Parametric Study on Steel-Foamed Concrete Composite Panel Systems

S.M.Leela Bharathi[#1], P.Prabha[*2]
[#]Assistant Professor, Civil Engineering Department, Karpagam University, Coimbatore -21, India
[*]Scientist, CSIR-Structural Engineering Research Centre, Taramani, Chennai-600113, India

Abstract

Construction in earthquake prone areas demand light weight, high strength, large ductility and avoidance of fragmentation of elements. Innovative precast light weight structural panels can make housing safer, durable, stronger, energy efficient, comfortable, affordable and speedier constructions. As a part of this effort, few experimental investigations are already carried out at CSIR-SERC on precast lightweight steel-foam concrete composite panels for use as load bearing walls and flooring systems. The previous experimental studies revealed that the proposed connection assembly to join the composite panels exhibited rigid connection behaviour. Further studies are required to optimize the connection assembly behaviour. In this paper, three dimensional (3D) nonlinear elasto-plastic finite element analysis is carried out for appropriately evaluating the load-deflection behaviour of connection assembly between steel-foam concrete composite wall-floor panels using ABAQUS software. The influence of various parameters that affect the connection assembly behaviour such as effect of bolt holes, angle thickness, number of bolts, grade and diameter of bolt and connecting panel thickness are varied in order to optimize the connection behaviour. The behavior of the model simulated by considering the effects of bolt holes matches significantly well with the experimental model with the overestimation of 11.6% in the load carrying capacity. Since the shear studs are not modeled for simplification purposes tie constraints are given in that portion which is the reason for overestimation of load carrying capacity. By taking this model as reference further parametric studies are carried out. From the parametric studies, it is found that the angle and connecting panel thickness has considerable effect on the strength and behaviour of connection assembly and it is found that 8 mm and 1.6 mm are the optimum thickness for the angle connection and the connection panel portion.

Keywords — *Precast light weight structural panel; steel-foam concrete composite panel; nonlinear elasto-plastic finite element analysis; tie constraints; shear studs.*

I. INTRODUCTION

In each and every achievement of mankind, infrastructure development plays an important role. The greatest challenges of construction in earthquake prone areas demand light weight, high strength, large ductility or deformability for in-plane and out-of-plane loading and avoidance of fragmentation of elements since these are the reasons for most of the damages and injuries due to peak dynamic loading. The lack of structural integrity of brick and masonry structures leads them to the catastrophic failure in seismic prone areas. Reinforced concrete and steel plate shear walls are traditionally used as axial or cyclic load-resisting systems in structures such as mid-rise and high-rise buildings. As a emerging trend in many countries composite walls are used as shear or core walls in steel frame buildings. Using composite action, the number of ingredient materials act monolithically to resist axial and lateral loads and high ductility of steel material leads to better seismic resistance. As an extension of the composite sandwich structures, Double skin profiled steel sheet composite wall (DPSCW) was devolped and the behavior under the construction and service loading conditions is studied by Wright and Gallocher (1995).

For the first time in CSIR-SERC, assembly level experimental study of double skinned load bearing wall to floor panel connection had been carried out by Prabha et.al., The composite panel consists of profiled cold-formed steel sheet, light weight infill foam concrete and through-through studs to achieve interaction between sheeting and concrete. From the above mentioned experimental results, the connection assembly behavior was found to be more rigid and hence the optimization of connection assembly is needed to be carried out.

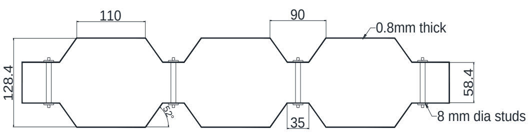
In this paper, the optimization of such composite wall to floor panel connection has been carried out by changing the dominant parameters which influence the system widely. Since this parametric study by conducting experimental work requires more time consumption and workmanship, the variation in each parameter is done by trial and error method using the Finite Element software Abaqus. For this purpose six parameters are chosen such as effects of bolt holes, connection angle thickness, bolt diameter, connection portion thickness and number of bolts and grade of bolts. From this parametric study it is found that if the angle connection thickness is reduced below 8 mm there is a significant draw down in maximum bending resistance. And there is no change in the bending resistance if the connection portion thickness is reduced below 1.2 mm.

II. MATERIALS AND METHODS

Before analyzing the influence of various parameters in the composite wall to floor panel connection assembly it is required to simulate the same numerically using the Finite Element (FE) software Abaqus. Then only the results can be interpreted and compared with original experimental structure and the influence of various parameters can be studied easily. The FE Simulation of the same experimental composite wall to floor panel connection assembly was already done (by the same authors of this paper in [1]) in the previous numerical study and bolts and studs were not modeled for simplification. The failure mode and the deformation of the specimen predicted by FE model was found to be agreeing well with the experimental observations. The difference in the ultimate capacity between FEA and experimental results is about 18% and this overestimation was imparted due to the utilization of node to node tie constraint to simulate the behaviour of studs/bolts in the model. Hence by taking the above mentioned numerical assembly as reference the parametric studies are carried out in this paper.

