Scientific Research and Essays Vol. 5(6), pp. 533-544, 18 March, 2010
Available online at http://www.academicjournals.org/SRE
ISSN 1992-2248 © 2010 Academic Journals

Full Length Research Paper

Finite element analysis of the beam strengthened with prefabricated reinforced concrete plate

Alper Büyükkaragöz

Civil Engineering Department, Engineering Faculty, Kırıkkale University, 71451, Kırıkkale, Turkey.
E-mail: akaragoz@kku.edu.tr, akaragoz09@yahoo.com. Tel: +90 318 357 42 42 Fax: +90 318 357 24 59.

Accepted 17 February, 2010

Various strengthening methods were frequently carried out in the world to strengthen weaker cross section beams. In this study, one beam strengthened by bonding with a prefabricated plate which has 80 mm thickness underneath and one control beam were produced. The specimens were tested in the laboratory and a single load was applied on the middle of the beam. The results of the experiments were compared with the results obtained from the beam modelled with ANSYS finite element program. When the results of the experiments were compared with the modelled computer program, it was shown that the results of computer model gave similar results to the real behaviour.

Key words: Beam, prefabricated plate, finite element method, ANSYS.

INTRODUCTION

For reinforced concrete, improvement of calculation methods and analysis of behaviour by either creating a model on computer or counting with analytical calculation methods are used extensively in recent years. The behaviour of a reinforced concrete element is generally observed by conducting experiment on laboratory environment; but this process considerably takes up much time. The studies are limited due to the problems in providing the materials and the proper conditions to conduct the experiments and scarcity of usage of materials which are constituted according to certain size and number of elements. Modelling of all these processes unlimitedly in computer is dependent on the capacity of the computer being used. While modelling on the computer, properties and limit conditions of materials should be defined properly and completely.

ANSYS finite element program is chosen for this study. Finite element method is a numeric method which can solve complex and difficult physical problems (which engineers often come up with) with acceptable approximation. ANSYS program based on finite element method has been developed in such a way that it finds itself a bigger practice field since 1971 (Lawrance, 2002).

As concrete is a material showing nonlinear behaviour during loading, it is modelled in such a way that it will show a nonlinear behaviour with ANSYS finite element program (Barbosa and Riberio, 2004).

Finite element method is a numeric method used by different engineering branches in order to solve the problems of engineering requiring special analyses such as stress analysis. The first date of finite element method usage is as old as 1900s. After Boeing used this method on airfoils in the 1950s, it was accepted by the other engineering branches in the 1960s. The first book on the finite element method totally was broadcast by Zienkiewicz and Cheung (1967). In literature there are a particular number of studies that have performed reinforced concrete modelling with ANSYS.

Barbosa and Riberio (2004) studied on the subject of nonlineer modelling of reinforced concrete construction elements in ANSYS finite element program. In their study two different modelling were made for the same beam. In the first model while concrete was defined with solid65 element, link8 bar elements were used for reinforcement (discrete reinforcement). In the second model, the reinforcements were defined with volumetric proportions in the structure of solid65 elements (smeared reinforcement). Both models were analysed four times. In all analyses the results of the load-displacement curves were very similar at low cycling.

In his study Fanning (2001) modelled reinforced concrete beams which have two seperated bars in ANSYS finite element program; one of them was 3.0 m spaced classically and the other one is 9.0 m spaced pre-stressed and compared the results that were obtained with the experimental results. Solid65 and link8 elements were used respectively for concrete, steel and pre-stress cables. Strains of concrete and steel were accepted as same.

Arnesen et al. (1979) developed two programs in his study. In the first one plane stress problem was evaluated and plasticity theory was used. In this program a two-dimensional beam was analysed. The data of this beam was taken from Bresler-Scordelis beam. For modelling concrete, two-dimensional plane elements having four node points and four rebar, bar elements having two node points were used. Under the pressure, the behaviour of concrete material was defined as linear elastic up to the elastic proportionality limit defined by Von Mises ellipse and after that point it was defined as linear plastic. Also the behaviour of reinforcement was defined as plastic after yielding point.

Faherty (1972) analyzed reinforced concrete and prestressed concrete beams using a finite element method. In the study, the linear behavior of reinforced concrete and prestressed concrete beams, anchorage slip, bilinear steel properties and crack progress in the concrete were examined. Furthermore, Faherty examined the prestressing strength, elastic prestressing loss and tension stress caused by the increase in load over concrete in prestressed concrete beams.

Hemmaty et al. (1993) considered a nonlinear adherence-shear law based on the experimental studies between concrete and reinforcement in the modelling of reinforced concrete elements. While modelling adherence-shear relationship, they stated that it can be benefitted from material models and they performed the main modelling with COMBIN39 spring elements. Consequently, the crack results obtained from four different material models defined in ANSYS for spring element were compared with the cracks that occured in the specimens on which tension experiment was carried out and it was determined that in reinforced concrete, reinforce-ment adherence can be modelled realistically by using appropriate adherence-shear law.

Willam and Tanabe (2001) carried out a finite element analysis of reinforced concrete structures concerning seismic behavior of structures, cyclic loading of reinforced concrete columns, shear failure of reinforced concrete beams and concrete steel bond models.

Kachlakev et al., (2001) studied beams externally streng-thened with reinforced plastic carbon fiber (CFRC) with no stirrups being used in the experiment.

Shing and Tanabe (2001) studied the inelastic behavior of reinforced concrete structures under seismic loads. This study contains applications of the finite element method of reinforced concrete columns, the analysis of reinforced concrete components in bridge seismic design and the modelling of the shear behavior of reinforced concrete bridge structures.

Wolanski (2004) modelled reinforced concrete and prestressed concrete beams. During the first step of this study, reinforced concrete model was tested and the fracture loads of bending and shearing reinforcement on the beam were determined through the model. The results obtained from the reinforced concrete model were applied to the prestressed beam.

Ellobody et al. (2006) studied on the behavior and design of axially loaded concrete-filled steel tube circular stub columns. The external diameter of the steel tube-to-plate thickness ranged from 15 to 80 mm covering the compact steel tube sections. The results obtained from the finite element analysis were verified against experimental results.

