Pakistan Journal of Science (Vol. 66 No. 1 March, 2014)

IDENTIFICATION OF SHEAR CRACKS IN REINFORCED BEAMS USING FINITE ELEMENT METHOD (ANSYS)

H.U. Khan, M.N. Rafique, S. Karam, K. Ahmad and A. Bashir

Department of Civil Engineering, University of Engineering and Technology Lahore, Pakistan
Corresponding Author e-mail: hassaan.ullah.khan@hotmail.com

ABSTRACT: Analytical determination of displacements and stresses in reinforced concrete material was difficult task and engineers had to rely on empirical formulas because concrete consists of heterogeneous material and creep and shrinkage influenced deformations in it. Due to these complexities engineers in past had been facing difficulties in coping such problems, but with the advancement of digital computerization and modern numerical methods for analysis such as finite element method, these problems can be addressed in a very efficient way. There were two ways to carry out modeling in ANSYS software, one was smeared approach and the other one was discrete. In the past, Smeared approach was used to identify the cracks in RC beam using ANSYS but in this work it was extended using discrete approach of modeling and shear cracks were identified in RC beam and load deflection curve was simulated which showed good agreement with the experimental results.

Keywords: shear cracks, static analysis, reinforced beam, non linear analysis, simulation.

INTRODUCTION

Concrete structural elements behave differently under different variety of loading. The identification and calculation of these responses is very laborious and requires lot of expense and time. But now a days there are several techniques available to solve this problem, amongst those indigenous techniques the widely used one is finite element method. Finite element method is a numerical analysis method that divides the structural element into smaller parts and then simulates static loading conditions to evaluate the response of concrete and pre stressed concrete members. The use of this technique is increasing because of enormous advancement of engineering and computer knowledge. This method responds well to non linear analysis as each component possesses different stress strain behavior. This behavior is efficiently addressed by software ANSYS which provides number of elements for modeling of materials and apply loads to evaluate the response. The objective of this study was to make a comparison between experimentally tested RC beam and the one modeled using ANSYS by incorporating discrete approach as suggested by Dahmani , etal(2010). The model beam of Ayman and Banerjee (2007) was taken as the reference beam for our analysis and shear cracks are compared with it as obtained from ANSYS.

MATERIALS AND METHODS

Failure Criteria for Concrete: The model developed using ANSYS is capable of predicting failure for concrete materials. Both cracking and crushing failure modes are accounted for. The two input strength parameters i.e. ultimate uniaxial tensile and compressive strengths are needed to define a failure surface for the concrete. Consequently, a criterion for failure of the concrete due to a multiaxial stress state was calculated by William and Warnke's (1975) constitutive model for multiaxial stresses. Bangash (1989) proposed that in a concrete element, cracking occurs when the principal tensile stress in any direction lies outside the failure surface. After cracking, the elastic modulus of the concrete element is set to zero in the direction parallel to the principal tensile stress direction. Crushing occurs when all principal stresses are compressive and lie outside the failure surface, subsequently, the elastic modulus is set to zero in all directions and the element effectively disappears.

Finite Element Modeling: Experimental RC beam specimen was analyzed by using ANSYS which is an engineering simulation commercially used software offering a comprehensive suite that spans the entire range of physics, providing access to virtually any field of engineering simulation that a design process requires. The software use it's tools to put a virtual product through a rigorous testing procedure such as testing a beam under different loading scenarios before it becomes a physical object. ANSYS can carry out advanced engineering analyses quickly, safely and practically by variety of contact algorithms, time based loading features and nonlinear material models. In this study it used to carry out discrete modeling of RC beam to analyze it under static loading conditions.

Reinforced Concrete: For modeling of concrete the ANSYS used an element named as Solid65 which is non linear model of brittle material similar to concrete. It was an eight node solid iso parametric element with three degrees of freedom at each node.