A. Geometric Modeling

After many trial and error methods the efficient corrugated panel profile is chosen from the previous experimental work carried out by Prabha et.al. The composite panel of width 685 mm and overall thickness 130 mm consists of trapezoidal crest and trough portions of length 110 mm at the interval of 35 mm. In this corrugated cross section, a pair of 0.8 mm thick cold formed steel sheets is separated by a 128.4 mm thick light weight foam concrete and the composite action is ensured by using 8 mm diameter studs in each rib portion as shown in Figure 1.



All Dimensions are in mm
Fig 1: Wall Panel Profile (inner to inner)

B. Assembly

In the assembly level behavioural study, the miniature model of G+1 storey load bearing structure is carried out by using the above mentioned composite panel profile. Hence in this case a Floor Panel (FP) is connected to the Wall Panels(WP) in top and bottom of the FP. Each WP of high 1050 mm is connected to the 860 mm span FP by means of ISA 100 x 100 x 8 mm angle connections. Angle section is connected to the WP and FP by using 6 nos. of (2nos. per trough) 16 mm dia. 8.8 grade bolts. The clearance allowance of 5 mm is also given in between WP and FP as a tolerance value.

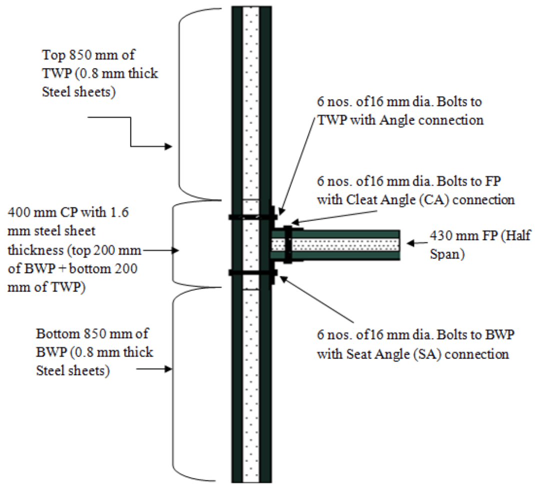


Fig 2: Composite Wall to Floor Panel Assembly

The connection between steel sheeting to foam concrete is ensured by means of 8 mm diameter studs at 250 mm spacing. By considering the axissymmetric condition in Finite Element Analysis (FEA), only half of the floor span (430 mm) is simulated. To achieve wall to wall interaction a composite Connection Panel (CP) with the same cross sectional profile is inserted into the Bottom Wall (BW) and Top Wall (TW) panel. The height of the CP is 400mm in which bottom half is inserted into the upper 200 mm of the BW panel and top half is inserted into the bottom 200 mm of the TW panel as explained in the Figure 2. This overlapping of cold formed steel sheets increases the thickness of sheeting thickness in this connection portion as 1.6 mm. Since the optimization of the connection behaviour is the major objective the influence of the bolts, only in the connection portion is taken in to consideration.

C. Material Properties

The composite wall and floor panel consists of 128.4mm thick foam concrete as infill material confined between a set of 0.8mm thick cold formed steel sheets. The properties of cold formed and hot rolled steel are found by using tension coupon test. The foam concrete properties are determined by using cube compression test. The tests are conducted and the test procedures are explained in the paper [1].

1) Cold Formed Steel

The composite panel system consists of cold formed steel sheets to confine the core foam concrete. To determine the material properties, sample specimens are made from these sheets for tension coupon test. The specimen dimensions are kept as per IS 1608 provisions. Totally three specimens are fabricated for the test. The test set up of the tension coupon test is shown in the Figure 3. The centre of the specimen is connected to the extensometer which is also connected to the automatic data acquisition system. The elongation of the specimen for the corresponding load increment is recorded automatically. From the data acquisition system the extension of the steel sheet, corresponding load increments and strain values are taken out.

a. Test set up b. Specimen after fracture

Fig 3: Tension coupon test

Young's modulus is calculated by measuring the slope of the stress-strain curve within the elastic limit. The average of the tension coupon test results are taken as the input for analysis. The plastic strain is calculated by subtracting the maximum elastic strain corresponding to the lower yield point from the total strain at that particular point. The stress-strain values between yield point to ultimate point (as shown in Table I) is given as the input plastic properties in Abaqus/CAE.

TABLE I

PLASTIC PROPERTIES OF COLD-FORMED STEEL

S.no	f_y in N/mm^2	Plastic strain $\varepsilon_{p\,(\mu m/m)}$
1.	190.85	0
2.	200.69	0.00161
3.	211.33	0.00425
4.	241.87	0.0129
5.	297.28	0.04282
6.	380.01	0.11845
7.	400.53	0.18607
8.	420	0.19

2) Foam Concrete

The core material used in the composite panel is light weight foam concrete (LFC). The main purpose of the LFC is to arrest the pre bucking failure of steel sheets in case of compression member and to reduce the ponding effect in case of flexural member. Ordinary Portland cement (OPC) of grade 53 conforming to IS 1269 (1987) is partially replaced by fly ash. The binder is mixed with the fine sand passing through 1.18 mm sieve conforming to IS 383 by the ratio of 1:0.87. Water to binder ratio is kept as 0.39. KV LITE a protein based chemical foaming agent is used to produce foam. By maintaining the water to foam ratio as 100:3.4 (in litres) the density of the foam kept as 70 to 80 g/lit. In order to define the plastic properties of concrete, the concrete damage plasticity model is chosen in which five parameters are used. The first parameter is dilation angle (ψ) which is the sloping angle taken from the plane of pressure invariant Vs second stress invariant. The rate at which hyperbolic flow potential approaches its asymptote is defined by flow potential eccentricity (ε) which is taken as second parameter in plastic properties. The third parameter which is taken into account to define the plastic properties of concrete is the ratio between initial compressive yield stress at equibiaxial state and uniaxial state (fb0/fc0).

