Analysis of Composite Panels of Profiled Steel Sheet with Ferrocement by Using Finite Element Method

https://doi.org/10.32792/utq/utj/vol14/4/9

Prof. Dr. Nabeel Abdulrazzaq Jasim(1)

M. Sc. Mohammed Abd Al-baqer Jennam(2)

1 University of Basrah, Department of Civil Engineering, Basrah, Iraq.

nabeel_ali58@yahoo.com

2 Real-estate Registration Office, Ministry of justice, Iraq.

mohammedcivilll11@gmail.com

Abstract

This paper concerns with the application of finite element technique to the nonlinear analysis of the behavior of profiled steel sheet ferrocement (PSSF) composite panels. The composite panels were analyzed using nonlinear three dimensional finite element models. The constitutive models of the nonlinear material behavior for ferrocement matrix, PSS, and steel wire mesh were used. ANSYS version 16.1 program code was used to analyze the three dimensional model. Ferrocement was modeled by using the 8-noded isoperimetric brick elements (SOLID65) with three degrees of freedom, while the profiled steel sheet was modeled as SHELL181 element with four-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes. A nonlinear spring element, COMBIN14, was used to represent the mechanical connectors, while to simulate the adhesive epoxy layer solid element SOLID185 was used. It was defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The wire mesh reinforcement was modeled as a volume ratio

distributed within the ferrocement brick elements. The analytical results of load-deflection response have been compared with available experimental works.

Keywords: Ferrocement, Composite Panel, Finite Element modeling, PSSF.

1-Introduction

The last years had shown significant steps in research in order to develop ferrocement as an active building material. Ferrocement is a form of reinforced concrete made of wire mesh and mortar, which has unique values of serviceability and strength. It can be constructed from easily available materials and it does not require a large number of skilled workers. It has many applications and uses in the area of low-cost houses, agriculture, and industry [1].

In civil engineering construction, different materials can be arranged in an optimum geometric configuration, with the aim that only the desirable property of each material will be utilized by virtue of its designated position. The structure is then known as a composite structure. Composite structures combine two or more materials in a unit structure to provide tangible benefits and a versatile solution to suite different applications. A composite system reduces the unnecessary and unwanted material properties, such as weight and cost, without sacrificing required capacity. A structure can be considered composite only so far as the various components are connected to act as a single unit. The structural performance depends on the extent to which composite action can be achieved. Composite structures, in general, have a higher stiffness and a higher load bearing capacity when compared with their non-composite counterparts. Hence the composite sections have got smaller section depth [1].

In the last 30 years several approaches have been used for the analysis of ferrocement structures, but the first use of finite element technique to analyze such structures was done by Prakhya and Adidan [2], who analyzed ferrocement slabs using rectangular hetrosis elements. Bin-Omer et al. [3] presented a computational model based on the Timoshenko beam finite element formulation using quadratic isoparametric elements with 3 degrees of freedom to analyze flanged ferrocement beams. Boshra Aboul-Anen et al. [4] used ANSYS software with Eight-node solid isoparametric element to represented ferrocement slab to study the composite action between the ferrocement slabs and steel sheeting. The objective of this paper is to develop a non-linear finite element model to simulate the behavior of profiled steel sheet ferrocement (PSSF) composite panels. The validity and calibration of the theoretical formulations and the program used is judged through comparison of analytical results with experimental data.

2- Modeling of (Profiled steel sheet with ferrocement) composite panels system

Three dimensional finite element modeling technique was used to investigate numerically the behavior of (PSSF) panels that were tested experimentally under the effect of static load [An Experimental Study of Composite Panels of Profiled Steel Sheet with Ferrocement, Which will be published later]. The software ANSYS finite element program version 16.1, Implementing ANSYS Parametric Design Language (APDL), is used in this analysis.

One of the main advantages of ANSYS is the integration of the three phases of finite element analysis: pre-processing phase, solution phase and post- processing phase. Pre-processing routines in ANSYS define the model, subdivide the problem into nodes, elements, and apply loadings and boundary conditions. Displays may be created interactively on a graphics terminal as the data are input to assist the model verification. Post-processing routines may be used to retrieve analysis results in a variety of ways. Plots of the structure deformed shape and the stress or strain contours to be obtained [5].

Nonlinear finite element analysis is adopted in the present study in order to investigate the behavior of the experimentally tested composite panels. ANSYS, including a variety of routines, allows for the implementation of specific material models to represent the profiled steel sheet, ferrocement, mechanical connectors, and epoxy adhesive layer, and boundary conditions.

2-1 Modeling of Profiled Steel Sheet

Profiled steel sheet is modeled, in this study, with SHELL181 element. It is used for analyzing thin to moderately thick shell structures. It is a four-node element with six degrees of freedom at each node: translations in the x, y, and z directions, and rotations about the x, y, and z-axes, as shown in Figure (1).

SHELL181 is well-suited for linear elastic, elastoplastic, creep, hyperelastic, and nonlinear material properties. Change in shell thickness is accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. The thickness of this element can be considered as a constant or varying smoothly over the area of the element by considering various values of the thickness at its four nodes. This element may be used for layered applications for modeling composite shells or sandwich construction. The basic formulation of this element was explained in the ANSYS 16.1 theoretical manual [6].

