Nonlinear Finite Element Analysis of Reinforced Concrete Beams Strengthened with Externally Bonded Steel Plate using ANSYS

Dr. Mohamed Rauof Abdulkadir[1] - Professor
Zana Abdalla Aziz[1] - Asst lecturer
Jaza Hassan Muhammad[1] - Asst lecturer
[1] Department of Civil Engineering – College of Engineering - University of Sulaimani

Received : 23/11/2016
Accepted : 06/04/2017
DOI Link: https://doi.org/10.17656/sjes.10045

Abstract

The present work presents a numerical study to simulate the behavior of reinforced concrete beams strengthened with steel plates. The study is carried out using two un-strengthened RC beams and two RC beams strengthened with (1mm, and 3mm) steel plates. The beams are modeled and analyzed by nonlinear FEM using ANSYS v14.5. The numerical results are in good agreement with experimental load-displacement curves and ultimate load carrying capacity.

Keywords: Beam, strengthening, steel plates, ANSYS.

1. Introduction

In recent years, repair and retrofit of existing structures have been one of the most important fields of interest and research due to emerging of new materials and strengthening technique. Strengthening of structures is carried out due to many reasons such as designing errors, under estimated loads, risk from overloading due to seismic effects, alteration of building function and restoration of strength capacity due to fire and other environmental effects.[1]. Investigations including experimental work may become costly and requiring time, whereas the use of finite element analysis to study the response of reinforced concrete members under loading is less costly. Today many computer software are available that can model the nonlinear behavior of reinforced concrete elements and has made numerical modeling much more faster. [2].

Regarding R.C. beams strengthened with externally bonded materials, there are many theoretical and experimental investigations.

Alfeehan [3], investigated four beams of 100 x 150 mm cross section and 1500 mm long, the beams were reinforced with different reinforcement for flexural reinforcement top and bottom, and for shear reinforcement ϕ6mm stirrups at 60mm center to center were provided. The first beam was made without strengthening while the other three beams were strengthened with external steel plates of thicknesses 0.5mm, 1mm and 1.5mm. The reinforced concrete beams were analyzed using ANSYS software, the concrete was modeled using SOLID 65 element, LINK8 was used for modeling of reinforcing bars, the external steel plate had been represented by SHELL63 element. The study showed that the experimental failure load increase by 48%, 59% and 88% for the beams strengthened with 0.5, 1, and 1.5 mm steel plates respectively.

Abbas [4] analyzed the reinforced concrete beams using finite element method, using three dimensional solid element SOLID65 for concrete, and a solid element SOLID45 for steel plate, and a three dimensional layered element SOLID46 for carbon fiber reinforced polymer plate (CFRP). The results of his analysis showed that the average strength for all beams strengthened with steel plate were larger than the average strength for the beams strengthened by CFRP plates because the steel plate axial stiffness was more than twice the stiffness of CFRP.

Mahjoub, and Hashemi[5] tested experimentally and analyzed theoretically four high strength reinforced concrete beams, in which two beams were strengthened with FRP sheets. The elements used for modeling of the materials were SOLID65 for concrete, LINK8 for steel bars, and SOLID46 for FRP sheets. It was shown that the finite element model results show a good agreement with observations and data from the experimental full-scale beam tests, and significant increase in the flexural strength can be achieved by bonding CFRP sheets to the tension face of high strength reinforced concrete beams.

More and Kulkarni[6] tested a twenty simply supported rectangular beams of size 100mm x 150mm x 1200 mm strengthened with Aramid fiber polymer sheets, and created a models by ANSYS software. Element SOLID65 was used to model the concrete, LINK8 element was used to model steel reinforcement with two nodes element, SOLID45 element was used to model steel bearing plates, and SOLID45 element was used to model FRP composites. The results showed that the ultimate load carrying capacity for 0% damaged degree beams were increased after strengthening with single layer and double layers of 100 mm width AFRP strip by 27.59% and 48.27% respectively compared with control beam. The beams with 0%, 70% and 80% damaged degree showed a higher performance in terms of load carrying capacity, while 90% and 100% damage degree beams did not show appreciable increase in load carrying capacity.

2. Objective of the research

The use of steel plate as strengthening or repairing material is one of the cheap strengthening methods. The aim of the present work is to investigate the effect of using steel plate for flexure strengthening on the behavior of reinforced concrete beams, and to develop a model for analysis of such beams by using the (ANSYS 14.5) software.