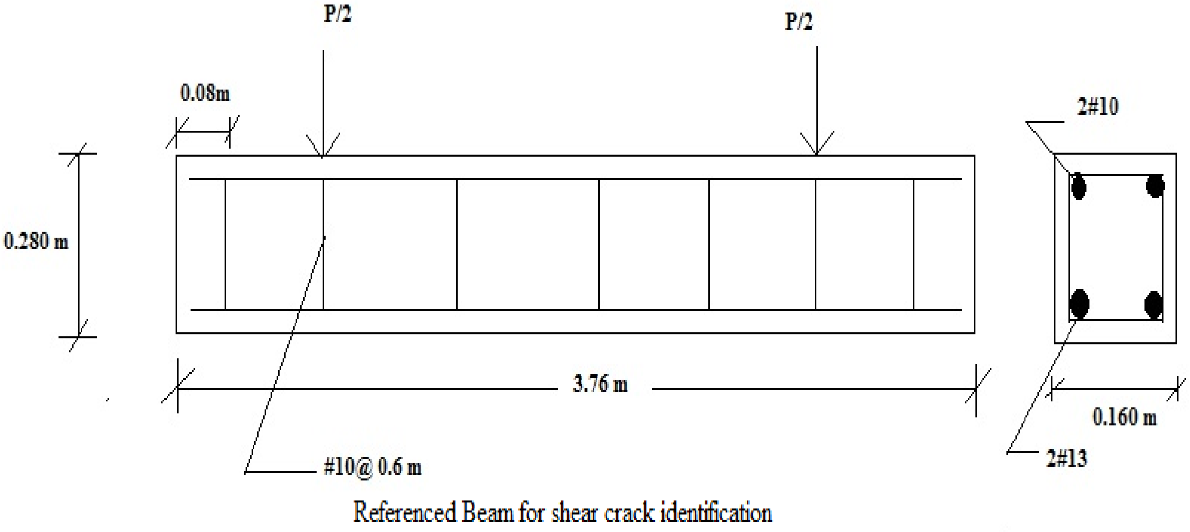


Fig-1 Reference Beam Dimensions

Table-1 Material Properties for FE model in ANSYS

Material	Density (kg/m^3)	Elastic Modulus (N/m^2)	Poison's Ratio	Fc' (e6 N/m^2)	Fy (e6 N/m^2)
Concrete	2400	24644.62e6	0.3	27.54	-
Reinforcing Steel	7850	2e11	0.2	-	280

Table-2 Properties of steel and concrete

Area of main steel	129e-6 m^2 (#13)
Area of hanger	71e-6 m^2 (#10)
28 days compressive strength fc'	27.54e6 N/m^2

Steel Reinforcement: for the modeling of steel ANSYS provided an element named as Link180 There were two ways to use it one was smeared and the other is discrete, discrete was considered to be more convergent as it subtracts the area of steel from total concrete which was the actual scenario where as in smeared the steel was embedded in the concrete and behaved as one unit which was not the actual case.

Experimental Data: Ayman and Banerjee (2007) carried out their experiment showing the average ultimate failure load of as-built specimen as 86 kN. The width and height of the beams tested were 0.160 mm and 0.280m respectively, the length of the beam was 3.76m with supports located 0.08m from each end of the beam as shown in Fig.1. The mild steel flexural reinforcements used were 2#13 bars, 2#10 hanger bars and shear reinforcement included #10 U-stirrups. Cover for the rebar was set to 40mm in all directions.

The experimental failure shown in Fig-2 which is similar to crack obtained through ANSYS in Fig-12

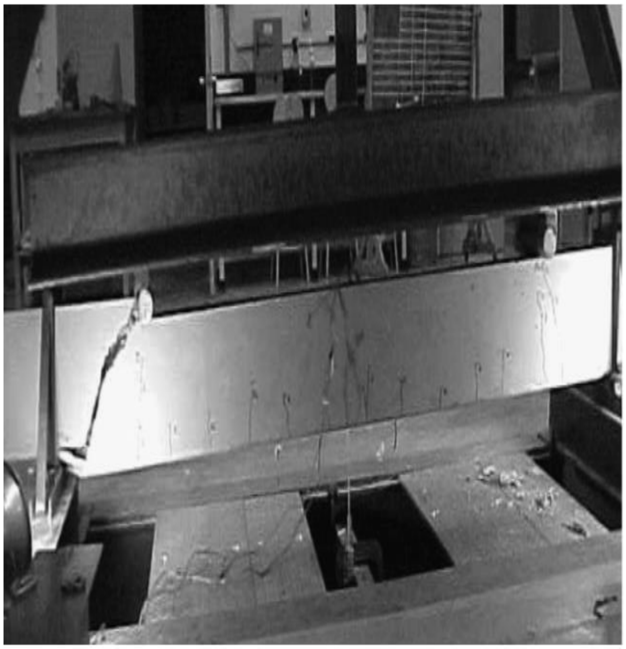


Fig-2 Shear Crack in Experimentally Tested Beam

The compressive uniaxial stress-strain relationship for the concrete model was obtained using implementation of William and Warnke's (1975) constitutive model for concrete in ANSYS which defined 9 different constants as shown in Table-3