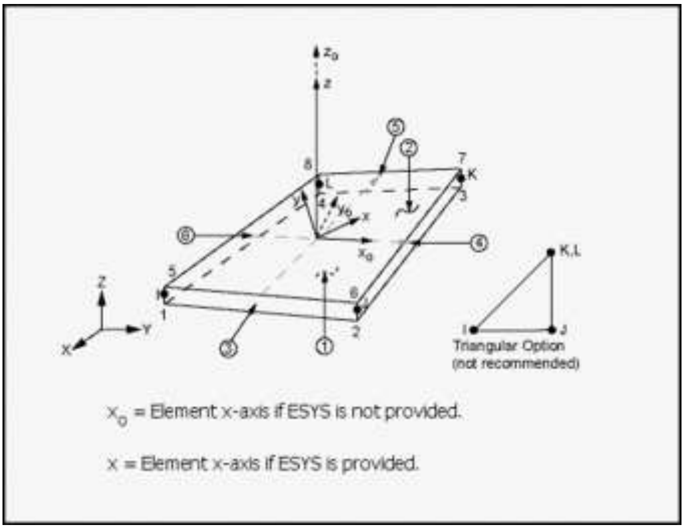


Figure (1) Geometry of Shell 181 (three- dimensional shell element) [6]

2-2 Modeling of Ferrocement Slab

The choice of the proper element is very important matter in the finite element formulation. For a reinforced concrete idealization, the element type depends upon the geometry of the structure and the number of independent space coordinates necessary to describe the problem. In the present study, SOLID65 three-dimensional reinforced concrete brick element, Figure (2), is used to model the ferrocement slab.

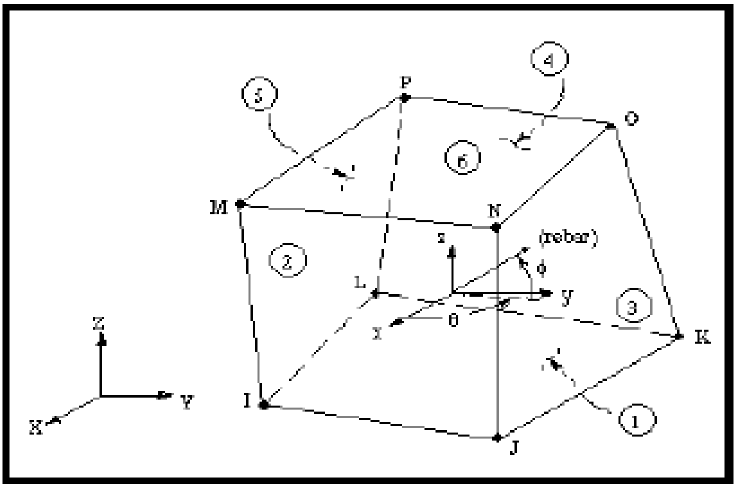


Figure (2) SOLID65-3D Reinforced concrete element [6]

SOLID65 can be used for the three-dimensional modeling of solids with or without reinforcing bars (rebars). The element is defined by eight nodes having three degrees of freedom at each node, translation in the x, y, and z directions. The element is capable of cracking in tension, in three orthogonal directions, crushing in compression, and plastic deformation. The rebars are capable of tension and compression, but not shear. They are also capable to reveal plastic deformation. The most important aspect of this element is the treatment of nonlinear material properties [5]. SOLID65 allows the presence of four different materials within each element; one matrix material and a maximum of three independent reinforcing materials. The matrix material is capable of directional integration point cracking and crushing besides incorporating plastic and creep behavior [7].

2.3 Representation of Wire Mesh

The steel is assumed to be distributed over the concrete element, with a particular orientation angle θ, Figure (3). A composite-reinforcement constitutive relation is used in this case. To derive such a relation, perfect bond must be assumed to occur between the concrete and the steel.

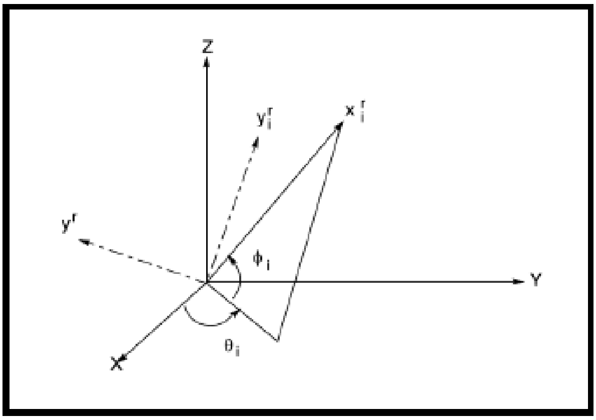


Figure (3) Reinforcement orientation for distributed model

2.4 Modeling of Mechanical Connectors

A nonlinear spring element, COMBIN14 in ANSYS, is used to represent the mechanical connectors behavior. COMBIN14 has longitudinal or torsional capability in 1-D, 2-D, or 3-D applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y, and z directions. No bending or torsion is considered, as shown in Figure (4).

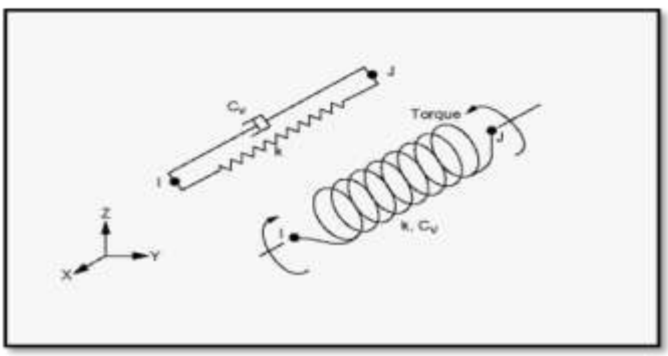


Figure (4) Geometry of Combin14

2.5 Modeling of Epoxy Adhesive Layer and interface surface

Two methods can be used to simulate the epoxy adhesive layer. In this study it is represented by using solid element SOLID185, (the element is used for the 3-D modeling of solid structures. It is defined by eight nodes having three degrees of freedom at each node: translations in x, y, and z directions as shown in Figure (5) [6].