The ratio between the second stress invariant on tensile meridian and compressive meridian is known as yield shape parameter (K) which considered as the fourth parameter. To define the visco plastic regularisation of the concrete constitutive equations in Abaqus / Standard, the viscosity parameter (μ) is used. As per the work done by Abdullah in the paper [4], the plastic parameters explained above cannot be found out from the experimental results. They were assumed using the values from the normal strength concrete. The plastic parameters used in this project are listed in Table II.

TABLE III

PLASTIC PROPERTIES OF FOAM CONCRETE

Dilation angle	Eccentricity	f_{b0}/f_{c0}	K	Viscosity parameter	
15	0.1	1.16	0.67	0	
Compressive behaviour			Tensile behaviour		
S.No	Yield stress f_y in MPa	In elastic strain μm/m	S.No	Yield stress f_y in MPa	Cracking strain μm/m
1.	1.76	0	1.	1.25	0
2.	2.144	0.0005	2.	1.25	0.00125
3.	2.544	0.0006			
4.	2.896	0.0007			
5.	3.248	0.0008			
6.	3.632	0.0009			
7.	3.76	0.00105			
8.	3.848	0.0011			
9.	3.96	0.00122			

3) Angle Connections

The elastic and plastic properties are measured from the normal bar tension test. The plastic strain in a particular point is calculated by excluding the maximum elastic strain corresponding to lower yield point of the stress strain graph. The yield stress used for the ductile material in the analysis is engineering stress. The plastic properties used are listed in the Table III. The engineering stress is calculated by using the ratio between applied load and original non-deformed area of cross section that is in perpendicular direction to the loading direction.

TABLE IIIII

PLASTIC PROPERTIES OF HOT ROLLED ANGLES

Sl. no	Yield stress in MPa	Plastic strain in μm/m
1	250	0
2	250	0.01125
3	410	0.12375
4	420	0.18

III. PARAMETRIC STUDY OF THE NUMERICAL ASSEMBLY

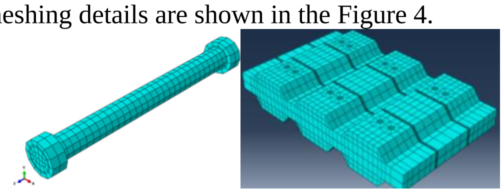
To optimize the connection portion behaviour of the composite wall to floor panel assembly, the effects of various parameters are studied by using the finite element software Abaqus. For this purpose the following six parameters are chosen such as,

- ❖ Effects of bolt holes,
- ❖ Connection angle thickness,
- ❖ Bolt diameter,
- ❖ Connection portion thickness
- ❖ Number of bolts and
- ❖ Grade of bolts.

The influence of each parameter on the assembly is validated by using the results of simulated Finite Element model which behaviour is similar to the experimental model.

A. Meshing Details

The 4 node Shell element with Reduced integration technique (S4R- a 4 node doubly curved thin or thick shell, reduced integration, hourglass control, finite membrane strains) is used for cold formed steel sheets. Continuum 3 dimensional 8 node solid element Reduced integration technique (C3D8R- An 8 node linear brick, reduced integration, hourglass control) is used for core foam concrete and bolts. The meshing details are shown in the Figure 4.

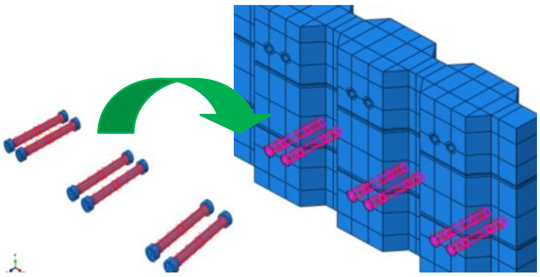


a. Meshing of bolts **b. Meshing in CP**

Fig:4 Meshing details

B. Interactions

To define the proper contact between the members the interactions are given (as shown in Fig.5). The interaction between concrete to concrete is given by using the friction coefficient 0.4. The steel to concrete interaction is given by using friction coefficient 0.45. The steel to steel interaction friction coefficient is taken as 0.3. These friction coefficients provide natural restraint to the movement and these are taken from the engineering material property table. As the threads are not modelled in the bolts, the surface to surface contact between the shank and the bolt holes are assumed to be frictionless.