Table-3 Shear Transfer Coefficients

1	Shear transfer coefficient for an open crack	0.3
2	Shear transfer coefficient for a closed crack	0.6
3	Uniaxial tensile cracking stress	1.9e6
4	Uniaxial crushing stress (positive)	-1
5	Biaxial crushing stress (positive)	0
6	Ambient hydrostatic pressure	0
7	Biaxial crushing stress (positive) under the ambient hydrostatic stress state	0
8	hydrostatic blax	0
9	Tensile condition factor	0.6

Typical shear transfer coefficient range from 0.0 to 1.0, with 0.0 representing a smooth crack (complete loss of shear transfer) and 1.0 representing a rough crack (no loss of shear transfer). The shear transfer coefficient for open and closed cracks was determined using the work of Kachlakev and Miller (2001) as a basis. The uniaxial crushing stress in this model was based on the uniaxial unconfined strength. It was entered as -1 to turn off the crushing capability of the concrete element as suggested by past researcher Kachlakev and Miller (2001).

Constitutive model for concrete: The following equations were used to compute the multilinear isotropic stress-strain curve for the concrete as presented in Fig-3 MacGregor (1992)

$$f= \frac{E_c\varepsilon}{(1+(\varepsilon/ \varepsilon_o)^2)}$$

$\varepsilon_o= 2f_c^{'}/E_c$

$E_c=f/\varepsilon$

where:

f = stress at any strain ε , psi

ε = strain at stress f

ε_o = strain at the ultimate compressive strength $f_c^{'}$

The multilinear isotropic stress-strain implemented requires the first point of the curve to be defined by the user. It must satisfy Hooke's Law $E= \sigma/\varepsilon$ used by Chote and Barzin (2009)

The following curves are obtained for concrete and steel in ANSYS V13

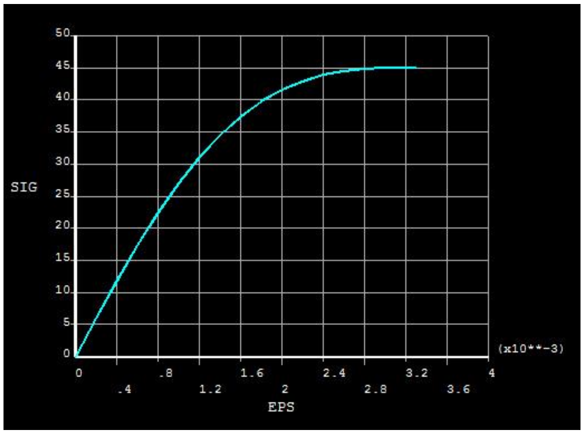


Fig-3 Stress Strain Curve For Concrete

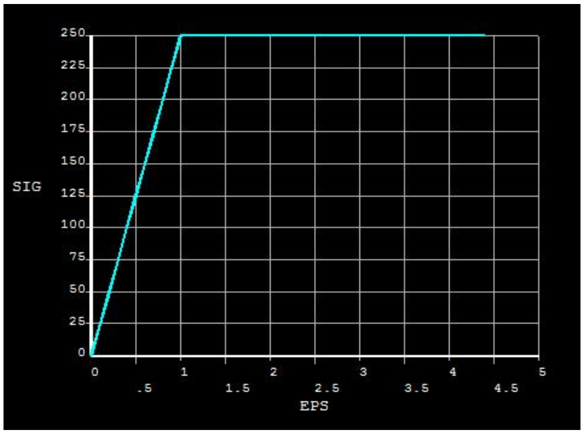


Fig-4 Stress Strain Curve For Steel

Table-4 showing co-ordinates of beam model in ANSYS

Table -4 FE beam

Orientation	**Co-ordinates**
x_1,x_2 ,x co-ordinate	0,0.160 m
y_1,y_2,y co-ordinate	0,0.280 m
z_1, z_2,z co-ordinate	0,1.880 m