Pham et al. (2006) studied the bond characteristic between CFRP and concrete. The behavior of 12 shear-lap specimens was modelled using a combination of smeared and discrete cracks. The Rankine's and nonlinear fracture mechanics, where both modes I and II fractures are accounted, failure criteria were used for the smeared and discrete cracks, respectively. The smeared crack model is based on Rankine's failure criterion, whereas the discrete crack model is based on nonlinear fracture mechanics.

Ellobody and Young (2006) used an accurate nonlinear finite element model to analyse the behavior and design of axially loaded concrete-filled square hollow section (SHS) and rectangular hollow section (RHS) steel tube columns. The nonlinear material models for confined concrete and steel tubes were carefully modelled in the finite element analysis.

Buyukkaragoz and Dogan (2007) modelled beams produced by using 5 mm thick sandwich panels in finite element program and they compared the obtained results with the experimental results. In the end of the study, they found out that the strength of beams is proportional to the strength of shear studs.

In this study, an analysis of the column strenghtened by prefabricated plate is conducted for the first time with ANSYS program that has begun to be used scarcely in the field of civil engineering. It is thought by the author that this study will be a guide for the further studies.

In this study SOLID65 element is chosen for concrete modelling with ANSYS program and the founded results are compared with the results of experiments. Discrete reinforcement technique is used as the rebar definition method. In the literature reinforcement modelling is generally defined as discrete and smeared. While the reinforcement is defined as the percentage inside SOLID65 element in the smeared modelling, it is constituted from nodes that construct meshes and element definition is made in the discrete modelling (Kachlakev et al.,, 2001). In order to make the concrete show a nonlinear behaviour, the stress-strain values derived from Hognestad Concrete Model were calculated and entered as data in the program (Ersoy and Özcebe,

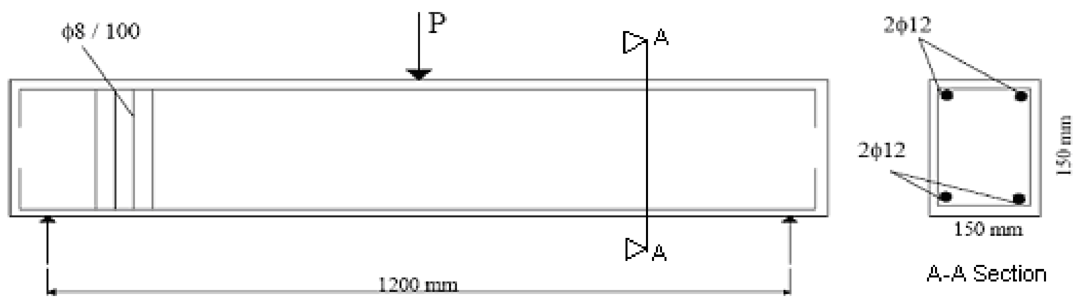

Figure 1. Reinforcement detail of control specimen.

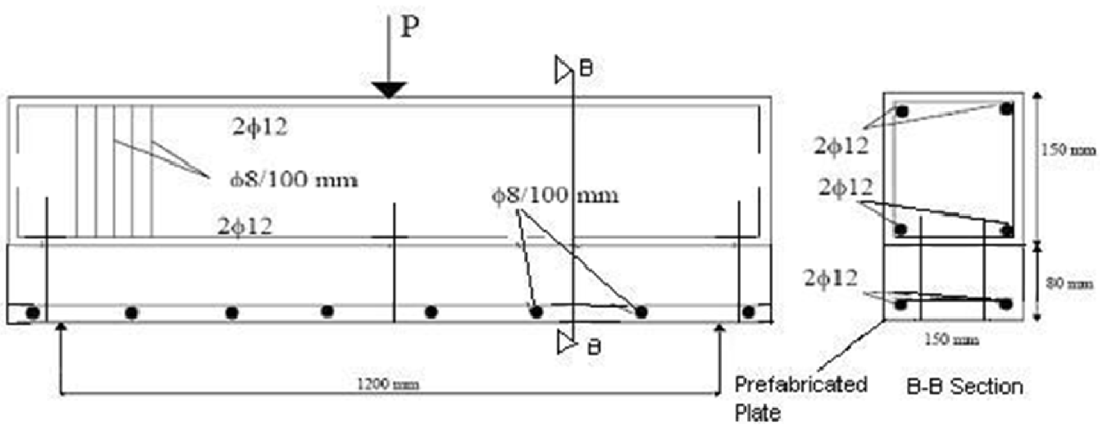

Figure 2. Reinforcement detail of strengthened specimen.

2001). Modulus of elasticity, poisson ratio, yield strength are defined and entered as data in the elastic-full plastic material model for rebar.

MATERIALS AND METHODS

Experimental study

Tekin et al. (2007) modelled two specimens in their study. Control beam element and reinforcement details of beam element strengthened by prefabricated plate are given in Figure 1 and Figure 2. Prefabricated plate is bonded to the beam with epoxy in strenghtened specimen. Being 1200 mm between supports, 150 150 mm cross-sectional specimens are tested in a steel frame designed to enable vertical load implementation. During the experiment the loading is performed with a hydraulic jack which has 500 kN pressure capacity. The obtained results are saved in the computer with a load gauge. The properties of the reinforcement bars, avarage concrete strengths of the specimens and epoxy are presented in Tables 1, 2 and 3 respectively.

The finite element model

The material models used for transferring material properties of the specimen to the designed computer program are given in this section. ANSYS (Lawrance, 2002) finite element program is used for modelling. Material properties of concrete, reinforcement and adhesive are defined and entered as data in the model. Some criteria given below are based on to define the material properties:

Discrete modelling is used for reinforcement steel by defining the element between the nodes in the preformed meshes.
Strain ratio between the concrete and steel is supposed to be equal, therefore it is accepted that there is a unique adherence between the concrete and steel. Due to this, an extra spring-like adherence element is not defined between concrete and steel.

