Nonlinear Finite Elements Modeling of Hybrid Reinforced Concrete Beams

التمثيـل اللاخطـي بالعناصـر المحـددة للعتبـات الخرسانيـة المسلـحة المهجنـة

م.م أكـرم شاكـر محمـود / قسم الهندسة المدنية/كلية الهندسة/ جامعة الانبار/ماجستير هندسة إنشائية
Mobile: +964-7901582965 / akramsh1@yahoo.com Email:
م.م شـذى صـاحب كـريم/ قسم هندسة الطرق والنقل/ كلية الهندسة/ المستنصرية/ ماجستير هندسة إنشائية
Mobile: +964-7703408951 /Email: shsk_alasadi@yahoo.com

Abstract:

This paper presents results from a nonlinear finite element analysis of hybrid reinforced concrete rectangular section beams. The concept of hybrid R.C. concrete structures was applied by many designers for use in constructions of bridge superstructures. Although the steel reinforcement is often modeled as a linear-elastic material until failure, the linear finite element analysis could not predict the behavior of the R.C. concrete beams model up to failure. Results from material tests and loading tests of beams model indicated that the nonlinearity of concrete should be considered to predict the failure modes. In this study, an effort was made to implement a rate-independent plasticity model, which describes a constitutive model of concrete types (Normal and High Strength Concrete), in a finite element code. Results obtained from the numerical analysis showed a good correlation with the experimental results.

Key Words: Nonlinear Finite elements, Hybrid Concrete, Normal and High Strength Concrete Modeling, and ANSYS.

الخلاصة:

يُعنى هذا البحث بنتائج تحليل لاخطي بالعناصر المحددة لعتبات مسلحة مهجنة ذات مقاطع مستطيلة. يُطبق مبدأ العتبات المهجنة من قبل الكثير من المصممين وذلك باستخدامه في الجسور. بالرغم من استخدام التحليل المرن-الخطي في تمثيل خصائص حديد التسليح لكن لا يمكنه تقدير السلوك للعتبات الخرسانية المسلحة وتَتبُعُها حتى حمل الفشل. اعتمدت النتائج من فحوصات المواد والأحمال على نموذج السلوك اللاخطي حيث في هذا البحث أُستخدم نموذج اللدن للخرسانة لوصفه بنوعيه (خرسانة ذات تحمل اعتيادي وعالي) في العناصر المحددة. وقد أظهرت النتائج التي استخرجت من التحليل العددي توافق جيد مع النتائج العملية.

Introduction:

Concrete structural components require understanding the responses of those components to a variety of loadings. There is several methods for modeling the concrete structures through both analytical and numerical approaches.
Finite element analysis (FEA) is a numerical method widely applied to the concrete structures based on the use of the nonlinear behavior of material. FEA provides a tool that can simulate and predict the responses of reinforced concrete members. A number of commercial FEA codes are available in e-markets along with the use of FEA that increased because of progressing knowledge and capability of computer package and hardware.

Any attempts for engineering analysis can be done conveniently and fast using such versatile FEA packages. These result in the modernization of structural modeling by new generation practical engineers, in order to verify their structural designs. Nonlinear material models have been integrated in many of general purpose finite element codes. These nonlinear models play a vital role in nonlinear response analyses since each material component tends to possess the complicated stress-strain behaviors.

Among those packages, ANSYS [1] provide a three-dimensional element (Solid 65) with the nonlinear model of brittle materials similar to the concrete material. The element features in a smeared crack analogy for cracking in tension zones and a plasticity algorithm to take into account the concrete crushing in compression zones. This eight noded solid isoparametric element with the integration points for the cracking and crushing checks. The linear elastic behavior governs the analyses until exceeding either the specified tensile or compressive strengths. Once the principal stresses at the integration points reach the tensile or compressive strength, the cracked or crushed regions will form perpendicular with the locally redistributed residue stresses to the direction of principal stresses. These require the nonlinear iterative solver with high performance computer [2]. Hybrid beam use is a new technology for construction of the R.C. beams by reducing the compression concrete zone through using high strength concrete for this zone and the using normal strength concrete for the remaining area of the section; this type of beams is usually used for economic factor [3].