By using these co-ordinates the following model is made as shown in Fig-5

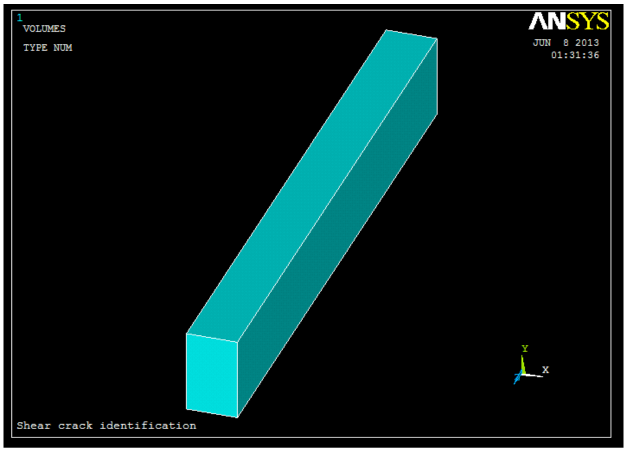


Fig-5 Modeled Beam

After giving material properties for steel and concrete such as elastic modulus Eb , ultimate uniaxial compressive strength $f c$, steel grade, ultimate uniaxial tensile strength (modulus of rupture) $f r$, Poisson's ratio for both steel and concrete, shear transfer coefficient, and compressive uniaxial stress-strain relationship the key points for the bars are specified to have the shear stirrups, main and hanger bars as reinforcement are shown in Fig-6