3- Finite Element Formulation

The three–dimensional body in the finite element analysis is represented by a finite number of elements and a finite number of nodes that are identified on each element, where the elements are to be joined. The equilibrium equation for a nonlinear structure in a static equilibrium is derived using the principle of virtual work. This principle states that if a general structure in equilibrium is subjected to a system of small virtual displacements within a compatible state of deformation, the virtual work due to the external action is equal to the virtual strain energy due to the internal stress [8].

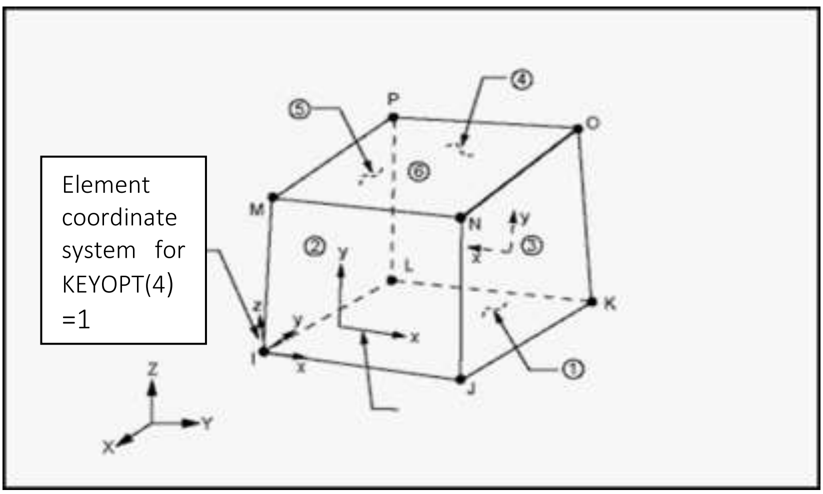
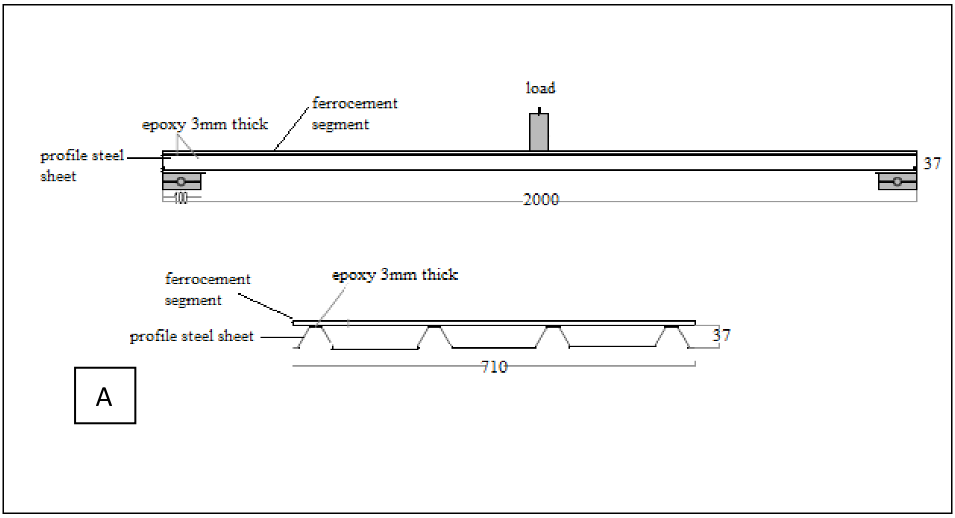


Figure (5) Geometry of Solid185 (three- dimensional solid element)

In the analysis of the composite (PSSF) elements, the behavior of a nonlinear material is due to the continual and sudden change in the element stiffness which arises from cracking, crushing of ferrocement, and yielding of steel sheet. These represent the main sources of the nonlinearity. A combined incremental-iterative technique is used to solve the nonlinear problem. Modified Newton-Raphson method is used where the stiffness matrix is updated at the second iteration of each load increment.

4- (PSSF) Composite panels and Materials

The geometry and the details of cross section for composite panels are shown in Figure (6). The composite PSSF panels were prepared with dimension of (0.71x2.0) m and tested under the action of line load at mid-span. All the tested PSSF composite panels comprise of single ferrocement panel connected on one side of the profiled steel sheet. The thicknesses of the ferrocement used are (10, 20,40) mm, and the thickness of the profiled steel sheet is (0.8) mm for all specimens. The composite panel specimens were fabricated by connecting the precast ferrocement segments on the ribs of PSS. Mechanical connectors and other suggested method of connection by using epoxy adhesive resin alone and epoxy adhesive resin in additional to mechanical connecters were examined. A summary of the materials properties of the selected specimens are listed in Table (1).