The objective of this study was to develop a numerical model where nonlinear material properties of hybrid concrete beams can be included in detailed analyses for static investigations in the future. Due to its availability to inclustry, a commercial FEA package is preferable in order to use the model in general design practice. A three-dimensional nonlinear finite element model of hybrid concrete beams was developed by a general purpose finite element analysis package, ANSYS 10. The concrete section was modeled using SOLID 65 solid element where the compressive crushing strength of concrete is facilitated using plasticity algorithm and the concrete cracking in tension zone is accommodated by the nonlinear material model .
For the use of truss element (LINK8) is used for discrete reinforcement modeling (main and transverse reinforcements). However, it was assumed that perfect bonding between concrete and steel reinforcement occurs during loading.

Specimens Details:

The hybrid beam was cast and tested by Kareem [3]. Experimental programs used with four point bending moment test. The dimensions of all the beams are geometrically similar, having rectangular cross-section, of dimension (175×275 ×3000) mm. The beams are tested simply supported over (2800) mm span and loaded at two points having a distance of (933.33) mm between them. The tested compressive strength of cored concrete were 72 MPa for high strength concrete and 22 MPa for normal concrete strength. The yield of steel reinforcements is (400 MPa) with bilinear strain –stress with tangent 200 MPa. The cross section details are selected from a reference for modeling by FEA, as shown in Fig. (1). The experimental results, ie. Load-deflection curve, strain of sections and so on, can also be found in that reference [3].

Material Models:

Concrete:

ANSYS software has the ability to model different types of material properties. Cracking of concrete and stress-strain relation in tension is modeled by a linear elastic tension stiffening relationship. The cracking stress of concrete is taken equal to the modulus of rupture calculated according to the ACI 318-08 code [4]. In the present study the ability of concrete to transfer shear forces across the crack interface is accounted for by using two different shear retention factors (β) for cracked shear modulus, it is assumed equal to 0.3 for opened crack and 0.8 for closed crack. The concrete constitutive relationships under multiaxial state of stresses is based upon the Willam and Warnke[5] model which is considered as an appropriate model to describe the concrete failure. In this model the yield condition is approximated by five parameters and it is used to distinguish linear from non-linear and elastic from inelastic deformations using the failure envelope defined by a scalar function of stresses $f(\sigma)=0$ through a flow rule, using incremental stress-strain relations. In the present work the equivalent uniaxial stress-strain relationship for concrete under compression is represented by using the equation suggested by Desayi and Krishnan[6] to compute the multilinear isotropic stress-strain curve for concrete in the form [7]:

$$\sigma = \frac{E_c\ \varepsilon}{1+\left(\dfrac{\varepsilon}{\varepsilon_o}\right)^2} \qquad \text{……… (1)}$$

Where, E_c is the elastic modulus of concrete calculated as per the ACI 318-08 code [5] and it is equal to 22185 MPa for $f_c^{'} = 22MPa$ and 41800MPa for $f_c^{'} = 70MPa$. ε = strain corresponding to stress σ and ε_o is the strain at peak stress equal to $2f_c'/E_c$ and $f_c^{'}$ is the concrete compressive strength. Twelve points are used to represent the stress strain curve using Eq.(1) starting from the elastic stress limit (0.3 $f_c^{'}$) up to $f_c^{'}$=22 or 70 MPa and last point is corresponding to ultimate strain (ε_u) equal to 0,0035 and 0.0045 at peak stress, respectively. The resulting multilinear isotropic hardening stress-strain curve for normal and high strength concrete is shown in Figs. (2) & (3) respectively.