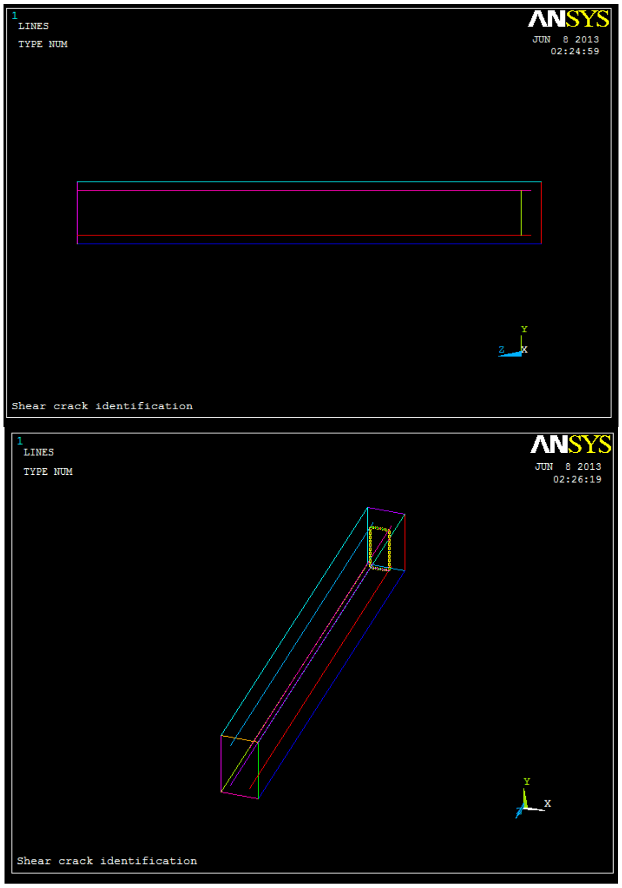


Fig-6 Stirrup and reinforcement

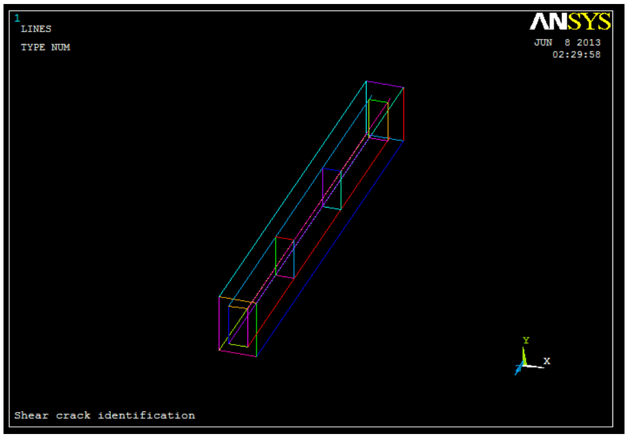


Fig-7 FE Modeled RC Beam Having Stirrups

The stirrup formed using the key point is shown in Fig-6 and is selected and copied at required spacing and at required number of times. So finally the beam is modeled as shown in Fig-7 For the sake of ease due to symmetry in specimen the right half of the beam is modeled that requires less time of processing and the side of the beam in cross section is restrained in the two perpendicular components only the UY is allowed to have the degree of freedom as shown in Fig-8, similarly instead of using metal base plate another concept is utilized which is fixing the nodes that act as support providing zero degree of freedom i.e zero displacement.

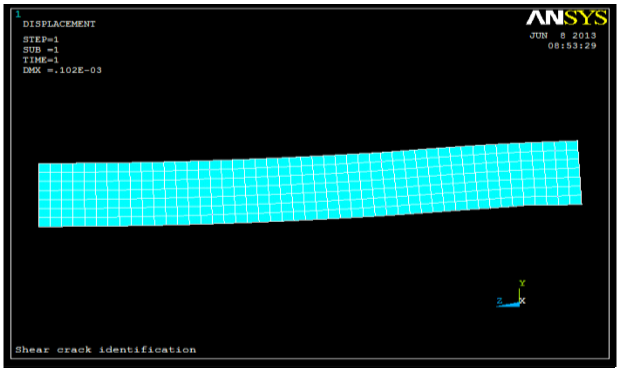
Fig-8 Support and Side Restraints

RESULTS

After meshing the model, the load is applied in increments and at different loads, the deflections are obtained, the stress contour showed the variation of stresses along the member with deformed shape as shown in Fig-9 and initial cracking stress contour in presented Fig-10

ANSYS V13 used to check the response of the structural member under static load by discrete approach gave almost the same ultimate load and shear cracks were identified along with acceptability of ANSYS software to analyze and predict the cracking pattern which was in harmony with experimental results. This study also depicts and explains how to model shear stirrups along with reinforced bars using half model specimen provided it is symmetrical. This not just saves the time of iterations and analysis but also facilitates the user to carry out modeling quickly. The initial cracking load for both FE beam and experimental beam was 18.75 KN and the final cracking load was 86.8KN and 86KN respectively which was in harmony.

Comparison of FE model and Experimental Results: The graph is plotted between load and deflection as obtained through ANSYS and experiment. Results are shown, before the load of 18.75 KN where initial cracks appeared, the deflection was same in both experimental and FE model, but when the initial cracking started the curve showed slight variation. It might be because we have considered a perfect bond between steel and concrete showing more stiffness and no slippage but in actual case there was always certain slippage which might be the reason that FE model was giving lesser deflection and more stiffness. The initial cracking load for both FE and experimental beam was 18.75KN which were in harmony while ultimate load was 86KN for experimental and 86.8KN for FE model which was also satisfactory.



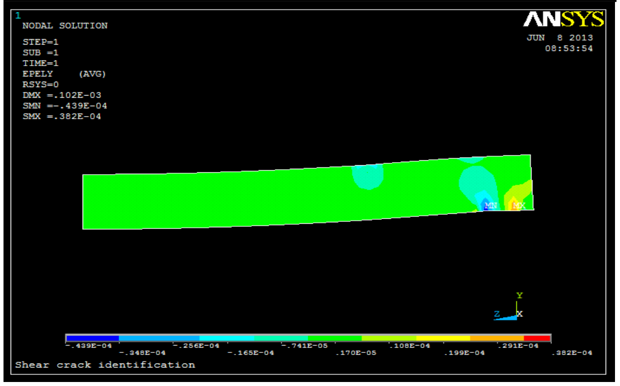


Fig-9 Deformed Shape Fig-10 Stress Contour at Initial Crack

The specimen failed at load of 86.8KN and the deformed shape along with stress distribution is shown in Fig-11