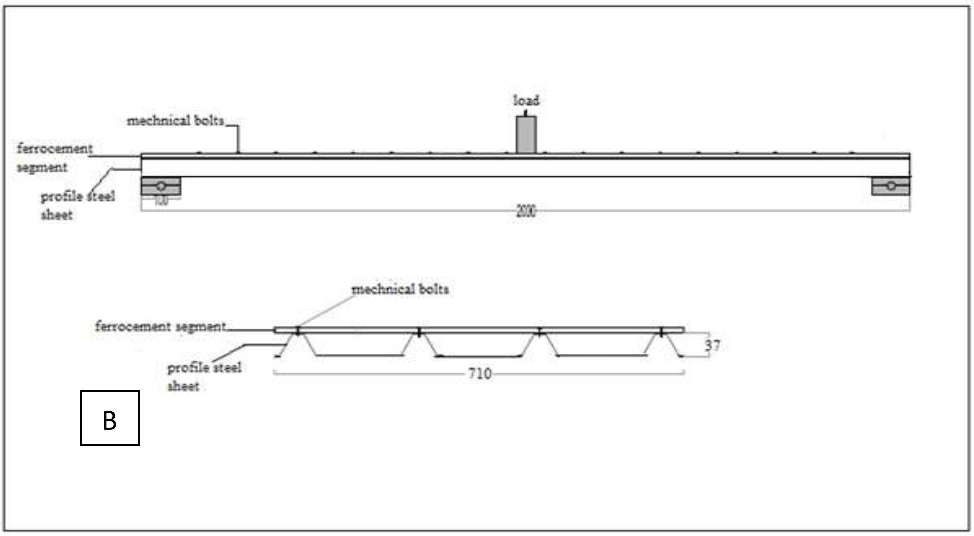


Figure (6) Typical PSS with ferrocement Panel Connected by (A) Epoxy Adhesive (B) Mechanical Fasteners.

Table (1). Materials properties

Materials	properties	value
Mortar	Young's modulus (MPa)	30000
	Compressive strength (MPa)	35 ,45
	Poisson's ratio	0.2
	Flexural strength *fr* (MPa)	5.5
	Crushing Strain,	0.0035
Steel Mesh	Diameter, mm	1
	Grid size, mm	12.5 x 12.5
	Young's modulus, MPa	92000
	Yield stress, MPa	415
Profiled Steel Sheet	Elastic Modulus(MPa)	200000
	Density(kg/m3)	7850
	Poisson's ratio	0.3
	Yield strength(MPa)	230
Epoxy Adhesive Resin	Density l/cm3	1650
	Poisson's ratio	0.4
	Elastic Modulus MPa	3300
Mechanical Connectors	Bolt Diameter (mm)	10
	Stiffness (N/mm)	5000

5- RESULTS AND DISCUSSION

A nonlinear finite element analysis has been carried out to analyze the composite PSSF panels. The analysis is performed by using ANSYS finite element computer program (Version 16.1). The ability of the method to simulate the behavior of this type of composite systems is demonstrated through the analysis of the tested panels. The results obtained by using finite element method are compared with the experimental results in terms of the load-deflection behavior and ultimate load values. The validity and accuracy of this method are examined.

The model was implemented by employing the ANSYS Parametric Design Language (APDL), implied in ANSYS software program. The implementation was started with the definition of the element types, element real constants, material properties and the geometry of the composite panel. Then, the mesh, coupling and linkage (connection) between the elements were added. After this step was completed, the solution processor was used to define the analysis type and analysis options, applying loads and boundary conditions, specifying load step options, and then initiating the finite element solution and calculating the deflection, stresses and strains at integration points of created small elements.

5-1 Mesh Size

An important step in the finite element modeling is the selection of the element mesh size. A convergence of results is obtained when an adequate number of elements is used in a structure. In the present study, and for best convergence, the overall span of each panel is divided into 80 elements (in y-direction). The composite panel cross section (in z-direction) is divided into one element for the ferrocement. In x-direction, the ferrocement panel is divided depending on the cross section of the profiled steel sheet. The rib flange and trough flange of PSS is divided into two and six elements, respectively, and the web into one elements. A typical discretization of the composite system is shown in figure (7) and figure (8). Three dimensional 8-noded brick element with three dof at each node is selected to represent the ferrocement, whereas three dimensional 4-noded shell elements with six dof at each node is selected to represent the PSS, and 8-noded brick element with three dof at each node is selected to represent the epoxy adhesive resin. Combine14 element with longitudinal capabilities is chosen for the modelling of bolt and in addition to this, surface to surface contact elements were introduced between the PSS and ferrocement elements to constitute the contact and sliding between these elements and for preventing the elements from piercing one another.

Displacement boundary conditions are needed to constrain the model to get a unique solution. The support was modeled in such a way that a roller was created. The translations UZ and UY on a single line of nodes on the left supporting plate were given values of zero. While, the right supporting plate was restrained with a single line of nodes in z-direction only. By doing this, the panel will be allowed to rotate at the support.

The external applied load represented by the plate in the actual experimental tests is modeled by an eight-node solid element, Solid185, at the mid span location. The application of the loads up to failure was done incrementally as required by the newton- Raphson procedure. Therefore, the total applied load was divided into a series of load increments (load steps).

5-2 Ultimate load

The ultimate loads of the PSSF composite panels obtained by the finite element analysis are summarized along with the corresponding experimental values in Table (2). Comparison between the results of finite element method and the experimental values of ultimate load shows reasonable agreement. For the three methods of connections, the predictions by the finite element method are very close to corresponding experimental values of ultimate loads