Reinforcing steel:

Modeling of reinforcing steel in finite elements is much simpler than the modeling of concrete. Link8 element was used to model steel reinforcement. This element is a 3D spar element and it has two nodes with three degrees of freedom – translations in the nodal x, y, and z directions. This element is also capable of plastic deformation. A perfect bond between the concrete and steel reinforcement is considered. However, in the present study the steel reinforcing was connected between nodes of each adjacent concrete solid element, so the two materials shared the same nodes. Steel reinforcement in the experimental beams was constructed with typical steel reinforcing bars. Elastic modulus and yield stress for the steel reinforcement used in this FEM study follow the design material properties used for the experimental investigation. The steel for the finite element models is assumed to be an elastic- perfectly plastic material and identical in tension and compression as shown in Fig. (4). A Poisson's ratio of 0.3 is used for the steel reinforcement.

Steel Plates:

Loading and anchorage plates are adopted as Bilinear Kinematics Hardening (BKIN). The adopted yield stress is equal to 400 MPa and the hardening modulus is taken equal to 200 MPa; while the elastic modulus is equal to 200000 MPa and Poisson's ratio 0.3.

Finite element Analysis:

Two beams are analyzed by nonlinear finite element using the proposed concrete. The first beam has $2\phi16mm$ (specimen symbol 2HY16) and the second beam has $2\phi25mm$ (specimen symbol 2HY25) as the main flexural reinforcements. Modeling of reinforced concrete specimens is using by Soild65 and Link8 elements for concrete and steel reinforcements respectively. The concrete elements of high and normal strength have perfect bond by using same nodes on these elements. The concrete and reinforcement elements are arranged as shown in fig. (5), also boundary condition of half beam is shown.

The experimental and FEA load-deflection curves obtained for the beams are illustrated in Figs. (6) & (7). The curves show good agreement between finite element analysis and experimental results throughout the entire range up to failure mode. For all beams the finite element model is stiffer than the actual beam in the linear range. Several factors may cause the higher stiffness in the finite element models. The bond between the concrete and steel reinforcing is assumed to be perfect(no slip) in the finite element analyses, but for the actual beams the assumption would not be true since slip occurs therefore the composite action between the concrete and steel reinforcing is lost in the actual beams. Also the microcracks produced by drying shrinkage and handling are present in the concrete to some degree. These would reduce the stiffness of the actual beams, while the finite element models do not include microcracks due to factors that are not incorporated into the models. After the initiation of flexural cracks, the beam stiffness was reduced and the linear load –deflection behavior ended when the internal steel reinforcement began to yield. As shown in Figs. (6, 7), the second beam (2HY25) is stiffer than the first beam (2HY16).

Figs. (8) & (9) show the strains along the midspan section depth at three stages, before cracking, 60% of ultimate load (approximately near the yielding stage), and ultimate load stages. Four fiber points are chosen in the section to plot this relation. The first point is at top of section which has max compressive stress, the second point at the interfaces between to concrete materials, the third point is in the steel reinforcement fiber level, and fourth point is at the max tension fiber level.

Figs. (10) & (11) have explained the crack patterns in two beams and compared with photo from the lab [6]. Very closed direction and types of the cracks are notice. Ansys has capability to display the cracks at locations of cracking or crushing in concrete elements. Cracking is shown with a circle outline in the plane of the crack, and crushing is shown with an octahedron outline. The first crack at an integration point is shown with a red circle outline, the second crack with a green outline, and the third crack with a blue outline.

The first beam namely failed in flexure, where the cracks are located in mid third span of beam fig. (10,b) ,while second tested beam is failed in diagonal tension forming a through crack fig (11,b). the comparison of the crack / crush pattern predicted to that observed in experiment indicted that the Ansys model predicts the zones of critical cracks quite accurately.