Concrete

In this study, Hognestad concrete model was used due to lack of confinement for the concretes (Hognestad, 1951). The stress–strain values obtained from this model were used in the definition of the multilinear isotropic model. In addition the Willam–Warnke failure model (Willam and Warnke, 1974) used in the definition of the concrete. In the Hognestad concrete model, the part of stress–strain curve until the peak considered to be parabolic in the second degree; and the downward part considered to be linear. In the model, the formula for the parabola of the curve until the peak is given in Equation (1) and the maximum deformation is given in Equation (2). Where $f'c$ is the ultimate compressive strength; σ_c the compressive strength at i^{th} point; ε_{co} the strain at the ultimate compressive strength $f'c$; ε_c the strain at the σ_c

Table 1. Yielding and tensile strength of reinforcements.

Diameter (mm)	Class	Yield strength (MPa)	Ultimate strength (MPa)
ϕ 8	S420a	565	670
ϕ 12	S420a	512	590

Table 2. Compressive strengths of specimens.

Specimen	Cylinder diameter (mm)	Axial load (kN)	Compressive strength (MPa)
Control	150	640	28.4
	150	650	28.8
	150	670	29.8
Strengthened	150	690	30.7
	150	680	30.2
	150	650	28.8

Table 3. Properties of epoxy.

Name	Compressive strength (MPa)	Bending strength (MPa)	Concrete stick (MPa)	Modulus of elasticity (MPa)	Density (MPa)
CX 31 Ceresit	75	15	5	10000	0.18

compressive strength; E_c is the modulus of elasticity of the concrete:

$$\sigma_c = f'c\left[\frac{2\varepsilon_c}{\varepsilon_{co}} - \left(\frac{\varepsilon_c}{\varepsilon_{co}}\right)^2\right] \tag{1}$$

$$\varepsilon_{co} = \frac{2f'c}{E_c} \tag{2}$$

According to the multilinear isotropic model suggested by Von Mises (Wolanski, 2004), the behavior of the concrete in the principle stressing space with three axis is defined in Equation (3). Where σ_y is the threshold stress in passing from elastic to plastic behavior; σ_e is the equivalent stres less than treshold stress and σ_1, σ_2 and σ_3 are principal stresses.

$$\sigma_{y\geq}\sigma_e = \left[\frac{1}{2}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]\right]^{\frac{1}{2}} \tag{3}$$

The Willam–Warnke failure model is used for modelling the failed collapsing surface of concrete without reinforcement under stress with three axis. If the calculated principle stress is more than threshold stress, its behavior is considered to be nonlinear. In this case, the calculated principle stresses are used to determine the failure situation using the Willam–Warnke model. If Equation (4) is obtained using these principals, it means that stresses occur on the failure surface (William and Warnke, 1974). Where σ_a and τ_a are average stress components, z is the apex of the surface and f_c is the uni-axial compressive strength, r is the position vector locating the failure surface with angle θ :

$$\frac{1}{\rho}\frac{\sigma_a}{f_c} + \frac{1}{r(\theta)}\frac{\tau_a}{f_c} = 1 \tag{4}$$

The use of the Willam–Warnke mathematical model of the failure surface for the concrete has the following advantages (Özcan et al., 2009):

Close fit of experimental data in the operating range.
Simple identification of model parameters from standard test data.
Smoothness (continuous surface with continuously varying tangent planes).

Convexity (monotonically curved surface without infection points).
Solid65 element is used for the concrete model in the reinforced concrete beam model. The choice of Initial Stiffness is used for Solid65 element type in ANSYS finite element program. The Initial Elastic Stiffness value is used for each iteration in Initial Stiffness approach. Solid65 element is a solid with eight joints having the properties of cracking under tension, crashing under compression, plastic deformation and creeping, which is defined for concrete and

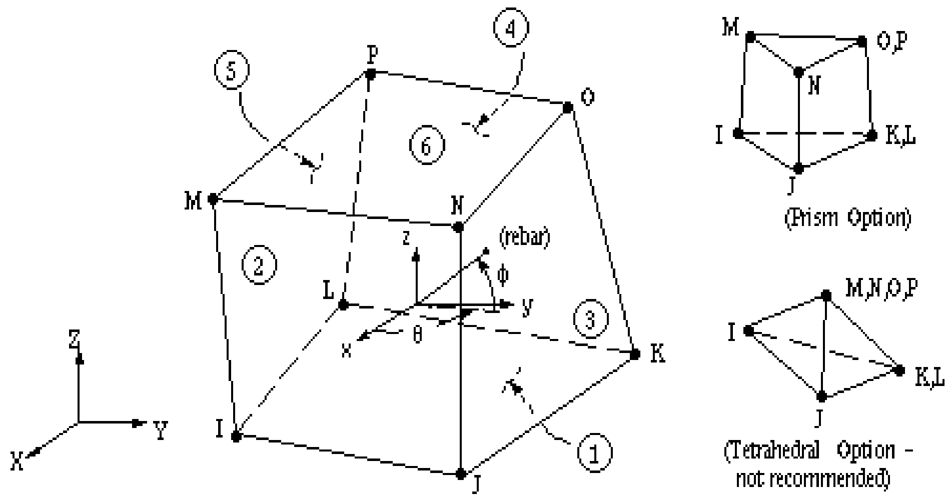

Figure 3. Solid65 element.

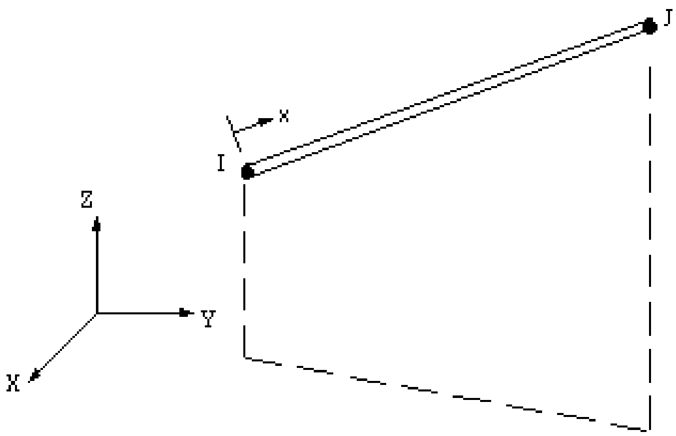

Figure 4. Link8 element.

reinforced concrete elements (Figure 3). The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y and z directions. Up to three different rebar specifications may be defined.