3. Finite element analysis

Finite element method (FEM) is a numerical method used for solving a differential or integral equations and obtaining an approximate solutions to a wide variety of engineering problems. It has been applied to a number of physical problems, where the governing differential equations are available. The method essentially consists of assuming the continuous function for the solution and obtaining the parameters of the functions in a manner that reduces the error in the solution [7]

ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers [8].In this work, a three-dimensional finite element modeling by using ANSYS 14.5 software has been conducted. Materials idealization and the elements used to build these models are listed below:

3.1. Elements type

The three dimensional element Solid65 is used for modeling solid concrete beams. This element has eight nodes as shown in Fig. (1) with three degrees of freedom at each node, which are translations in the x, y, and z directions. The Solid65 element is capable to estimate plastic deformation, cracking in three orthogonal directions, and crushing of concrete [8].

The modulus of elasticity of concrete is calculated by equation (1) [9].

$$Ec = 4700\sqrt{f_c'} \quad \text{-------------------------- (1)}$$

Where:

Ec = Modulus of elasticity of concrete, MPa $f'c$ = compressive strength of concrete, MPa

The ANSYS program requires the uniaxial stress-strain relationship for concrete in compression. The simplified compressive uniaxial stress-strain curve shown in Fig. (2) is adopted [10].

Numerical expressions, equations (2) [11] had been used to construct the uniaxial compressive stress-strain curve for concrete.

$$f = \frac{E\ \varepsilon}{1 + \left(\dfrac{\varepsilon}{\varepsilon_o}\right)^2} \quad \text{--------------------- (2)}$$

Where:

f = Stress at any strain ε, MPa.

E= Constant (Same as initial tangent

modulus) such that, MPa. $E = \frac{2f'c}{\varepsilon_o}$

ε = Strain at stress f.

ε_o=Strain at the ultimate stress $f'c$.

LINK180 element is used to model steel reinforcement. This element is a 3-D spar element and it has two nodes as shown in Fig. [3]with three degrees of freedom, which are translations in the x, y, and z directions. This element is capable to estimate the plastic deformation.

The steel stress-strain relation used is shown in Fig. (4) which exhibits an initial linear elastic portion, a strain-hardening range in which stress again increases with strain. The extent of the yield plateau is a function of the tensile strength of steel. [12]

$Ew = 0.1Es$

Where,

Es = Modulus of elasticity of steel bar, 200000MPa

Ew = The tangent modulus of steel after yielding, MPa

ε_s = Strain in steel bar

ε_{yo} = Strain in steel bar at yield point

σ_s =Stress in steel bar

A 3D SOLID185 element is used for modeling the steel plate. It is defined by eight nodes having three degrees of freedom at each node: translations in the x, y, and z directions. The element exhibits plasticity, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elasto-plastic materials, and fully incompressible hyper-elastic materials [8].

For externally bonding between concrete surface and steel plates, CONTA174 element is used. CONTA174 is used to represent contact and sliding between 3-D "target" surfaces (TARGE170) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analyses. The element has the same geometric characteristics as the solid or shell element face with which it is connected. Contact occurs when the element surface penetrates one of the target segment elements (TARGE170) on a specified target surface. Coulomb friction, shear stress friction, user-defined friction with the USERFRIC subroutine, and user-defined contact interaction with the USERINTER subroutine are allowed. The element also allows separation of bonded contact to simulate interface delamination [8].

4. Beams geometry and material properties

The reinforced concrete beams tested by Al-Hassani, et al [13] are chosen to be modeland study their behavior using the described method.

The geometry of the beams along with the reinforcement details and the material properties as reported [13] are shown in Figs.(5 & 6) and Table (1). The testing program consisted of twelve reinforced concrete beams from three batches of concrete mixes and accordingly they were classified into three groups.

The first two groups were identical and the dimensions were kept to (125×160×1600 mm), while the dimensions of the last group were 185×165×1000 mm. All beams were simply supported beams, the clear span of Group (1) and Group (2) beams was 1500 mm and that of Group (3) beams was 900 mm. They were all tested by applying two central loads spaced 300 mm apart. Table (2) shows the values of experimental ultimate load.

5. Finite element investigation

From the testing program of AL-Hassani, et al. [13] four beams are selected for the numerical investigation using ANSYS, the beams are modeled as solid element. The first beam model 1600 mm long, with a cross-section of 125mm x 160mm is called Beam Control-1.

