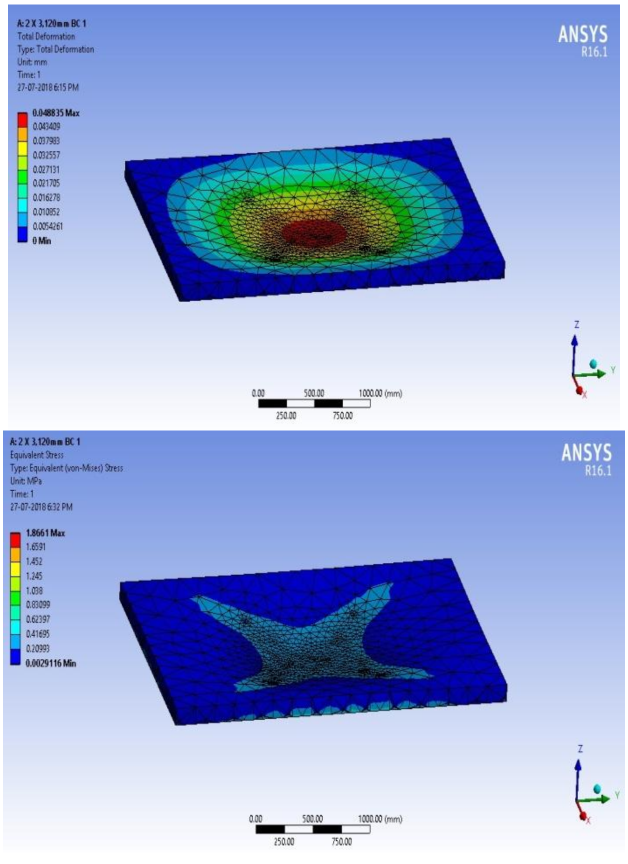


Figure 4: Deformation & stress for Solid slab

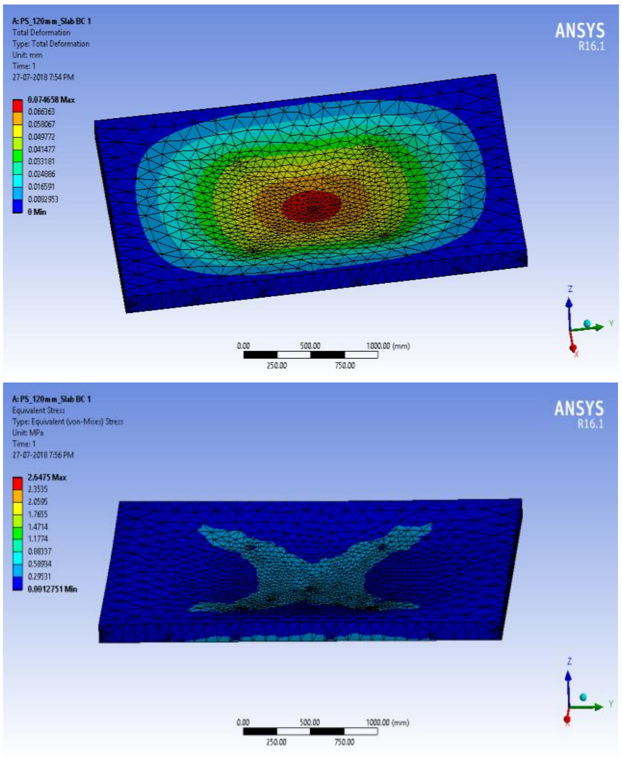


Figure 5: Deformation & stress for Sandwich slab

To derive the numerical simulations using Finite Element Methods were performed by the finite element program-ANSYS. To perform numerical simulations of the slabs by using finite element method, two dimensional models was often used when slabs were uniform in transverse axis. However, in the case of sandwich slabs, it was impossible to use two dimensional model because of the unequally section geometry along the longitudinal and transverse axis by sandwich shape. So, it was modeled using three-dimensional model which could be considered concrete web parts between sheet And to generate a finite element mesh of the sandwich slab. Tetrahedral elements which have four nodes were used because other mesh elements such as Hexahedral were not able to generate the geometry of the voided slab. Two material models were used to perform finite element method analysis.

The size of the slab is 3000mm x 2000mm x 120mm to both sandwich slab and for conventional slab with identical geometry.

The sandwich slab and solid slab applied UDL load for the surface area of about 3000mm x 2000mm over the top portion of the slab. An axially compressive load was applied uniformly on the top surface of slab with displacement control. The load was applied to an area and fixed of four end joint and the analysis is carried out.

Finite element analysis was carried out by using the FEA software-ANSYS to study structural behavior on the slab. The slab of Solid and sandwich slab were subjected to uniformly distributed load of 10 kN which was calculated through load calculation. The ultimate load, stress, deformation were measured by analytically. Solid slab and sandwich slab applying the 9 boundary condition provided in IS 456:2000 with the udl load of about 10 KN and causes deformation & stress.