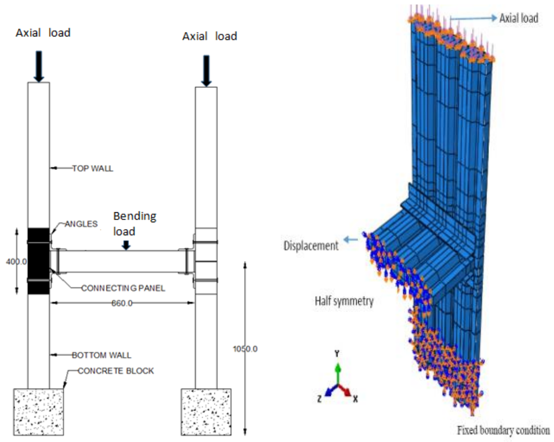


a. master surface **b. slave surface**

Fig:5 Interaction Between Bolt Shank and Bolt Holes

C. Loading and Boundary Conditions

As the assembly is having symmetry in both material properties and geometrical properties in z directions, only half of the assembly is taken for the analysis by considering z symmetric conditions. In the experimental study [2], the axial load of 2T is uniformly distributed to the wall panel top surface by means of loading plate. Therefore the total axial compressive force of 2T is applied as force per unit cross sectional area of top wall panel foam concrete on the top surface of the concrete alone. For steel sheets, the axial compressive load is equally divided as load per unit length of perimeter and is applied as shell edge load. The behaviour of floor under the action of bending is studied by applying concentrated load on the middle of the floor by an I beam having the flange width as 150mm and overall depth as 600 in the experiment. In analytical study this is achieved by applying displacement on the middle surface as shown in Fig 6.



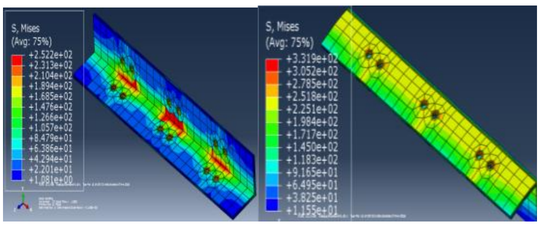
a. Experimental assembly **b. FE model**

Fig: 6 Loading Pattern

D. Results and Discussions

1) Effect of Bolt Holes

The numerical curve matches the experimental one in the plastic stage and after the ultimate point the curve follows the true stress-strain path. The maximum stress taken by the CA and SA is 252.2 MPa and 331.9 MPa respectively as shown in the Fig7. That is approximately 69% of the angle connection capacity is utilized. At the time of FP failure the CA and SA still have 40% and 21% of their capacity to withstand load.



a. Stress plot on CA **b. Stress plot on SA**

Fig: 7 Von Mises Stress Diagram on Angle Connection

The maximum bending load resisted by the wall to floor panel assembly at the support from the FEA is 122.45 kN which is 11.6% more than the experimental model [3] (109.7 kN) as in the Fig. 8a. And the maximum displacement measured under the

SA from the FEA is 4.76 mm which is 10.01 % lower than the experimental one measured under SA by left side LVDT (5.29mm) as shown in the Fig. 8b. Hence after considering the bolts modelling in FEA, the structural behaviour of the assembly matches with the experimental study. The results of this numerical study on the original assembly by considering bolt actions in the connection panel portion are taken as the reference for the parametric study and the analysis results are given in the following paragraphs.

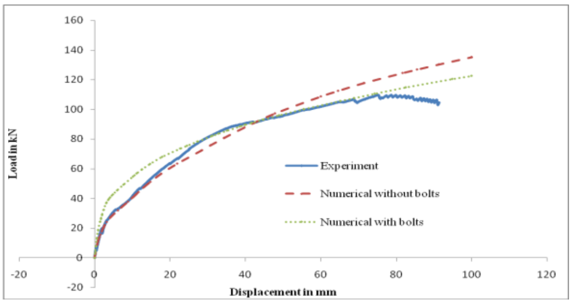


Fig: 8a Load Vs Displacement at Support

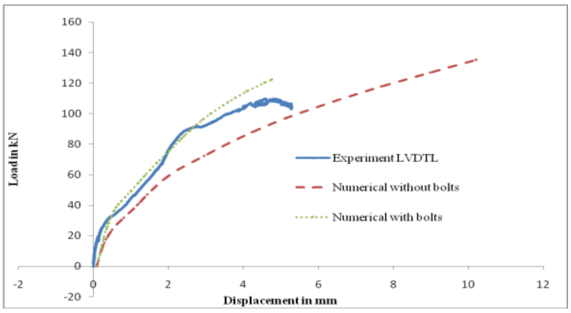
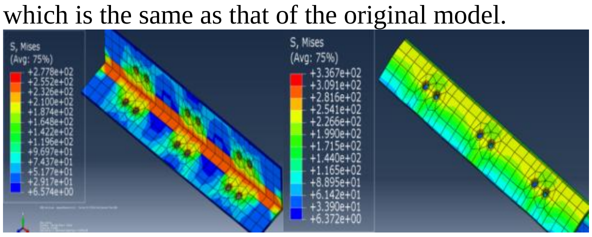


Fig: 8b Load Vs Displacement Under SA

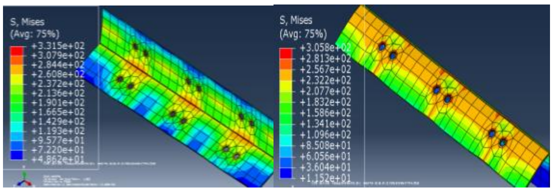
2) *Connection Angle Thickness*

The FEA result of each model with the different connection angle thickness such as 6 mm and 4 mm is compared with original model as shown in the Fig 11 a and b. The maximum bending load taken by the 6 mm thick connection angle assembly at support is 104.5 kN which is 14.65% lesser than the original model load carrying capacity. And the displacement measured under the SA is 4.76 mm which is the same as that of the original model.