The beam reinforcement at the bottom of the beam is 2ϕ12.7mm and the reinforcement at top of the beam is 2ϕ2.5 mm, stirrups are ϕ8 mm @ 100 mm c/c as shown in Fig.(5)[13].

The second beam named control-2 is 1000 mm long, with a cross-section of 185 mm x 165 mm. The beam reinforcement at the bottom of the beam is 2ϕ9.5 mm and 1ϕ12.7 mm, the reinforcement at top of the beam is 2ϕ2.5 mm, and stirrups are ϕ8mm @ 75 mm c/c as shown in Fig.(6)[13].The summary of the input data is given in Table (3).

After analyzing the control beams, another two beams are modeled, one of them have the same property of the beam control-1 but strengthened with 1 mm thickness of steel plate and named beam Group-1, the other one have the same property of the beam control-2, but strengthened with 3mm thickness of steel plate and named beam Group-3. The plates are bonded to the bottom of the beams. The length of the steel plate is 1000 mm for beam groups 1 and 750 mm for beam Group-3. Finite element analysis requires meshing of the model; hence the models are divided into a number of small elements. As shown in Fig.(7 & 8).

6. Analysis results and discussion

The test results of Al-Hassani, et al.[13] showed that the control beams failed in flexure at ultimate loads of 58.89kN,and 106.6kN for beam B1 of Group-1 and beam B12 of Group -3 respectively, while from the ANSYS analysis the ultimate loads were 55.125kN, and 111.78kN for Control-1 and Control-2, respectively, indicating close agreement with the experimental ultimate load, see Table (4). From the experimental study, the control beams (B1 & B11) failed in flexure, while the ANSYS control beams also failed in flexure by yielding of tensile steel bars, as shown in Figs.(9 & 10).

After the strengthening process, the strengthened beams of group-1 had failed due to yielding of steel plate at ultimate load of 65kN (average for the identical beams), while the ANSYS model of the strengthened beam for Group-1, failed also by yielding of the steel plate as shown in Fig.(11) same as the experimental beam however at ultimate load of 59kN.

For Group-3 the mode of failure changed from flexure to shear failure as shown in the Fig.(12) at ultimate load of 195.6kN (average). Similar to test results, the ANSYS strengthened beam for Group 3 failed by yielding of stirrups at ultimate load of 242 kN as shown in Fig. (13). The addition of the steel plate lead to shift the failure from flexure at mid span to shear failure due to yielding of the stirrups as shown in Figs. (13 and 14). Although Fig.(13) indicates as well that the main reinforcements have also yielded not at mid span where the strengthening plate is added but beyond the end of the plate at the shear span, however shear failure has been triggered first due to stirrups yielding rather than flexure failure due to yielding of main bars at that region. Fig.(15) also shows the stress contour of the same ANSYS strengthened beam group 3 where the steel plate stress near mid span is close to the experimental yielding strength, and the concrete stress at the top fiber has exceeded its compressive strength at load 242 kN which should indicate a compressive flexural failure as in an over-reinforced concrete beam due to the addition of the steel plate. However the actual failure load is much lower (195.6 kN) due to shear failure as mentioned above, and the beam did not show any crushing of the concrete at top fiber hence the ANSYS ultimate load could not be reached therefore the discrepancy in ultimate load is high.

Fig.(16) shows the load displacement curves at mid span for the control and strengthened (1 mm plate) beams for both experimental and ANSYS results. Displacement results compare fairly in both cases. Fig.(17) shows the load displacement curves at mid span for the control and strengthened (3 mm plate) beams for both experimental and ANSYS results. Displacement results indicate ANSYS results show more beam stiffness than the experimental beams B11 and B12.

7. Conclusions

1- The three-dimensional nonlinear finite element model presented in the present work by using the computer program (ANSYS V.14.5) is able to simulate the analysis of RC beams strengthened with steel plate. The numerical results are in good agreement with experimental ultimate load carrying capacity. This numerical study can be used to predict the behavior of strengthened reinforced concrete beams more precisely by assigning appropriate material properties.

2-The numerical results are in fair agreement with the experimental load-displacement behavior.

References

1- Balamuralikrishnan, R., and Antony Jeyasehar, C. “Flexural Behavior of RC Beams Strengthened with Carbon Fiber Reinforced Polymer (CFRP) Fabrics”; *The Open Civil Engineering Journal*, Vol.3, pp.102-109. 2009.