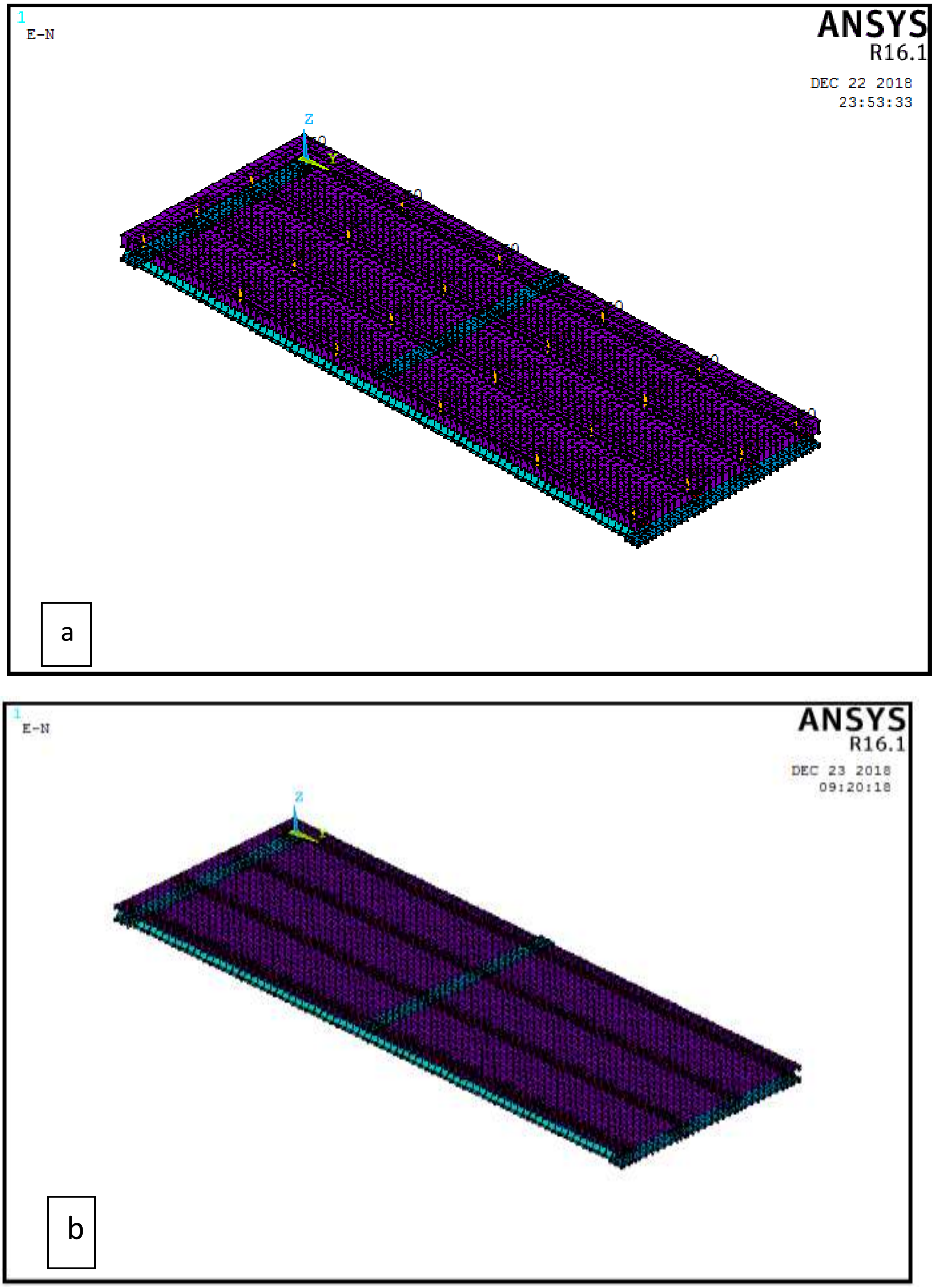
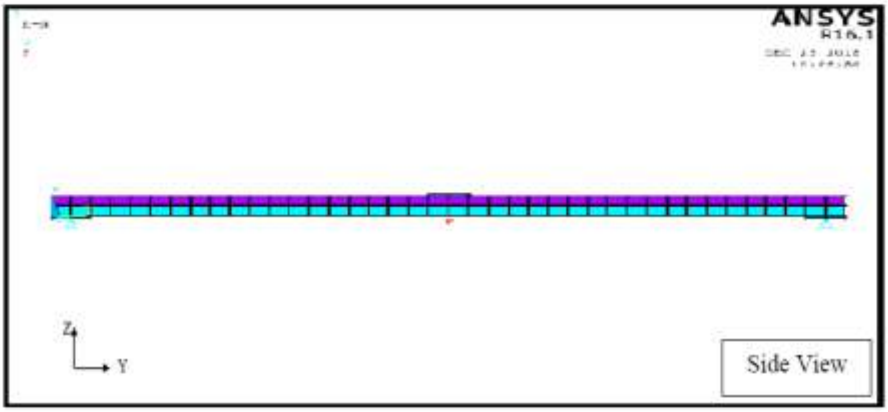
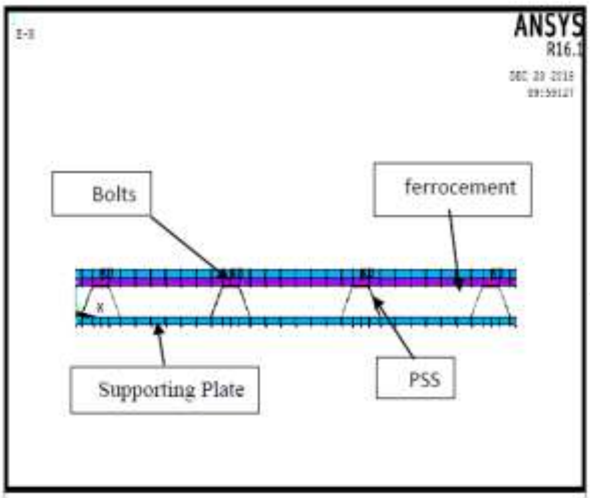