Steel reinforcement

The steel is a homogeneous and isotropic material which can be defined more easily and closer to real than concrete. Unlike concrete, its properties do not depend on environmental conditions and time. In this study the behaviour of the steel is idealized, the steel material is defined as Bilinear Isotropik based on Von Mises yielding criteria.

Link8 element is used in order to define the reinforcement that present in ANSYS finite element program (Figure 4). Link8 is a spar which may be used in a variety of engineering applications. Depending upon the application, the element may be thought of as a truss element, a cable element, a link element, a spring element, etc. The three-dimensional spar element is a uniaxial tension-compression element with three degrees of freedom at each node: translations in the nodal x, y and z directions. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, swelling, stress stiffening and large deflection capabilities are included.

Adhesive

In the model, epoxy is used to bond the prefabricated plate to the beam. Solid46 element is used for epoxy in the program (Figure 5). SOLID46 is a layered version of the 8-node structural solid (Solid45) designed to model layered thick shells or solids. The element allows up to 250 different material layers. If more than 250 layers are required, a user-input constitutive matrix option is available. The element may also be stacked as an alternative approach. The element has three degrees of freedom at each node: translations in the nodal x, y and z directions.

The real constant set defined for a total two different kinds of reinforcement (one stirrup and one longitudinal reinforcement) are given in Table 4.

Reinforcements and stirrups used in the specimen are modelled with discrete method by constituting element definition from the mesh nodes constructing the concrete. The model views of modelled reinforcements and stirrups are given in Figure 6 and Figure 7.

In order to obtain good results from the Solid65 element, the use of a rectangular mesh is preferred. The reference beam with 1500 mm length and 150 × 150 mm cross-section is divided into a total of 4800 rectangular prism shaped meshes. The strenghtened beam having 1500 mm length and 230 × 150 mm cross-section is divided into a total of 7200 rectangular prism shaped meshes. Single load is applied to the upper middle surface of the experiment element from 9 points. The beam is modelled as to be supported at two parts. The supports and the applied loads of specimens that are modelled are seen in Figures 8 and 9.

In Table 5, the material properties of the concrete and reinforcement materials that are defined in ANSYS finite element program are given. For concrete; the modulus of Elasticity and Poisson Ratios are defined under the title of Linear Isotropic. Also for concrete, stress-strain values (σ-ε) are defined by using Equation (1) and Equation (2) formulas for calculation under the title of Multilinear Isotropic. Open and closed crack shear transfer coefficients and concrete's tensile strength are defined under the title of concrete (Kachlakev et al., 2001). For reinforcement, the yield strength of reinforcement are defined under the title of Bilinear

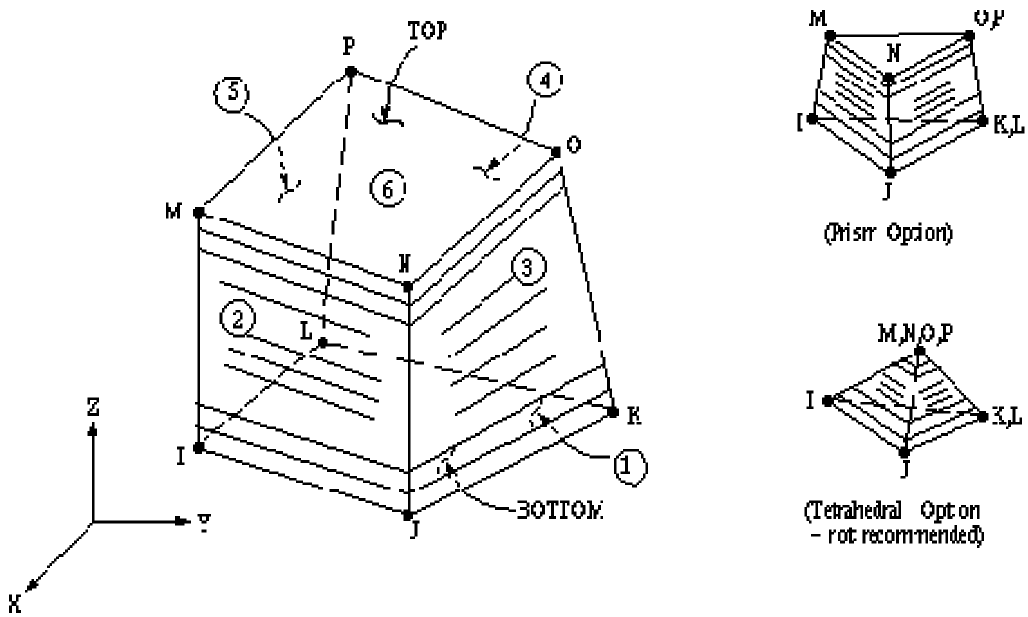

Figure 5. Solid46 element.

Table 4. Real constant set of Link8.