2- Musmar, M. A., Rjoub, M. I., and Abdel Hadi, M. A. “Nonlinear Finite Element Analysis of Shallow Reinforced Concrete Beams using SOLID65 Element”; ARPN Journal of Engineering and Applied Sciences; Vol.9, No.2, pp. 85-89. 2014.

3- Alfeehan, A.A., “Strengthening Of R.C. Beams By External Steel Plate Using Mechanical Connection Technique”; *Journal of Engineering and Development;* Vol.18, No.2, pp. 202-212, 2014.

4- Abbas, A.L., “Non –Linear Analysis of Reinforced Concrete Beams strengthened with Steel and CFRP Plates”; *First Engineering Scientific Conference, College of Engineering –University of Diyala*, pp. 249-256. December 2010.

5- Mahjoub, R., Hashemi, S., H.. “Finite Element Analysis of RC Beams Strengthened with FRP Sheets under Bending”;*Australian Journal of Basic and Applied Sciences,*Vol.4, No.5, pp. 773-778.2010.

6- More, R., U., Kulkarni, D., B., “Flextural behavioral study on RC beam with externally bonded Aramid Fiber reinforced Polymer”; *International Journal of Research in Engineering and Technology,* Vol.3,No.7, pp. 316-321.2014.

7- Tobeia, S. B., “Behavior of Corbels Strengthened with Carbon Fiber Reinforced Polymers (CFRP) – Numerical Study”; *Eng. & Tech Journal*; Vol.32, PartA, NO.10, 2014.

8- ANSYS, Inc., "ANSYS Mechanical APDL Theory Reference", Release 14.5, 2013

9- ACI Committee 318. Building Code Requirements for Reinforced Concrete, ACI 318-14, American Concrete Institute, Detroit, 315 p. 2014.

10- Kachlakev D.I, Miller T., Yim S., Chansawat K and Potisuk T., "Finite element modeling of reinforced concrete structures strengthened with FRP laminates”, Technical Report, Oregon Department of Transportation, pp.99, May 2001.

11- Gere, I.M. and Timoshenko, S.P., “Mechanics of material”, PWS Publishing Company, Boston, Massachusetts.1997.

12- Cervenka, V., Eligehausen, R. and Pulkl, R., “Sbeta-Computer Program for Nonlinear Finite Element Analysis of Reinforced Concrete Structures”, Report 90/1, Institute of Building Materials, University of Stultgort, 1990.

13- Al-Hassani, H. M., Al-Ta'an, S. A., and Mohammed, A. A., “Behavior of Damaged Reinforced Concrete Beams Strengthened with Externally Bonded Steel Plate”; Tikrit Journal of Engineering Sciences, Vol.20, No.2, pp.48-59. 2013.

إستخدام العناصر المحددة غير الخطية لتحليل العتبات الخرسانية المسلحة والمقواة بالألواح الفولاذية خارجيا وبإستعمال ANSYS

د . محمد رؤوف عبدالقادر [1] – استاذ

زانا عبدالله عزيز [1] – مدرس مساعد

جزا حسن محمد [1] – مدرس مساعد

[1] قسم الهندسة المدنية – جامعة السليمانية

المستخلص :

يقدم هذا البحث دراسةً عددية لتمثيل سلوك العتبات الخرسانية المسلحة والمقواة بالألواح الفولاذية . تم تطبيق الدراسة على عتبتين بدون تقوية وعلى عتبتين مع تقوية بالواح فولاذية احداها 1(135x)ملم والثاني (135x3)ملم . واستخدم العناصر المحددة غير الخطية لتمثيل وتحليل العتبات بإستعمال برنامج ANSYS14 . بينت النتائج العددية توافقا جيدا مع النتائج المختبرية لسلوك الإزاحة والحمل ومقدار الحمل النهائي .

الكلمات المفتاحية : العتبات الخرسانية المسلحة ، تقوية ، الالواح الفولاذية ، ANSYS .