a. Stress plot on CA **b. Stress plot on SA**

Fig: 9 Von Mises Stress Plot for 6 mm Thick Angle Connections



a. Stress plot on CA **b. Stress plot on SA**

Fig: 10 Von Mises Stress Plot for 4 mm Thick Angle Connections

The maximum stress taken by the CA and SA is 277.8 MPa (Fig 9 a) and 336.7 MPa (Fig 9 b) and approximately 73.15% of the ultimate capacity is utilized. Similarly in the 4 mm thick connection angle assembly resists the maximum bending load of 91.36kN at the support and the maximum displacement of 6.79 mm under the SA. By reducing the connection angle thickness from 8 mm to 4 mm, the displacement taken by the SA is increased by 42.65% and the capacity of the assembly is decreased by 25 % of the original model. The maximum stress taken by the CA and SA is 331.5 MPa and 305.8 MPa (Fig 10 a and b).

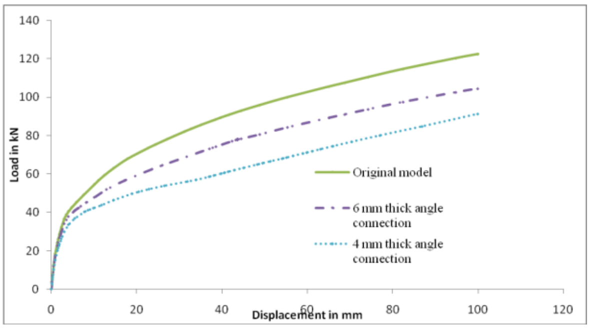


Fig: 11 a. Load Vs Displacement at Support for Variation in Angle Thickness

The approximate percentage of 75.87% capacity is utilized. At the time of FP the CA and SA still have 21% and 27% of their capacity to withstand load. By comparing the results of angle connection thickness variation in the wall to floor assembly, it is found that the decrease in the angle connection thickness would also reduce the capacity of the assembly.

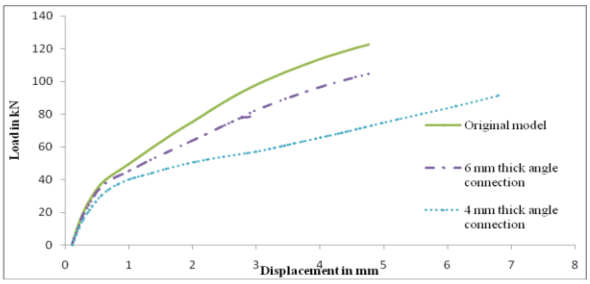
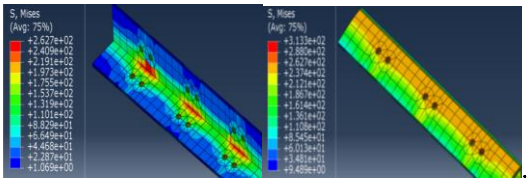


Fig: 11b Load Vs Displacement Under SA for Angle Thickness Variation

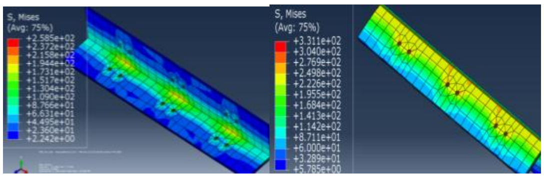
3) *Bolt Diameter*

The variation of load with respect to its corresponding displacement at the support as well as under the SA is plotted for each diameter variations in the Fig 12 a & b. And the curves are compared with the resultant curve of the original model. For the model having 12 mm dia. bolts the maximum bending load taken at the support is 117.18 kN which is 4.3% lesser than the original model and the maximum displacement measured under SA is 4.32 mm which is 9.2 % lesser than the original model. The maximum stress taken by the CA and SA is 262.7 MPa and 313.3 MPa in which average of 68.57% of ultimate capacity is utilized as in the Fig 13.



a.Stress plot on CA b. Stress plot on SA

Fig: 12 Von Mises Stress for Assembly Using 12mm Dia. Bolts



a. Stress plot on CA b. Stress plot on SA

Fig: 13 Von Mises Stress Plot for 8 Mm Dia. Bolts

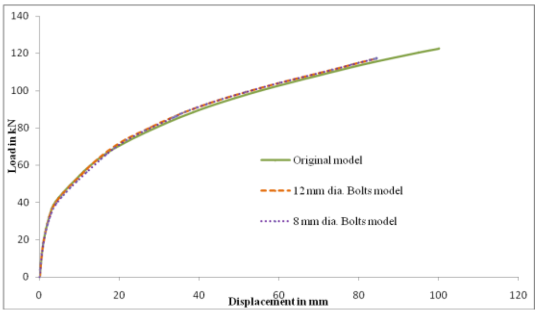


Fig: 14a. Load Vs Displacement at Support for the Effect of Bolt Diameter

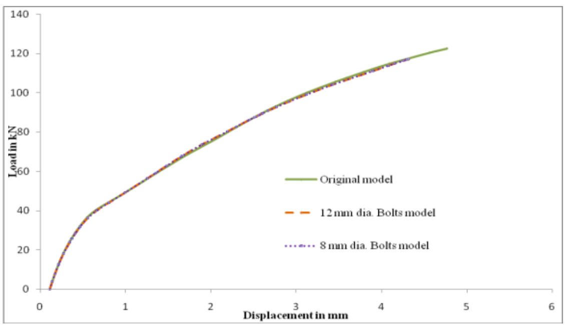
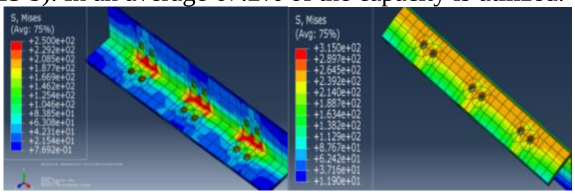