Real constant number	Element type	Diameter (mm)	Area (mm^2)
1	Link8	$\phi 8$	50
2	Link8	$\phi 12$	113

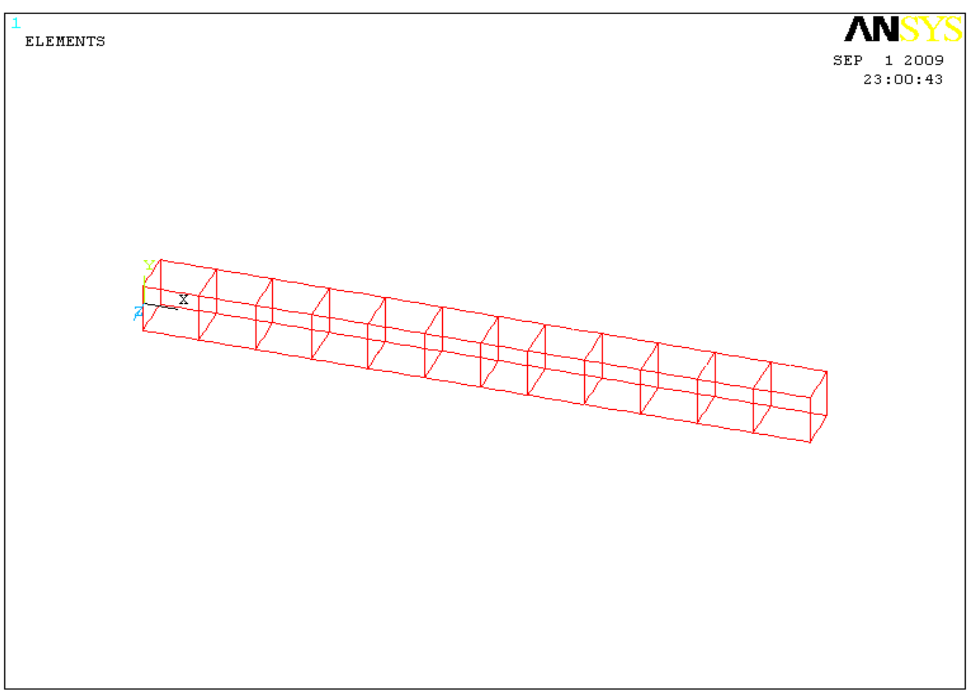

Figure 6. Reinforcements and stirrups modelling of control specimen.

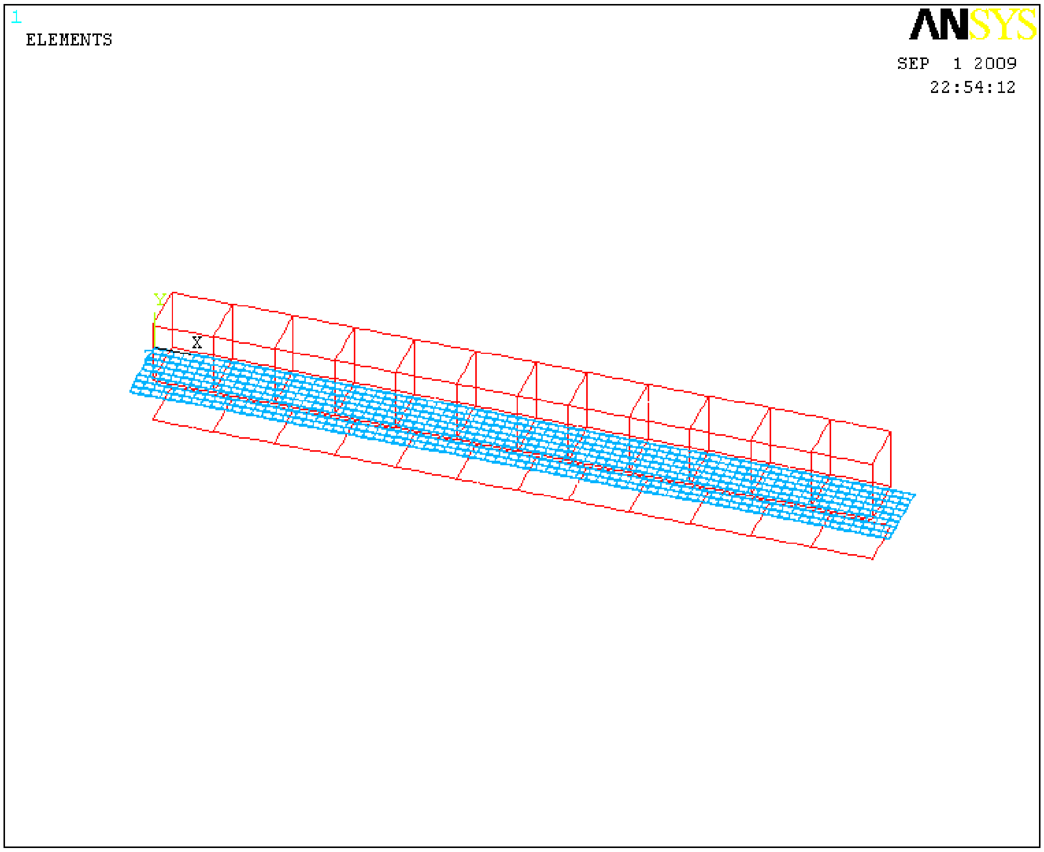

Figure 7. Reinforcements,strirrups and epoxy modelling of strengthened specimen.