Conclusions:

Based on the comparison of the predicted results of hybrid reinforced concrete beams have different reinforcement ratios with corresponding experimental data, following conclusions were drawn from this work:

1) The predicted load of hybrid beams at various stage was found to be in good agreement with test data.
2) Finite element model was able to simulate the behavior of test beam accurately , where stain distribution along sections depth have good agreements with experimental results.
3) Nonlinear finite elements analysis predicted the ultimate capacity and the crack patterns like the experiments. Also indicated the failure types.
4) The present finite element model can be used in additional studies to develop design rules for Hybrid reinforced concrete members.

References:

[1] ANSYS10, "*Documentation Manual of ANSYS V.10*", 2006

[2] Fanning, P. " *Nonlinear models of reinforced and post- tensioned concrete beam*", Electronic Journal of Structural Engineering, No.2, pp.111-119, 2001.

[3] Kareem Sh. S**.** " *Flexural behavior of Hybrid reinforced concrete Beams*" M.Sc. thesis, AI-Mustansiriya University, Iraq, 2006.

[4] ACI Committee 318, " *Building Code Requirements for Structural Concrete (ACI 318-08) and Commentary (ACI 318R-08)*", American Concrete Institute, Frarmington Hills, 2008.

[5] Willam, K.J. and Warnke E.D. " *Constitutive Model for the Triaxial Beahavior of Concrete*", Proceeding Int. Assoc. Bridge and Structural Engineering, ISMES, Bergamo, Italy, 1975.

[6] Desayi, P. and Krishnan " *Equation for stress-strain curve of concrete* ", ACI J.,61, pp. 345-350, 1964.

[7] Mac Gregor, J.G**.** , " *Reinforced concrete Mechanics and Design* " Prentice – Hall, Inc., Englewood Cliffs New Jersy, USA, 1997.

Figures:

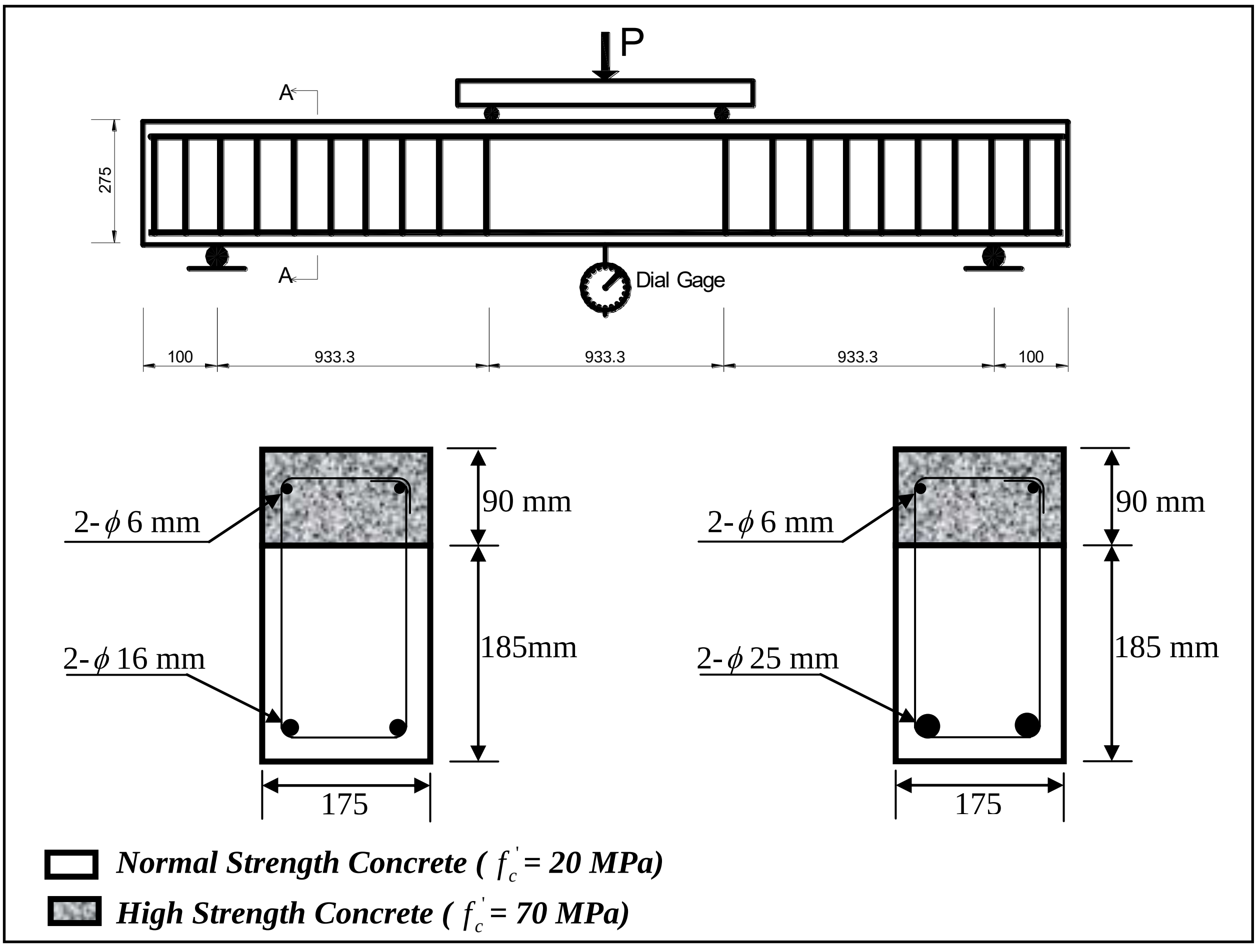


Fig. (1) Loading And Specimen Details.