3.2. Result discussion

There are 9 boundary conditions which are different support condition for both slab, Result Shows for the deformation & Stress. Above information was Carry out with the ANSYS. As per the Support condition it shows the stability of slab for solid slab & also Sandwich slab. The solid slab having less Deformation & Stress value & Sandwich slab Carried out it under the Criteria of IS 456:2000.That was individual result of each & every Boundary Conditions.

Show total result of deformation & stress for both the slab & chart for the deformation Vs Boundary Condition & also 2nd Chart about stress Vs Boundary Condition. These results shows us the stability of slab deformation & stress value can take this result to the Moment of the particular center of the slab as well as at boundary of slab. different conditions shows different result for the same slab.

3.3. Comparison of Result

Result Shows for the deformation & Stress. Above information was Carry out with the ANSYS. As per the Support condition it shows the stability of slab for solid slab & also Sandwich slab. The solid slab having less Deformation & Stress value & Sandwich slab Carried out it under the Criteria of IS 456:2000.

Table 5: Result Of Deformation & Stress

Boundary condition	Solid slab		Sandwich slab	
	Deformation	**Stress**	**Deformation**	**Stress**
1	0.0488	1.866	0.0746	2.647
2	0.0524	2.091	0.0779	2.853
3	0.0664	2.738	0.0927	3.6331
4	0.0709	2.727	0.0962	3.726
5	0.0539	2.086	0.0810	2.935
6	0.0784	3.119	0.1039	4.299
7	0.0737	3.045	0.0995	3.937
8	0.0828	3.339	0.1075	4.492
9	0.0869	3.471	0.110	4.597

Same as shown in figure 4 & 5, The result was derived for the different boundary condition, Total deformation shows centered deflection of slab,& also stress value at the center of the slab & at the end support where stress is at its maximum value, Result shown in table 5 & chart would be following,

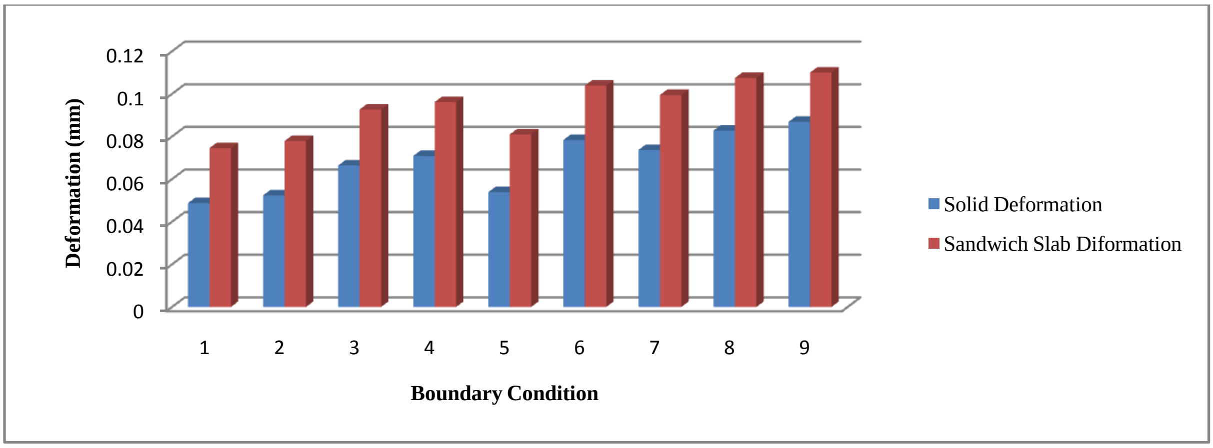


Figure 6: Deformation versus Boundary condition

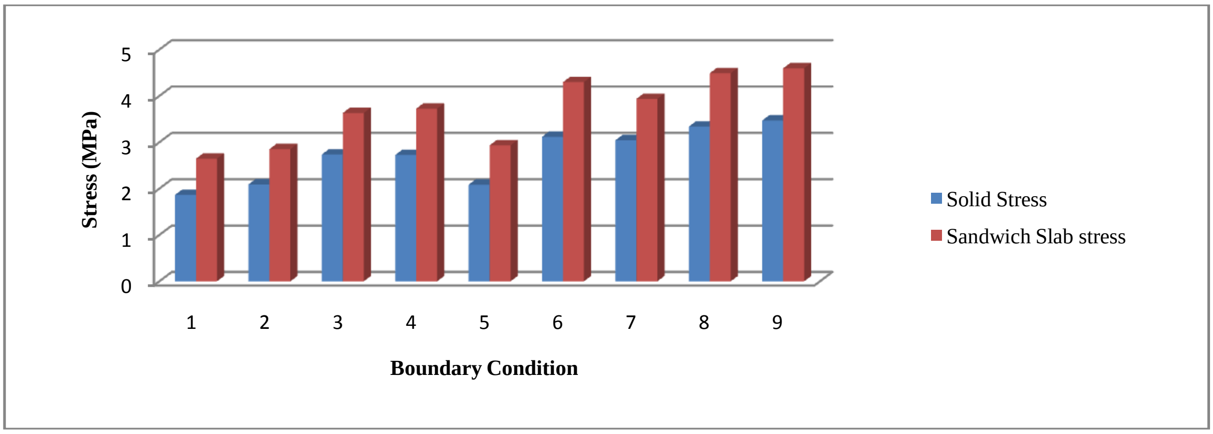


Figure 7: Stress versus Boundary condition

IV. Conclusion

In this study, several support conditions has been used for both the slabs. which were analyzed to compare with structure capacity and failure mode by using finite element method program ANSYS. Based on those result of numerical simulation, the conclusion derived.