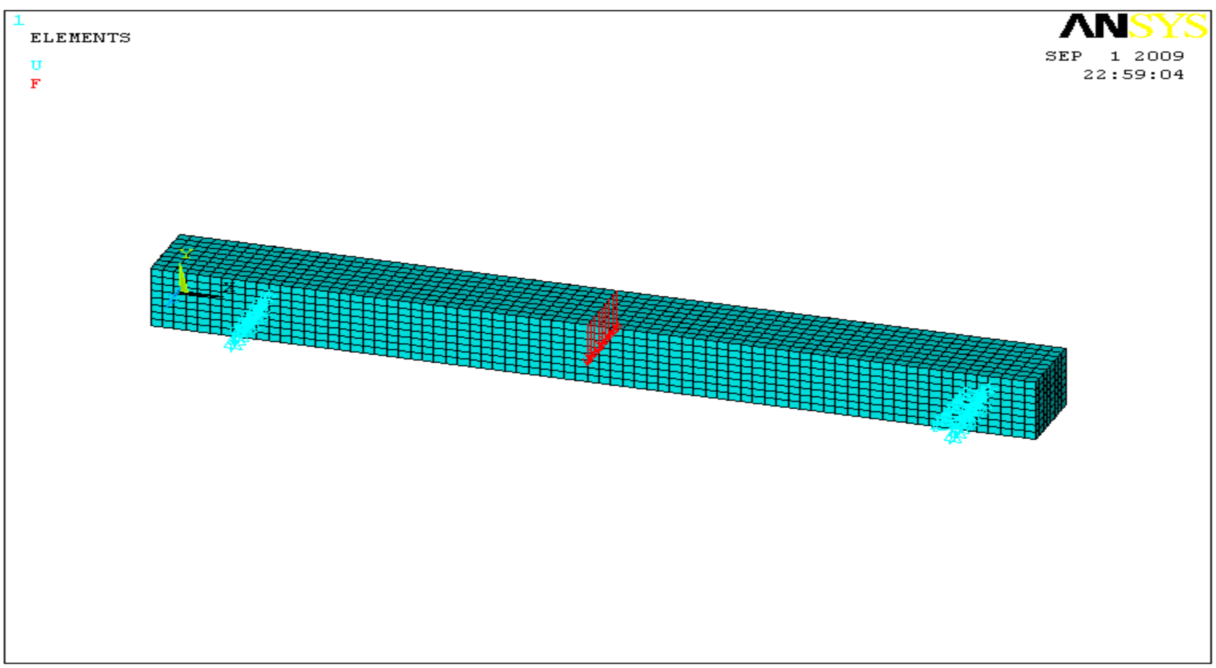

Figure 8. Loads and supports of control specimen.

Table 5. Properties of materials.

Material number	Element type	Material properties		
1	Solid65	**Linear ısotropic**		
		EX(MPa)	24019	
		PRXY	0.3	
		Multilinear elastic		
			Strain	**Stress (MPa)**
		1. Point	0.00035	7.54
		2. Point	0.00067	13.22
		3. Point	0.00090	16.60
		4. Point	0.0017	23.76
		5. Point	0.0021	24.65
		Concrete		
		ShrCf-Op.	0.4	
		ShrCf-Cl.	1	
		UnTensSt. (MPa)	1.98	
		UnCompSt. (MPa)	-1	
		BiCompSt.	0	
		HydroPrs.	0	
		BiCompSt.	0	
		UnTensSt.	0	
		TenCrFac	0	
2	Link8	**Linear Isotropic**		
		EX(MPa)	2,0E+5	
		PRXY	0.3	
		Bilinear Isotropic		
		fy (MPa)	430	
		Tan.Mod.	0	
3	Link8	**Linear Isotropic**		
		EX(MPa)	2,0E+5	
		PRXY	0.3	
		Bilinear Isotropic		
		fy (MPa)	322	
		Tan.Mod	0	
4	Solid46	**Linear Isotropic**		
		EX(MPa)	10000	
		PRXY	0.35	
		Density		
		DENS(MPa)	0.18	

Isotropic, Modulus of elasticity and Poisson ratios are defined under the title of linear ısotropic. For epoxy; linear ısotropic material properties and density are defined.

RESULTS AND SUGGESTIONS

The ultimate load and moment values obtained from the

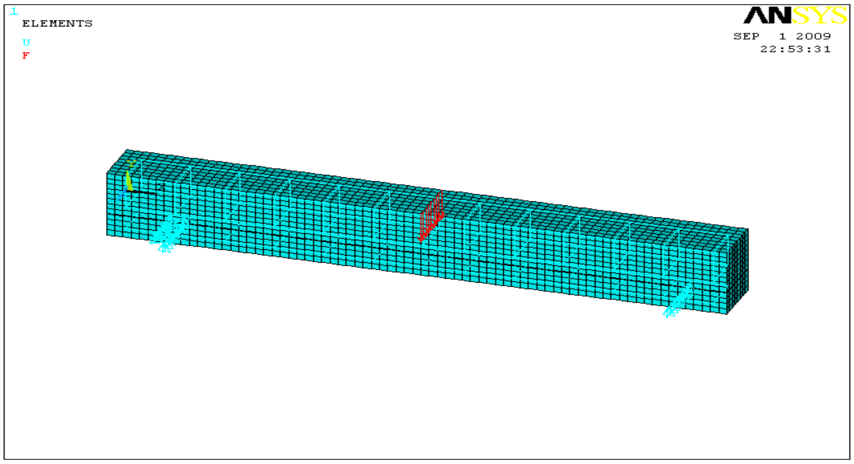

Figure 9. Loads and supports of strengthened specimen.

Table 6. Comparison of analytical and experimental results.

Specimens	Ultimate load (kN)			Moment capacity (kNm)		
	Experimental	Analytical	Ratio (%)	Experimental	Analytical	Ratio (%)
Control	52.7	47	12	15.8	14.1	12
Strengthened	70	66	6	21	19.8	6

experiment results and finite element analysis are compared in Table 6 and the similar values are achieved. It was observed that the control specimen collapsed at 52.7 kN load level at the end of experimental loading and the strengthened specimen collapsed at 70 kN load. The prefabricated plate that was bonded with epoxy to the underneath part of the control specimen increased the load carrying capacity of the section by 32%. The control specimen collapsed because of bending and it showed a flexural behaviour. The beam collapsed because of bending in strengthened specimen whereas the prefabricated plate showed a sudden brittle shear failure because of the horizontal bonding of the stirrups to the reinforcements and its unextension to the upwards. Transverse reinforcement should extend upwards like stirrup. For this, cross-section of the prefabricated plate should be increased and compression reinforcement should be placed on the top. In this case as the volume and weight of the plate would increase and hence bonding with epoxy will be quite hard.