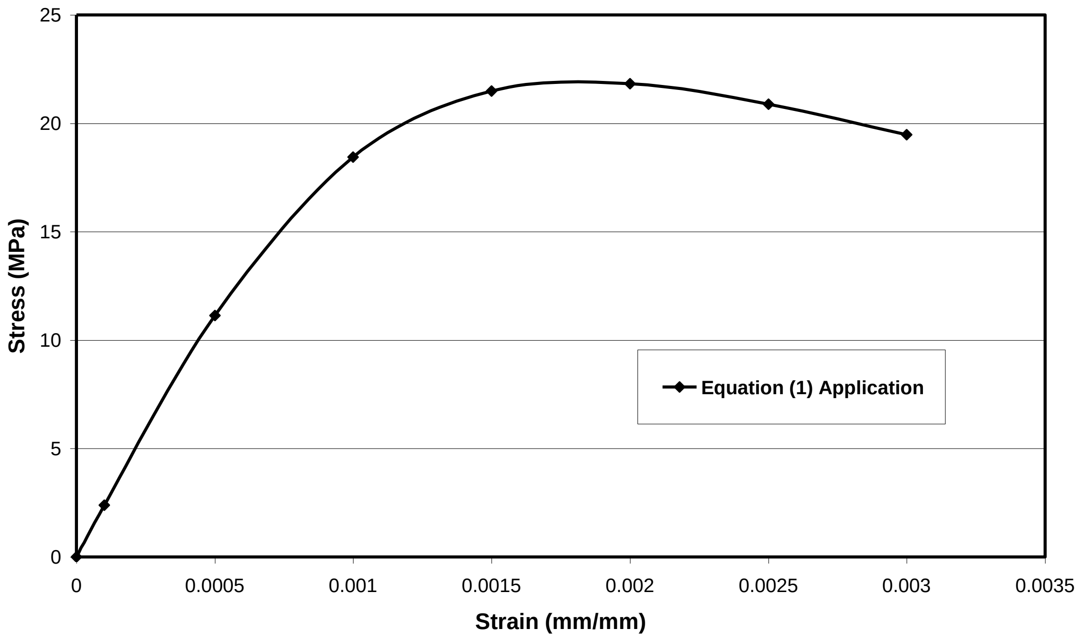


Fig.(2) Stress Strain relation of the Normal Strength Concrete

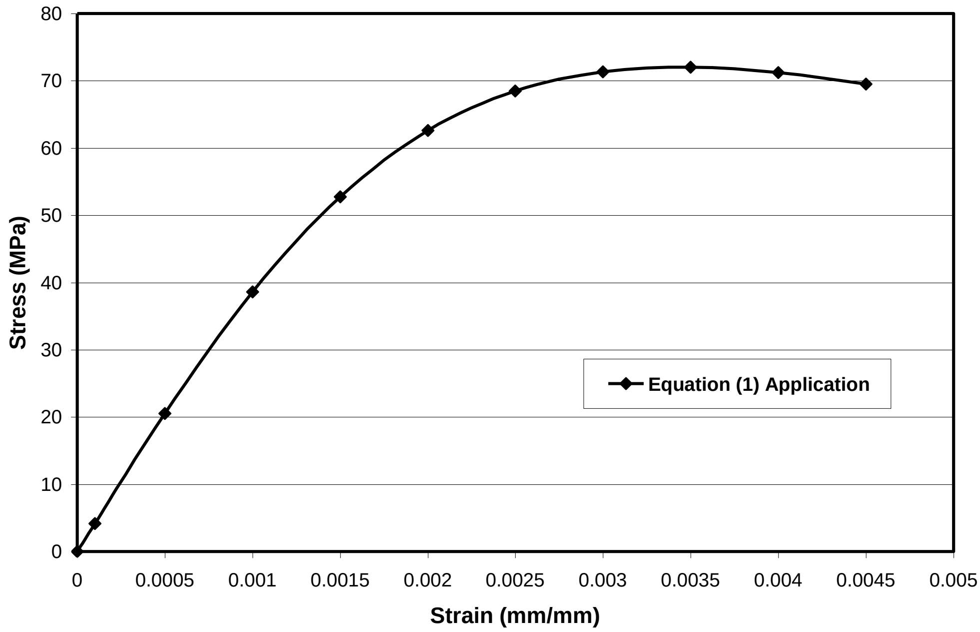


Fig.(3) Stress Strain relation of the High Strength Concrete