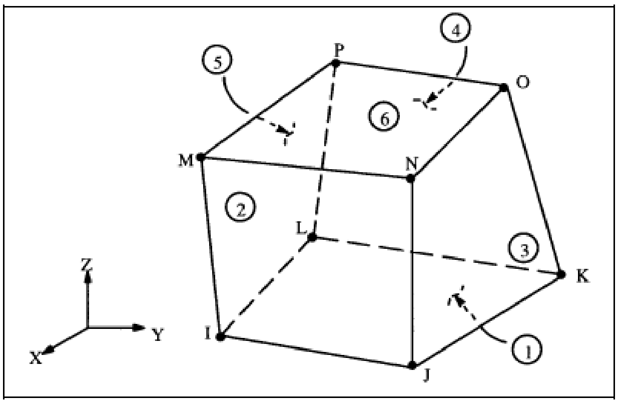

Fig. (1) : Solid 65- 3D reinforced concrete solid[8].

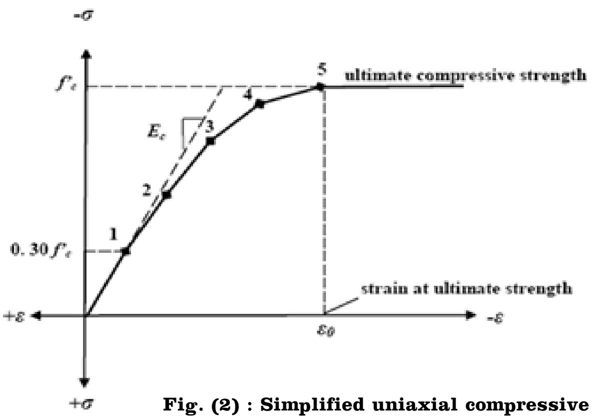

Fig. (2) : Simplified uniaxial compressive stress-strain curve for concrete [10].

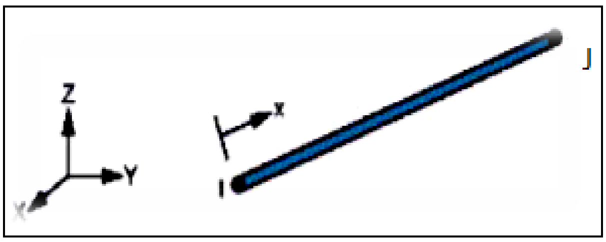

Fig. (3) : LINK180 element[8].

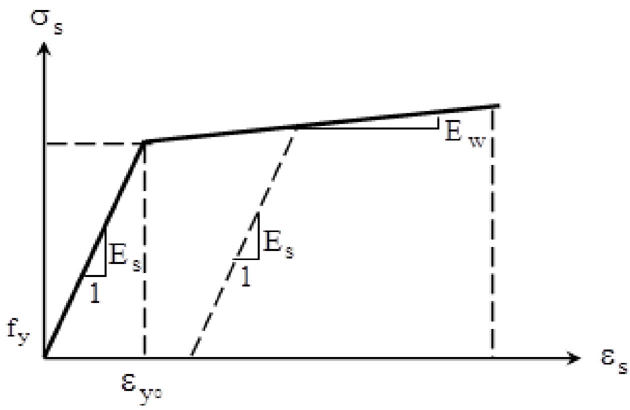

Fig. (4) : Stress-strain curve for steel bar.[12]

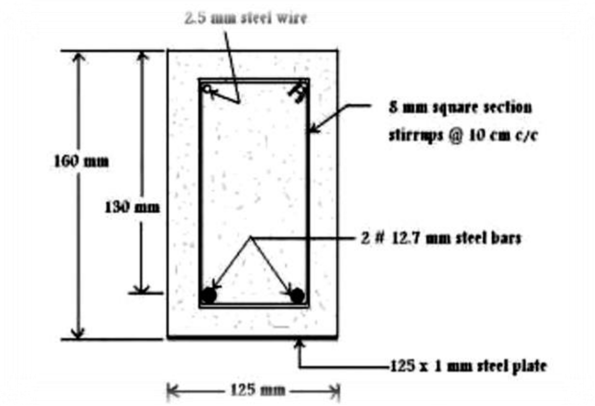

Fig. (5) : Detail of cross Section of Group (1&2) Beam[13].

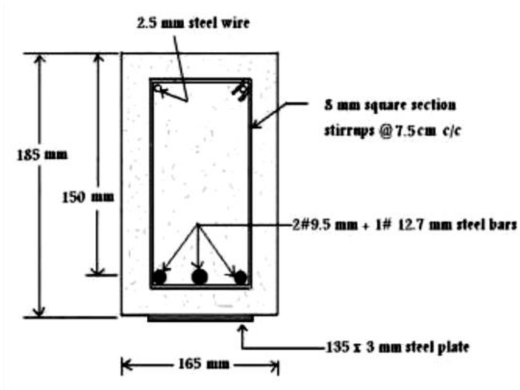

Fig. (6) : Detail of cross Section of Group (3) Beam[13].