Fig: 14b Load Vs Displacement Under SA for Bolt Dia. Variation

In case of 8 mm dia. bolt model also the results are same that of the 12 mm dia. bolt model and the maximum stress measured on the CA and SA is 258.5 MPa (Fig 14a) and 331.1 MPa (Fig 14 b). The percentage of ultimate capacity used here is 70.2%. At the time of FP the CA and SA still have 38.5% and 21.2% of their capacity to withstand load. Hence from this part of the parametric study it is noticed that the decrease in the bolt dia. would not affect the connection behaviour significantly.

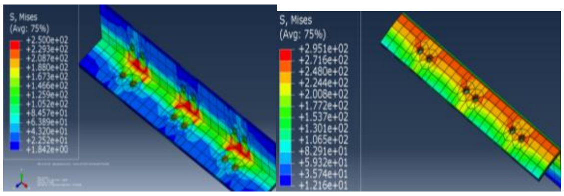
4) Connection Portion Thickness

The thickness values are varied to 1.2 mm and 1 mm and the variation of each is compared with the original model. For the steel sheet thickness of 1.2 mm, the maximum stress taken by the CA and SA connection is 250 MPa (Fig 15 a) and 315 MPa (Fig 15 b). In an average 67.2% of the capacity is utilized.



a. Stress plot on CA b. Stress plot on SA

Fig: 15 Von Mises Stress for 1.2 mm Thick Connection Portion



a. Stress plot on CA b. Stress plot on SA

Fig: 16 Von Mises Stress Plot for 1 mm Thick Connection Portion

The maximum bending load resisted by the assembly at support is 113.88 kN (Fig 17a) which is 6.9% lesser than the original model. And the maximum displacement measured under the SA is 4.81 mm (Fig 17b) which is 1.05% higher than original model. In the model simulated with the 1 mm thick connection portion, the maximum stress taken by CA and SA is 250 MPa (Fig 16a) and 295.5 MPa (Fig 16b) which is in average 64.9 % of the ultimate capacity. At the time of FP the CA and SA still have 40% and 30% of their capacity to withstand load The maximum bending load taken by the assembly at support is 109.43 kN which is 10.6% lesser than the original model. The variation in load-displacement behaviour of the assembly due to the change in connection portion thickness is shown in the Fig. 7.18 a & b. The maximum displacement measured under the SA in the 1 mm thick connection portion assembly is 5.25 mm which is 10.29% higher than the original model. The decrease in the connection portion reduces the capacity of the connection and the effect of connection portion thickness is not much significant after 1.2 mm.

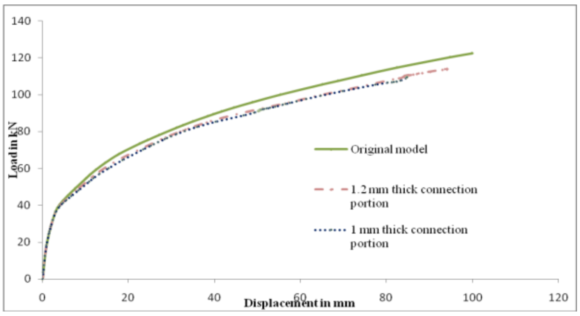


Fig: 17a. Load Vs Displacement at Support for the CP Thickness Variation

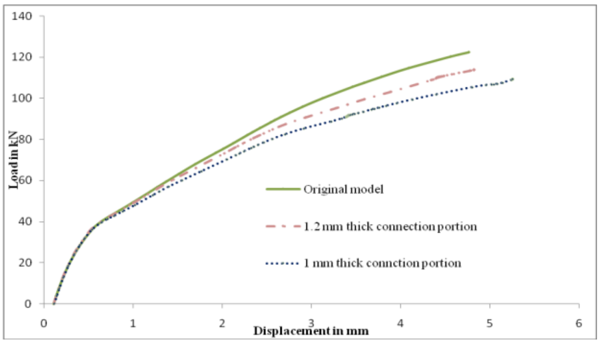
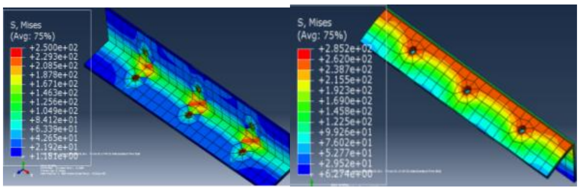


Fig: 17b. Load Vs Displacement Under SA for the CP Thickness Variation

5) *Number of Bolts*

From the FEA the maximum stress measured on the CA and SA is 250 MPa (Figure 21 a) and 285.2 MPa (Figure 21 b). At the time of FP the CA and SA still have 40.5% and 32% of their capacity to withstand load. That is from the ultimate capacity of the angle section 63.7% is utilized in an average. The maximum bending load resisted by the assembly is 118.85 kN (Figure 22 a) which is 2.94% lesser than the original model.