In the finite element analyses, pointed load was applied on the junction point of nine finite areas in the negative direction of y-axis as the upper surface of the beam was divided into eight finite fields throughout the depth of the beam. The first loading was started with a 900 N load, afterwards the analyses were continued with increasing the load by 900 N in each load step. In the control specimen, the first bending crack occurred on sub-front face of the beam in the loading direction and at 19 kN load. At 21 kN load level, new capillary cracks occurred on sub area of the beam. At 22 kN load level, a new bending crack occurred on 2 cm left side of the first bending crack. At 31 kN load level, numbers of bending cracks increased and the first bending crack expanded upwards making 90° angle with horizontal till the middle of the beam. At 36 kN load level, the crack that occurred in the front left face of the beam in the last left side moved upwards making 45° angle with horizontal. At 43 kN load level, the first bending crack reached to the surface of the beam and crashing begun on the concrete at the loading area. The maximum load level was reached at 47 kN and displacement in the direction of y-axis significantly increased and the specimen collapsed because of bending moment. A 12% difference was observed between the loads obtained from the analysis and the experiment. In the strengthened specimen, the first bending crack occurred on sub-front face of the beam in the direction of load at 27 kN load. At 35 kN load level, the numbers of the cracks increased. At 54 kN load level, a new bending crack occurred from 20 cm left side of load direction and it expanded 10 cm towards point load was applied.

When the load was increased at 66 kN, displacement in the direction of y-axis significantly increased. At this load level, while the beam element collapsed by bending, the plate collapsed by a brittle shear failure. The crack and stress distributions during failure derived from the finite element analysis are given in Figures 10, 11, 12 and 13 respectively. In Figure 14, the view of the strengthened specimen at the end of the experiment is shown.

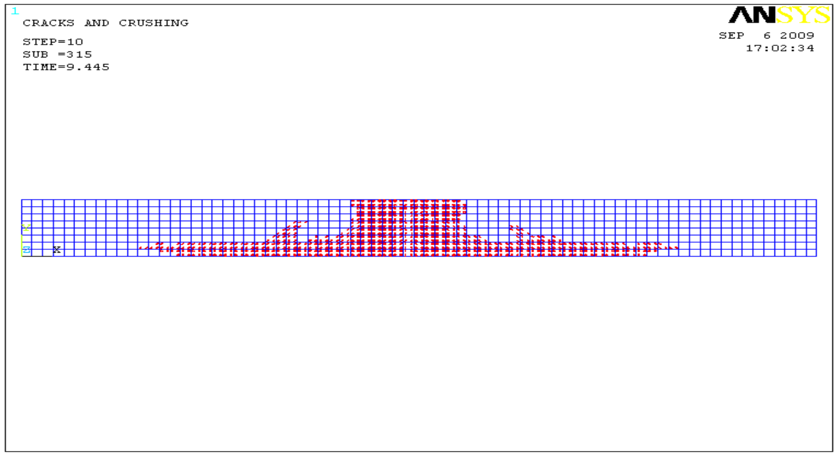

Figure 10. Cracks of control specimen at failure.

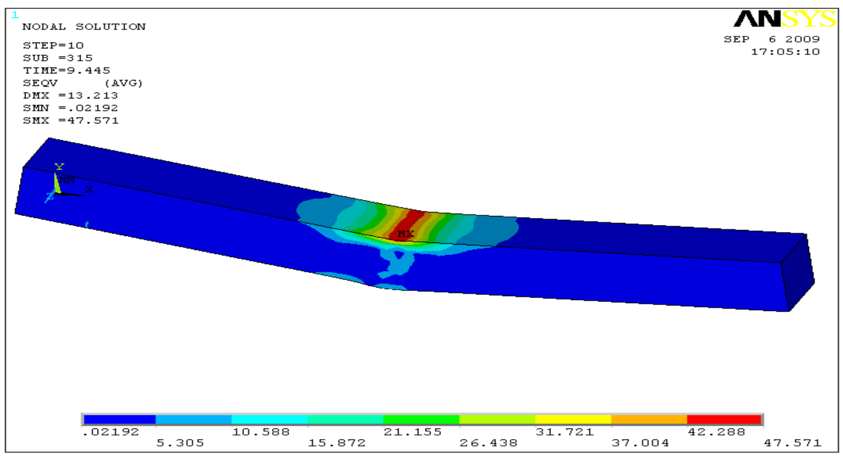

Figure 11. Stress of control specimen at failure.

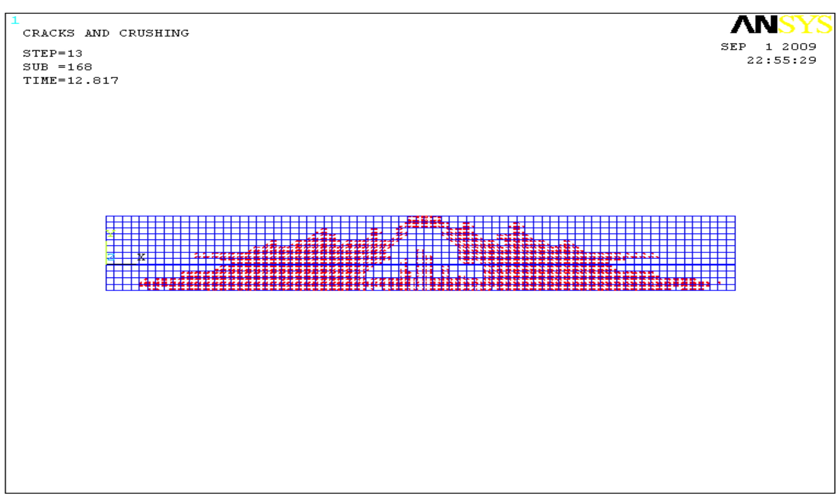

Figure 12. Cracks of strengthened specimen at failure.