Table (1) : properties of the steel reinforcement and steel plates of beams tested.

Material	Shape	Type of use	Strength (MPa)	
			Yield	Tensile
12.7mm	Bar	Flexure	579	672
9.5mm	Bar	Flexure	512	803
8mm	Square	shear	536	-
1mm thick	plate	strengthening	480	-
3mm thick	plate	strengthening	577	-

Table (2) : Results of ultimate flexural load of beams.

Group	Beam	Percentage of preloading ratio	Cube strength (MPa)	Test Ultimate Load (kN)	Ratio of test Ultimate load
	B1	-		58.89	100
	B2	0		68.99	117
1	B3	25	26.7	61.02	104
	B4	50		64.00	109
	B9	75		66.24	113
	B5	-		58.81	100
	B6	0		59.54	101
2	B7	25	23.9	67.61	115
	B8	50		62.77	107
	B10	75		61.11	104
	B11	-		106.60	100
	B12	0		181.60	170
3	B13	25	24.9	198.83	186
	B14	50		207.05	194
	B15	75		195.00	183

Table (3) : Model material properties.

Materials	Material model	Element type
Concrete	Linear isotropic Multi-linear isotropic Concrete	Solid 65
Steel Bar	Linear isotropic Bilinear isotropic	Link180
Steel Plate	Linear isotropic Bilinear isotropic	Solid 185
	Friction coefficient Emissivity	CONTACT 174surface to surface

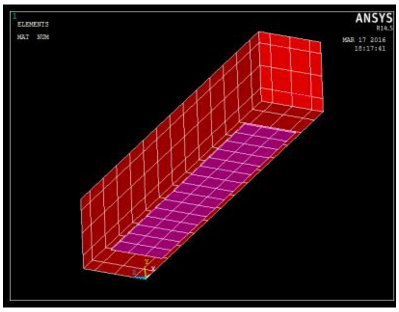

Fig. (7) : Meshing of concrete, reinforcement and Steel plate.

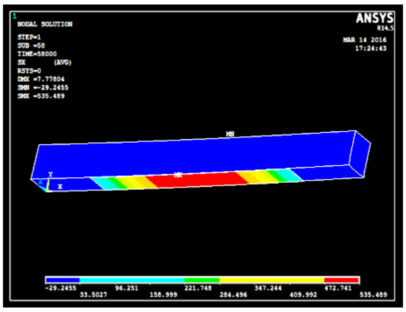

Fig. (11) : Yielding of Steel plate for ANSYS Group-1 ,MPa

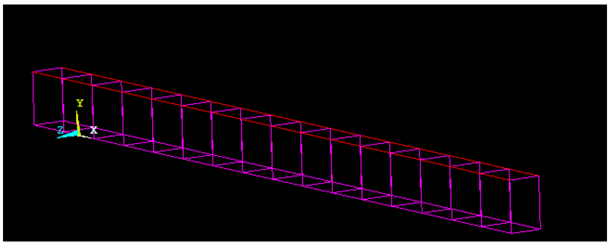

Fig. (8) : Meshing of reinforcement.

Fig. (12) : Cracking of Beam for Group-3 [14].