a. Stress plot on CA b. Stress plot on SA

Fig: 18 Von Mises Stress Plot for Change in Number of Bolts

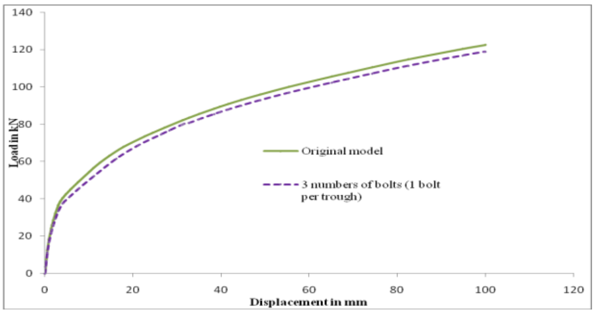


Fig: 19 a. Load Vs Displacement at Support for Variation in Bolt Numbers

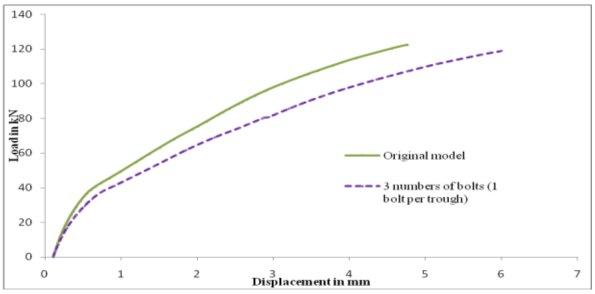


Fig: 19b. Load Vs Displacement Under SA For Variation In Bolt Numbers

The maximum displacement measured under the SA is 6 mm (Figure 22 b) which is 26 % higher than the original model. Even though the deformation is increased, the capacity of the assembly remains almost same that of the original model. In addition to that angle sections are entered into the plastic region when the floor panel fails. Hence the number bolts can be reduced to 1 bolt per trough.

6) *Grade of Bolts:*

The maximum load resisted by the support in the bending load case is 125.84 kN which is 2.76% higher than the original model and the maximum displacement measured under SA is 4.43 mm which is 6.93% lesser than the original model. From the graph it can be noticed that while reducing the grade of bolts there is not much significant change in the capacity of the wall to floor connection. Hence grade of bolts can be reduced to 4.6. The load Vs displacement behaviour is shown in the Fig 20 a and b. The maximum stress measured in the CA and SA is 250.9 MPa (Fig 21a) and 300.6 MPa (Fig 21b). The average ultimate utilized ultimate capacity is 65.5%. At the time of FP the CA and SA still have 40% and 28.4% of their capacity to withstand load.

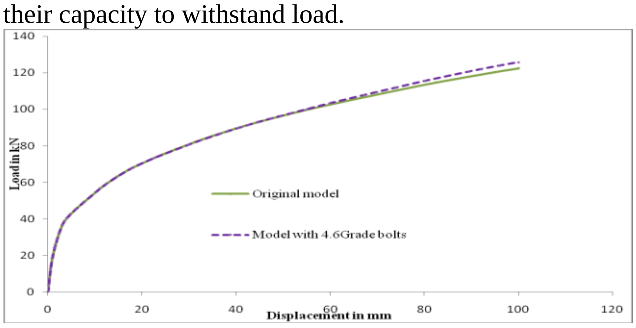


Fig: 20a Load Vs Displacement at Support for Variation in Grade Of Bolts

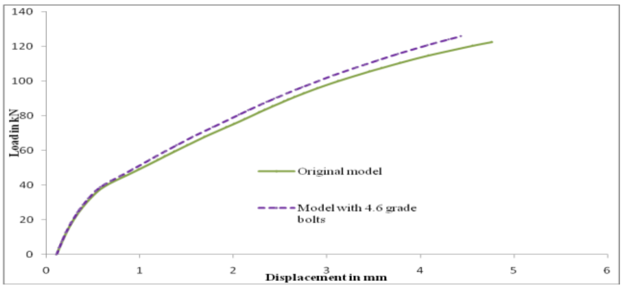
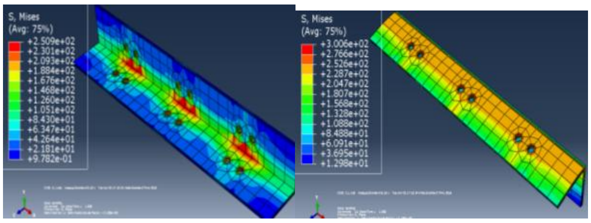


Fig: 20b Load Vs Displacement Under SA For Variation in Grade of Bolts



a. Stress plot on CA **b. Stress plot on SA**

Fig: 21 Von Mises Stress Plot for Variation in Grade Of Bolts

IV. SUMMARY & CONCLUSIONS

The experimental work on connection assembly behaviour between steel-foam concrete wall panel and floor panel of a G+1 building was conducted earlier at CSIR-SERC. It was found from the experiment, that the connection assembly exhibited rigid behaviour.