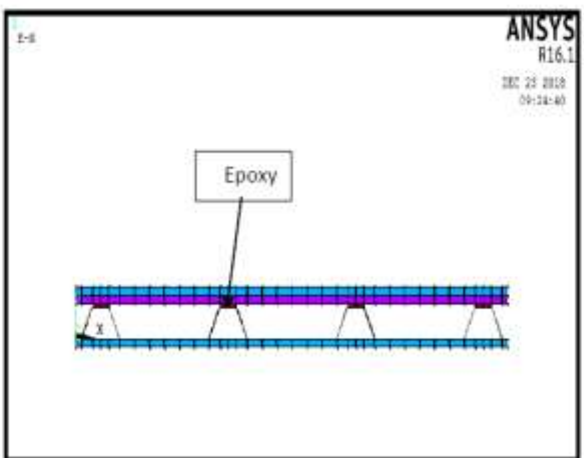
Figure (7) Finite Element Modeling of Composite Panels

Connected by: (a) mechanical fasteners (b) epoxy





Front View of Composite Panels Connected by Mechanical Fasteners



Front View of Composite Panels Connected by Epoxy

Figure (8) Discretization of Tested Composite Panels

Table (2):Results of ultimate loads

No.	Designation	Type of Connection	Spacing of bolts (mm)	Ultimate Load (kN)		$\frac{F_{FEA}}{F_{EXP}}$
				(F_{exp})	F_{FEA}	
1	2s1 (s=100)	Bolts	100	6	5.94	0.99
2	2s1 (s=50)	Bolts	50	7.9	7.8	0.98
3	2s\`1 (s=200)	Bolts	200	4.9	4.5	0.91
4	2s\`1 (s=100)	Bolts	100	6.3	6	0.95
5	2s\`1 (s=50)	Bolts	50	8.6	7.9	0.92
6	3s1 (E+s=150)	Epoxy and bolts	150	11	9.9	0.9
7	3s1 (E)	Epoxy	-	10.2	9.3	0.91
8	3s\`1 (E+s=200)	Epoxy and bolts	200	10.5	9.64	0.92
9	3s2 (s=150)	Bolts	150	11.68	11.40	0.97
10	3s2 (s=75)	Bolts	75	13.6	13.1	0.96
11	4s2 (s=150)	Bolts	150	12.5	12	0.96
12	5s2 (s=150)	Bolts	150	13.5	12.4	0.92
13	3s\`2 (E+s=380)	Epoxy and bolts	380	16.3	15.8	0.97
14	4s\`2 (E)	Epoxy	-	15.7	15.6	0.99
15	5s\`2(E+s=380)	Epoxy and bolts	380	16.65	16.25	0.97
16	5s4(E+s=380)	Epoxy and bolts	380	27.5	24.2	0.88
17	6s4 (E)	Epoxy	-	27	24.1	0.89

The symbols in Designation (2s1 , 2s\`1) mean :

2; The number (2) indicates the number of layers of wire mesh.
S; Ferrocement panel with compressive strength of 35 MPa.
1; The number (1) indicates the thickness of ferrocement panel in cm.
S\` ; Ferrocement panel with compressive strength of 45 MPa.

E ; Epoxy.
s; spacing of bolts in mm

5-3 Mid-span Deflection

The vertical deflections at the centre of bottom face of the composite panel for all tested panels were measured. The load versus deflection curves obtained from the finite element method together with the experimental plots are presented in Figures (9) to (15) which show in general a good agreement. The values from the finite element models were calculated at the same locations of the experimental measurements.

It can be noted from these figures and from Table (2) that the finite element solutions are in good agreement with the experimental results throughout the entire behavior.

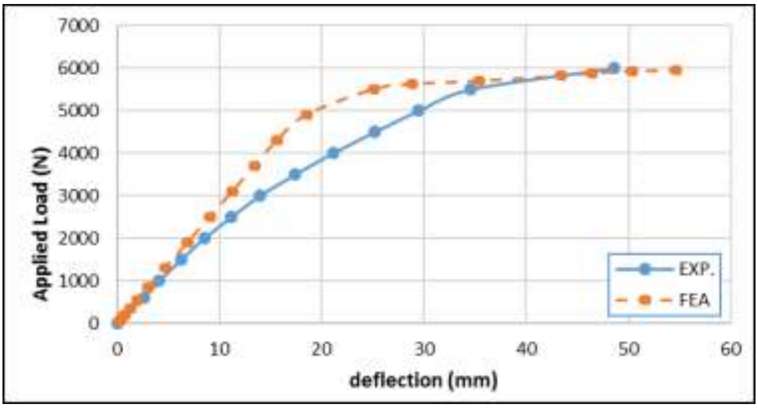


Figure (9) Mid-span Deflection Versus Applied Load for Composite panel 2S1 (S=100).

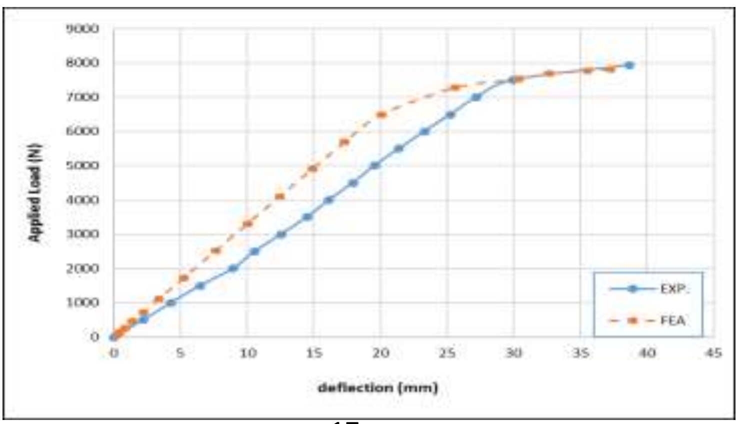


Figure (10) Mid-span Deflection Versus Applied Load for Composite panel 2S1(S=50)