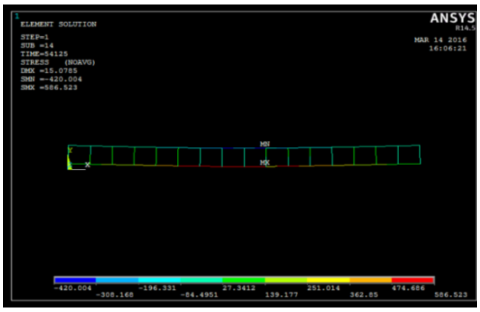

Fig. (9) : Yielding of tensile Steel bars for ANSYS control-1, MPa

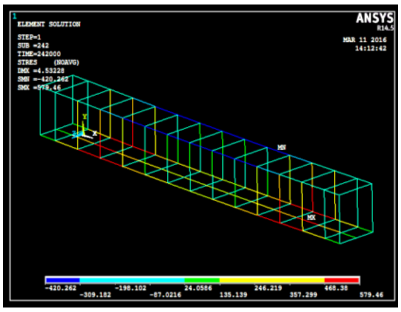

Fig. (13) : Yielding of Stirrups for ANSYS Group-3, Mpa

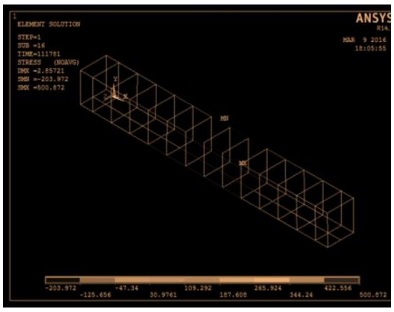

Fig. (10) : Yielding of tensile Steel bars for ANSYS control-2 , PMa

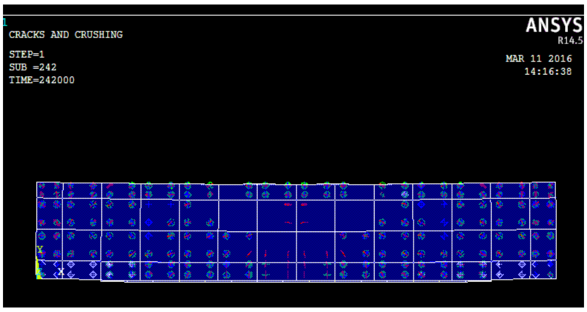

Fig. (14) : Cracks of Strengthened beam of Group-3.

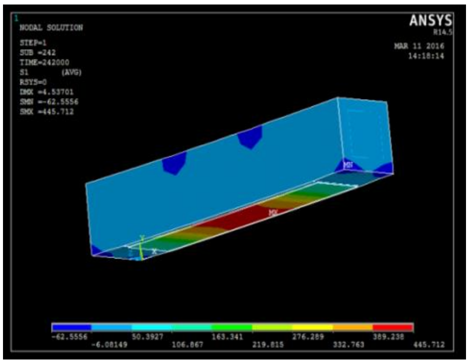

Fig. (15) : Stress contour of ANSYS strengthened beam of Group-3.

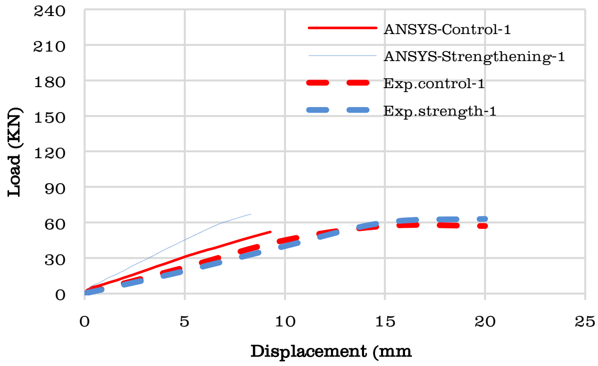

Fig. (16) : Load-Displacement curve for B1 and B2 and ANSYS beams.

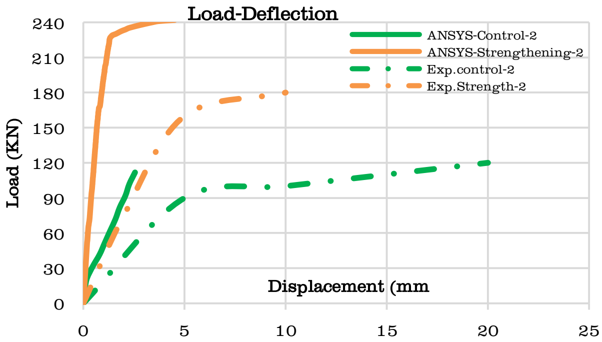

Fig. (17) : Load-Displacement curve For B11 and B12 and ANSYS beams

Table (4) : Experimental and ANSYS results

Group	Beam	Experimental Ultimate Load(kN)	ANSYS Ultimate load (kN)	ANSYS load/Expload	Type of failure	
					Exp.	ANS
1	Control-1(B1)	58.89	55.13	0.94	Flex.	Flex.
	Strengthened with 1mm plate (average)	65	59	0.91	Flex.	Flex.
3	Control-2(B11)	106.6	111.8	1.05	Flex.	Flex.
	Strengthened with 3mm plate (average)	195.6	242	1.23	Shear	Shear