Hence the optimization of connection components is the main objective of this project. No codal provisions are developed for the design of corrugated steel-foam concrete composite panels and experimental trial and error method will lead to time consumption, money and workmanship. Hence numerical study has been carried out in this project to optimize the connection behaviour of the wall to floor panel assembly using ABAQUS, finite element software. Before going for optimization, the effects of bolts in the connection portion have to be studied. For that purpose, an original model is simulated by using the same profile, stud configurations used in the experiment. The difference between the comparative study model and this original model is that the bolts in the connection portions are modelled and their effects are studied in the original model. After considering the effects of bolts, the results of the numerical study match closely with the experimental work with a little variation of 11.6%. By keeping the original model results as reference, the change in behaviour due the variation of angle connection thickness, bolt dia., connection portion thickness, number of bolts and grade of bolts are studied in detail.

- The finite element model developed by modelling of bolts in the connection portion has reported improvement in the load-deflection behaviour with the strength variation of around 11 %. This model is adopted for further parametric studies.
- It is observed that the increase in the angle thickness from 4 mm to 8 mm has increased the ultimate strength of the connection assembly by 34%.
- Similarly while decreasing the connection portion thickness, the changes in the performance of the assembly is significant in between 1.6 and 1.2 mm. After 1.2 mm, if the thickness is still reduced the behaviour is similar to that of connection with 1.2 mm thickness.
- The bolt dia., number of bolts in the connection assembly and grade of bolts has least effect on the load-deflection behaviour and strength of connection assembly.

ACKNOWLEDGMENT

The authors thank the Director and Advisor (Management), CSIR-Structural Engineering Research Centre (CSIR-SERC), Chennai, for their constant support and encouragement. This paper is published with the kind permission of the Director, CSIR-SERC, Chennai, India.

REFERENCES

[1] P.Prabha, S.M.Leela Bharathi, G.S.Palani, R.Senthil, R.Theenathayalan, "FEA Simulation of connection behaviour between steel-foam concrete composite panels", in *Journal of Structural Engineering (JoSE)*, Feb-March 2017, Vol. 43, Issue 6, pp.529-538.
[2] P. Prabha , V. Marimuthu , M. Saravanan , G.S. Palani , N. Lakshmanan , R. Senthil, "Effect of confinement on steel-concrete composite light-weight load-bearing wall panels under compression" in Journal of Constructional Steel Research 81 (2013) 11–19.
[3] Dr.Rangachari Narayanan, "Steel Concrete Steel Sandwich Construction" in Recent Research in Composite Construction, Nov 16 (2014).
[4] Redzuan Abdullah, "Finite Element Analysis of Precast Lightweight Foamed Concrete Subjected to Axial Compression" in Proceedings of the 3rd International Conference on Computer Engineering & Mathematical Sciences (ICCEMS 2014), 4-5 December 2014, Langkawi, Malaysia.
[5] P. Krishna Kumar "Behaviour of Connections for wall and wall to slab, steel foam concrete panels" in B.S. Abdur Rahman University, Chennai, M.E thesis, June 2015.
[6] Rahmani Kadarningsih, Imam Satyarno, Muslikh, Andreas Triwiyono "Proposals of beam column joint reinforcement in reinforced concrete moment resisting frame" in Procedia Engineering 95 (2014) 158 – 171.
[7] K. Sai Lakshmi, P. Prabha, B. Ajitha and G. S Palani "Inplane Shear Behaviour of Steel – Foam Concrete Composite Wall Panels" in International Journal of Engineering Research & Technology (IJERT), Vol. 4, Issue 03, March-2015, pp. 2278-81.
[8] N. Anandavalli, N. Lakshmanan, Nagesh R. Iyer, G. M. Samuel Knight and J. Rajasankar "A Novel Modelling Technique For Blast Analysis of steel-concrete composite panels" in Procedia Engineering 14 (2011) 2429–2437.
[9] Howard Wright "Axial and Bending Behaviorof Composite Walls" in Journal of Structural Engineering / July 1998.
[10] A. Rahai, F.Hatami "Evaluation of composite shear wall behaviour under cyclic loadings" in Journal of Constructional Steel Research 65 (2009) 1528–1537
[11] K. M. Anwar Hossain, "Behaviour of composite walls under monotonic and cyclic shear loading" in Structural Engineering and Mechanics, Vol. 17, No. 1 (2004) 69-85.
[12] Wahalathantri B.L, Thambiratnam D.P, Chan T.H.T and Fawzia.S, "A Material Model for Flexural Crack Simulation in Reinforced Concrete Elements Using Abaqus" in eddBE2011 Proceedings, pp.260-264.
[13] S. Suryani and N. Mohamad "Structural Behaviour of Precast Lightweight Foamed Concrete Sandwich Panel under Axial Load: An Overview" in International Journal of Integrated Engineering - Special Issue on ICONCEES Vol. 4 No. 3 (2012) p. 47-52
[14] A. Ramachandra Murthy, V. Ramesh Kumar, SmithaGopinath, PrabhatRanjan Prem, Nagesh R. Iyer and ReshmiBalakrishnan "Structural Performance of Precast and Cast-in-situ Ultra High Strength Concrete Sandwich Panel" in CMC, vol.44, no.1, pp.59-72, 2014
[15] BS EN 1992-1-1: 2004 London: British Standards Institution.
[16] Eurocode 4 Design of composite steel and concrete structures Part 1-1: General rules and rules for buildings.