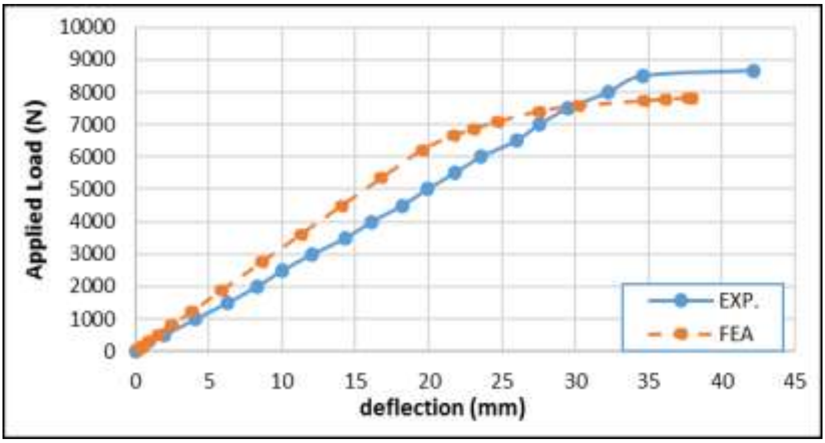


Figure (11) Mid-span Deflection Versus Applied Load for Composite panel 2S`1 (S=50).

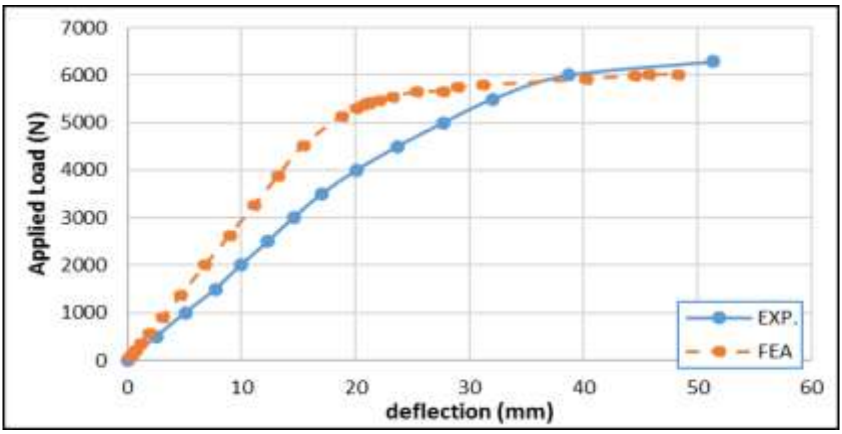


Figure (12) Mid-span Deflection Versus Applied Load for Composite panel 2S`1 (S=100).

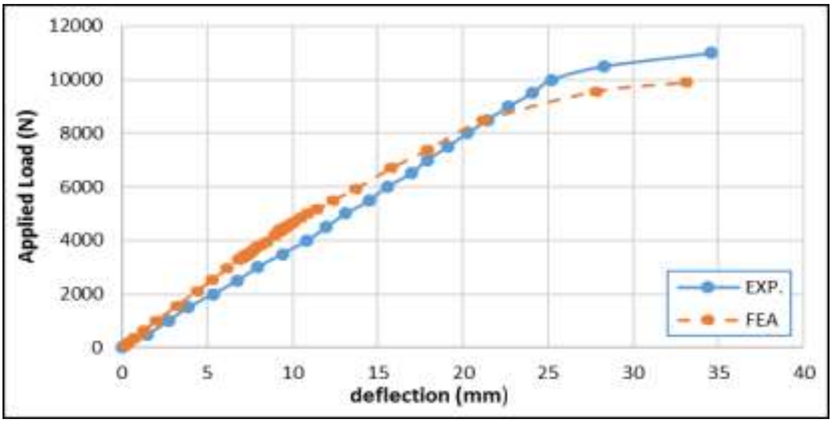


Figure (13) Mid-span Deflection Versus Applied Load for Composite panel 3S1 (E+S=150).

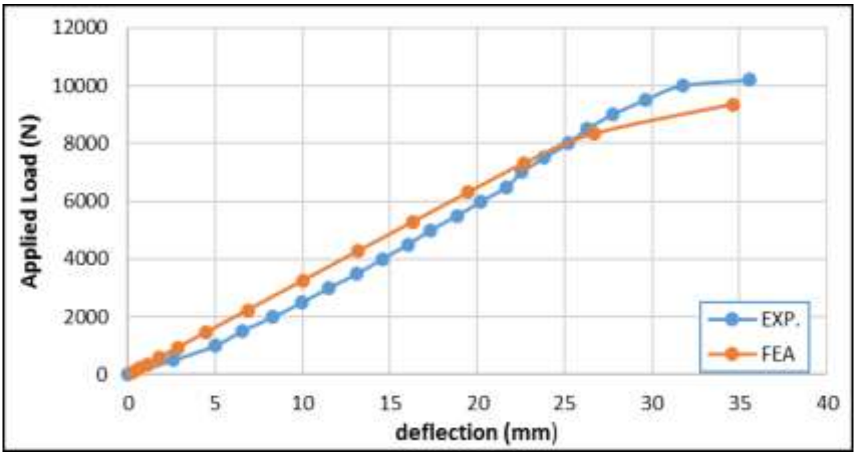


Figure (14) Mid-span Deflection Versus Applied Load for Composite panel 3S1 (E).