- The use of Sandwich slab in steed of using solid slab has reduced the self weight of the slab as shown in Table 2

- The deflections & Stress generated due to service load of 10KN, Sandwich specimens were a little higher than those of a solid slab in each boundary conditions.
- The concrete that used in sandwich slab then solid slab is 35 % to 40 % less.
- Uses 9 boundary condition as per IS 456:2000 that shows different deflection & stress value. It has used less concrete so less Co2 production.

V. Future scope of work

- Use another material for core member, which chemically & physically compatible.
- Usages of different Software other than ANSYS for the analysis of the slab.
- Applying the slab for the special design & analyze that slab.
- Experimental work could be carried out for this research.

References

[1] Pradeepa .S , Anitha J, N. Tamil Selvi, Pranav P. and Arpit Jaketia,"A Study on use of Reinforced Thermocol Panels as an Alternate Building Material" International Journal of Research in Advent Technology, Vol.4, pp 613-617, March 2016

[2] Zarnani, S.,Bathurst, R.J.,"Numerical parametric study of geofoam seismic buffers with different constitutive models" 9th International Conference on Geosynthetics, Brazil,pp 1665-1670, May 2010

[3] Nahro Radi Husein, V. C. Agarwal, Anupam Rawat, "An Experimental Study on Using Lightweight Web Sandwich Panel as a Floor and a Wall", International Journal of Innovative Technology and Exploring Engineering (IJITEE) ISSN: 2278-3075, Volume-3, Issue-7, pp 69-75, December 2013

[4] Rohit Raj, Manoj Kumar Nayak, Md Asif Akbari and P. Saha, "Prospects of Expanded Polystyrene Sheet as Green Building Material" International Journal of Civil Engineering Research.ISSN 2278-3652 Volume 5, pp. 145-150, Number 2014

[5] Ministry of Housing and Urban Poverty Alleviation, Government of India," Manual for Expanded Polystyrene (EPS) Core Panel System and its field",CSIR – Central Building Research Institute Roorkee, June 2017

[6] Shriyush Chauhan, Dr. Tarak Vora,"Experimental Investigation On Insulating Concrete Forms (ICF) Slabs" International Journal of Advance Engineering and Research Development (IJAERD),ISSN 2347-4470, Volume-5, Issue-4,April-2018

[7] Shriyush Chauhan, Dr. Tarak Vora,"A Comprehensive Review on Insulating Concrete Forms (ICF) Systems" Journal of Structural Engineering, its Applications and Analysis, Volume - 1, Issue - 1, Feb-2018

[8] Roshan Gawale, Shubham Mishra, Harshal Sambare, Jidhnesh Kothari,"Light Weight Concrete By Using Eps Beads" International Journal Innovative Research in Science & Engineering (IJIRSE), Volume - 2, Issue - 3, pp- 470-476, March-2016

[9] S. Ananda Selvan, Dr. P.Asha,"Experimental Study on Lightweight Polystyrene Sandwich Blocks for Replacement of Bricks" International Journal of Engineering Technology, Management and Applied Sciences (ijetmas), Volume 4, Issue 3, ISSN 2349-4476, pp 135-139, March 2016

[10] Zou, Y. and Leo, C.J. [1998], "Laboratory Studies on the Engineering Properties of Expanded Polystyrene (EPS) Material for Geotechnical Applications", Proceedings of the 2nd International Conference on Ground Improvement Techniques, 7-9 October, Singapore, pp. 581–588.

[11] Aurelian C. Trandafir, Steven F. Bartlett,"Seismic performance of double EPS geofoam buffer systems"Fifth international confrence on Geotechnical earthquake engineering and soil dynamics.,may 24-29,paper no.5.10a., san diago, california, 2010

[12] ANSYS, "ANSYS Help", Copyright (2009).

[13] ANSYS Mechanical APDL Contact Technology Guide.

[14] ANSYS Mechanical APDL Element Reference.

[15] ANSYS Mechanical APDL Theory Reference.

[16] ANSYS Workbench.

IOSR Journal of Engineering (IOSRJEN) is UGC approved Journal with Sl. No. 3240, Journal no. 48995.

Yash Atulkumar Jani "Analytical & Comparative Studies of EPS Core Member in Sandwich Slab Using ANSYS" IOSR Journal of Engineering (IOSRJEN), vol. 08, no. 8, 2018, pp. 42-48.