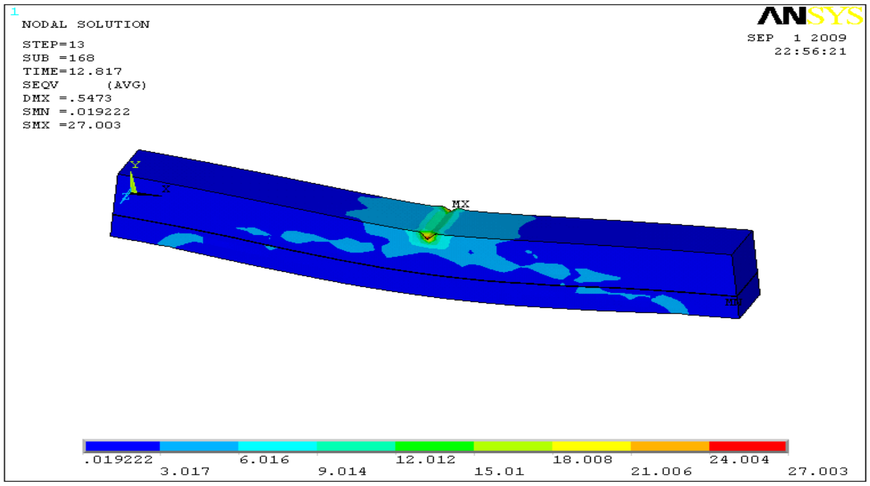

Figure 13. Stress of strengthened specimen at the failure.

Figure 14. Picture of strengthened specimen after failure.

In summary, the results obtained from the experimental behaviour of the beams were compared with the results obtained from the model which was constructed with the ANSYS finite element program. It was observed that the results are quite similar to each other.

The possible reasons of why the results obtained from the specimen and the model element are partially different from each other are listed below.

Due to the feature of its material, concrete has a heterogeneous nature so that it does not show the same material features under all directions. However, in the finite element model when the material feature is defined, the modelling is carried out accepting that concrete has a homogeneous mixture under all directions.

After the model is formed, the supports are defined most properly from each point consisting meshes. Mean-while, the defects in the mechanism of loading affect the supporting status in the experiment.

Stress-strain behaviour of the concrete is obtained using the formulas defined in Hognestad concrete model. Nonetheless, the properties of the concrete influence the stress-strain values in the experiment.

In order to conduct a more sensitive analysis, Solid65 element constituting the concrete can be divided into a greater number of elements during modelling. For this purpose, computers with a broader capacity can be used. What kind of a behaviour that the beam strengthened with bonding a prefabricated plate will show is generally observed by conducting an experiment on laboratory environment; but this process considerably takes up much time. The studies are limited due to the problems in providing materials and proper conditions to conduct experiments. Also, the insufficient number of materials that are constituted according to certain size and number of elements is another factor for the difficulties in experimental studies.

Modelling of all these processes unlimitedly in computer is dependent on the capacity of the computer being used. It was observed that the results obtained from ANSYS finite element program are considerably correlated to the results of the experiment. This shows that the modellings that are made with ANSYS finite element program give us reliable results similar to the previous reports in the literature. In conclusion the modellings that are made with ANSYS finite element program can be useful for saving money and time in terms of the specimen. Also, the design errors which can be made in the design stage or wrong material selection can be prevented. The auhor believes that this way of modelling will be a guide for the further experimental studies.

REFERENCES

Arnesen A, Sorensen SI, Bergan PG (1979). Nonlinear analysis of reinforced concrete. Compt. Struct. 12: 571-579.

Barbosa AF, Riberio GO (2004). Analysis of reinforced concrete structures using ANSYS nonlinear concrete model. Comput. Mech. 1(8): 1-7.

Buyukkaragoz A, Dogan O (2007). Çift cidarlı kompozit kirişlerin sonlu elemanlar yöntemiyle gerilme analizi'', Pamukkale Üniv. Müh. Bil.Der. 13(1): 39-45.

Ellobody E, Young B (2006). Nonlinear analysis of concrete filled steel SHS and RHS columns. Thin Walled Struct. 44: 919-930.

Ellobody E, Young B, Lam D (2006). Behavior of normal and high strength concrete filled compact tube circular stub columns. J. Construct. Steel Res. 62:706-715.

Ersoy U, Özcebe G (2001). Reinforced Concrete, Evrim Yayınevi, İstanbul, Turkey. p. 59.

Faherty KF (1972). An analysis of a reinforced and a prestressed concrete beam by finite element method. PhD dissertation, University of Iowa, Iowa, USA.

Fanning P (2001). Nonlinear models of reinforced and post-tensioned concrete beams. Elect. J. Struct. Eng. 2: 111-119.

Hemmaty Y, De Roeck G, Vandewalle L (1993). Finite element modelling of corner joints in reinforced concrete frames: International conference concrete 2000, economic and durable construction through excellence, UK. University of Dundee, Concrete Technology Unit, Dundee. pp. 465-472.

Hognestad E (1951). A study of combined bending and axial load in R.C. members. Univ. Illinois Eng. Exp. Stat. Bull. 399(2): 36-57.

Kachlakev DI, Miller T, Yim S, Chansawat K, Potisuk T (2001). Finite element modelling of reinforced concrete structures strengthened with FRP laminates. PhD dissertation, California Polytechnic State University, San Luis Obispo,USA.

Lawrance KL (2002). ANSYS Tutorial Release 7.0 and 6.1, SDC Publications, Canonsburg, 1.1- 2.25.

Özcan M, Bayraktar A, Şahin A, Haktanır T, Türker T (2009). Experimental and finite element analysis on the steel fiber reinforced concrete (SFRC) beams ultimate behavior. Construct. and Build. Mat. 23:1064-1077.

Pham H, Al-Mahaidi R, Sauma V (2006). Modelling of CFRP concrete bond using smeared and discrete cracks. Compos. Struct. 75:145-150.

Shing PB, Tanabe TA (2001). Modelling of inelastic behavior of RC structures under seismic loads. Dearborn, MI: American Society of Civil Engineers.

Tekin M, Alsancak E, Demir A, Şeker S (2007). Strengthening of beams with prefabricated reinforced concrete plates: Sixth national conference on earthquake engineering, İstanbul, Turkey.

Willam KJ, Tanabe TA (2001). Finite element analysis of reinforced concrete structures. Farmington Hills, MI: American Concrete Institute.

Willam KJ, Warnke EP (1974). Constitutive model for triaxial behavior of concrete. In: Proceedings of the international association of bridge and structural engineering conference; pp. 174-191.

Wolanski AJ (2004). Flexural behavior of reinforced and prestressed concrete beams using finite element analysis. Master thesis, Marquette University, Wisconsin, USA.