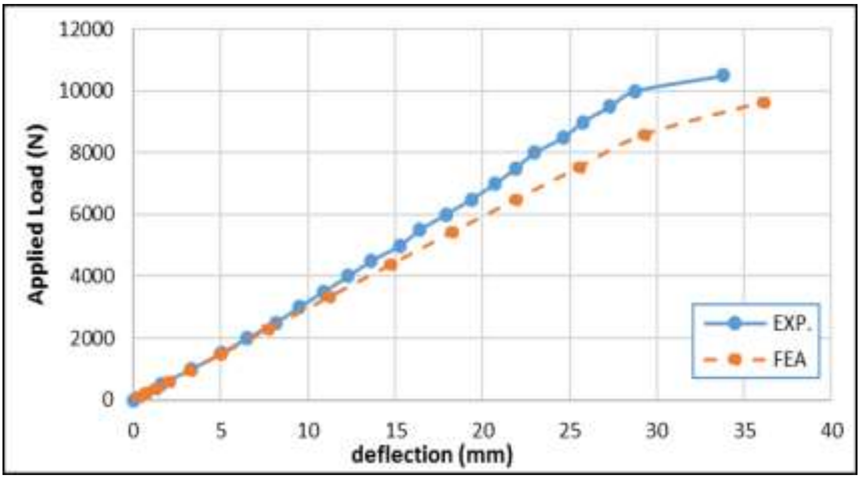


Figure (15) Mid-span Deflection Versus Applied Load for Composite panel 3S`1 (E+S=200).

6-Conclusions

In the nonlinear finite element analysis, the (ANSYS version 16.1) program is used with the following elements; three dimensional 8-noded brick element (SOLID65) to represent the ferrocement, three dimensional 4-noded shell elements (SHELL181) to represent the PSS and 8-noded brick element (SOLID185) for the epoxy adhesive resin. Combine14 element with longitudinal capabilities is chosen for modelling of bolts, and in addition to this the surface to surface contact elements are introduced between the PSS and ferrocement elements to constitute the contact and sliding between these elements and for preventing the elements from piercing one another. Acceptable agreement with the experimental results are obtained by this model. The average value of ratios of results of the finite element method to experimental results for ultimate load is 94%. Also the relationships of load-mid-span deflection are coincident.

References

1- Bouazaouia L., Jurkiewiezb B., Delmasa Y., and Lia A., "Static Behavior of A Full-Scale Steel - Concrete Beam with Epoxy-Bonding Connection", Journal of Engineering Structures, Vol. 30, February 2008, pp. 1981-1990.

2- Prakhya K. V. G. and Adidam, S. R.” Finite Element Analysis of Ferrocement Plates ”, J. Ferrocement, No.17, 1978, pp. 313-320.

3- Bin-OMAR, A. R. H.H. Abdel- Rahman, G. J. Al-Sulaimani, ” Nonlinear Finite Element Analysis of'Flanged' Ferrocement Beams ”, Computers & Structures, Vol. 31, No. 4, 1989, pp. 581-590.

4- Boshra Aboul-Anen, Ahmed El-Shafey, and Mostafa El-Shami, ” Experimental and Analytical Model of Ferrocement Slabs ” , International Journal of Recent Trends in Engineering, Vol. 1, No. 6, May 2009, pp. 25-29.

5- ANSYS, " Basic Analysis Guide", ANSYS Release 16.1, Inc, Copyrite ©2012.

6- ANSYS, " ANSYS Modelling and Meshing Guide", ANSYS Release 16.1, Inc, Copyrite ©2012.

7- Willam, K. J. and Warnke, E. P., "Constitutive Model for the Triaxial Behavior of Concrete", Proceedings, International Association for Bridge and Structural Engineering, Vol. 19, ISMES, Bergamo, Italy, 1975, pp.174.

8- Mottram, J. T., and Show, C. T., "Using Finite Elements in Mechanical Design". First Edition, McGrew-Hill Company, UK,1996.

تحليل الالواح المركبة من صفائح الفولاذ المضلعة والمربوطة بألواح الفيروسمنت باستخدام العناصر المحددة.

أ. د . نبيل عبد الرزاق جاسم [1]
م. محمد عبد الباقر جنام [2]
[1] أستاذ، قسم الهندسة المدنية، كلية الهندسة- جامعة البصرة، العراق.
[2] ماجستير هندسة مدنية، وزارة العدل ، العراق.

المستخلص:

يتضمن هذا البحث استخدام تقنية العناصر المحددة في التحليل الغير الخطي لسلوك الألواح المركبة المصنوعة من صفائح الفولاذ المضلعة والمربوطة بالواح الفيروسمنت. واستخدمت علاقات السلوك اللاخطي للفيروسمنت و صفائح الفولاذ المضلعة و شبكة التسليح. وتم اعتماد برنامج ANSYS version 16.1 لتحليل النموذج الثلاثي البعد، و جرت نمذجة الفيروسمنت باستخدام عناصر من نوع SOLID 65 بثمانية عقد ونمذجة صفائح الفولاذ المضلعة بعناصر SHELL181 بأربعة عقد فقط. استخدم COMBIN14 لتمثيل سلوك البراغي المعدنية، ولمحاكاة طبقة الايبوكسي اللاصقة استخدام عنصر صلب SOLID185. أما شبكة التسليح فقد تمت نمذجتها كنسبة من الحجم ضمن عناصر الفيروسمنت.و تم افتراض ترابط تام بين الفيروسمت وشبكة التسليح .ووجد بان نماذج العناصر المحددة تعطي قيم للأحمال القصوى وللإزاحات ذات اتفاق جيد مع النتائج المختبرية.