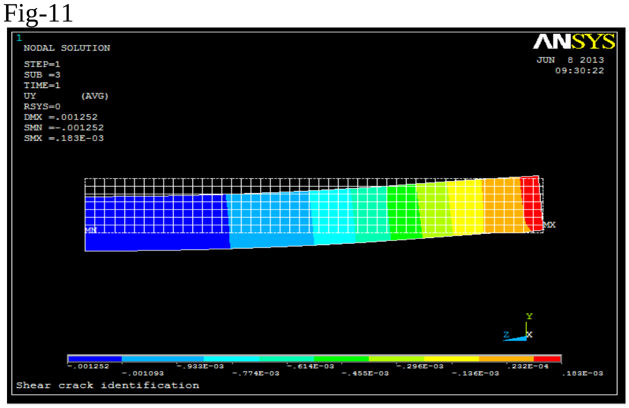


Fig-11 Deformed Shape and Stress Contour At Ultimate

At load of about 86.8 KN the member failed, and crushing of concrete also started at top and near the support as highlighted in Fig-12

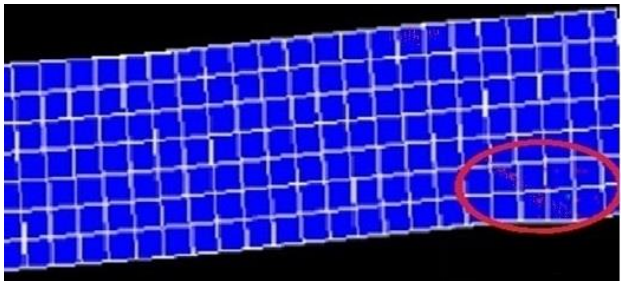


Fig-12 Shear Cracks at Ultimate Load

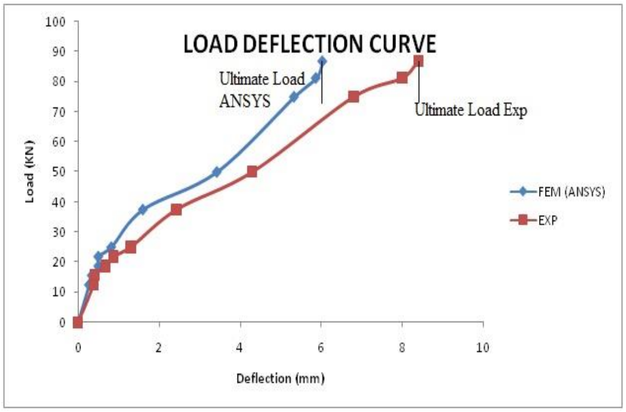


Fig-13

Conclusion:

1. ANSYS is time saving and cost efficient tool that helps in simulation and gives satisfactory results using discrete approach.

2. The initial cracking load is same for both FE and experimental beam.

3. The difference in the ultimate load might be due to perfect bonding in steel and concrete in simulated beam and thus the graph showed high stiffness.

Recommendation for Future Work: The literature review and method of modeling used for this study will not just save time but it also explains the method of modeling stirrups in FE beam. The bond between concrete and steel was considered perfect in our analysis which might be the reason that the FE model was giving lesser deflection due to more stiffness than actual beam in which bond was not perfect and certain slippage was there, therefore, it is recommended to use bond link element between concrete and steel for future studies for more accurate results and better calibration.

Acknowledgement: Authors are very grateful to University of Engineering and technology Lahore, Pakistan for their moral support.

REFERENCES

Ayman S.M, S. Banerjee Civil and Environmental Engineering Department, University of California at Irvine, California 92697,United States , Elsevier Composites: pg 781–793 (2007)

Bangash M. Y. H., Concrete and Concrete Structures, Numerical Modeling and Applications, Elsevier Science Publisher Ltd, London, pg 360-370 (1989)

Chote Soranakom and Barzin Mobasher, Flexural Design Of Fiber Reinforced Concrete, ACI Materials Journal Vol. 106-M52, pg 2 (2009)

Dahmani.L, A. Khennane and S. Kaci Strength of Materials, Vol. 42, pg 2 (2010)

Kachlakev. D and T. Miller, FE Modeling of Reinforced Concrete Structures Strengthened with FRP Lamiates, Final Report SPR 316, Oregon State University (2001).

MacGregor, J.G. Reinforced Concrete Mechanics and Design, Prentice-Hall, Inc. Englewood Cliffs, NJ pg. 25 (1992)

William. K. J. and E. D. Warnke, “Constitutive model for the triaxial behavior of concrete,” in Association for Bridge and Structural Engineering, ISMES, Vol. 19: pg 1-30,174 (1975).