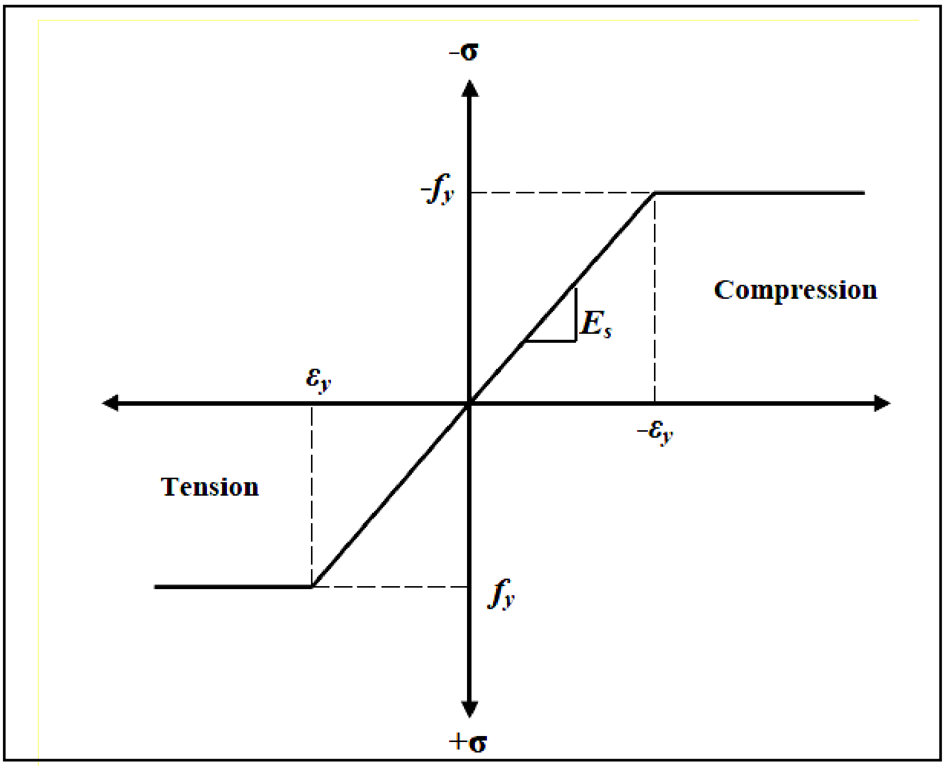


Fig. (4): Stress-strain curve for steel reinforcement

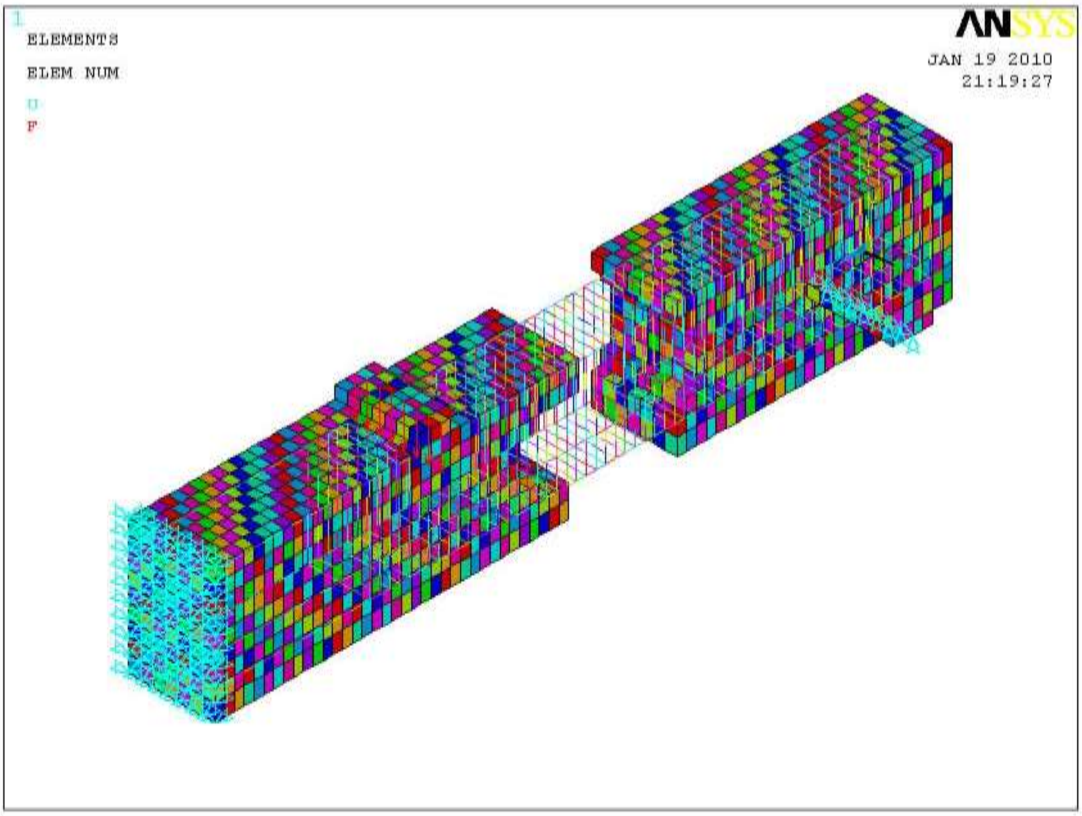


Fig.(5) Elements distribution of concrete and reinforcements for half length of beam

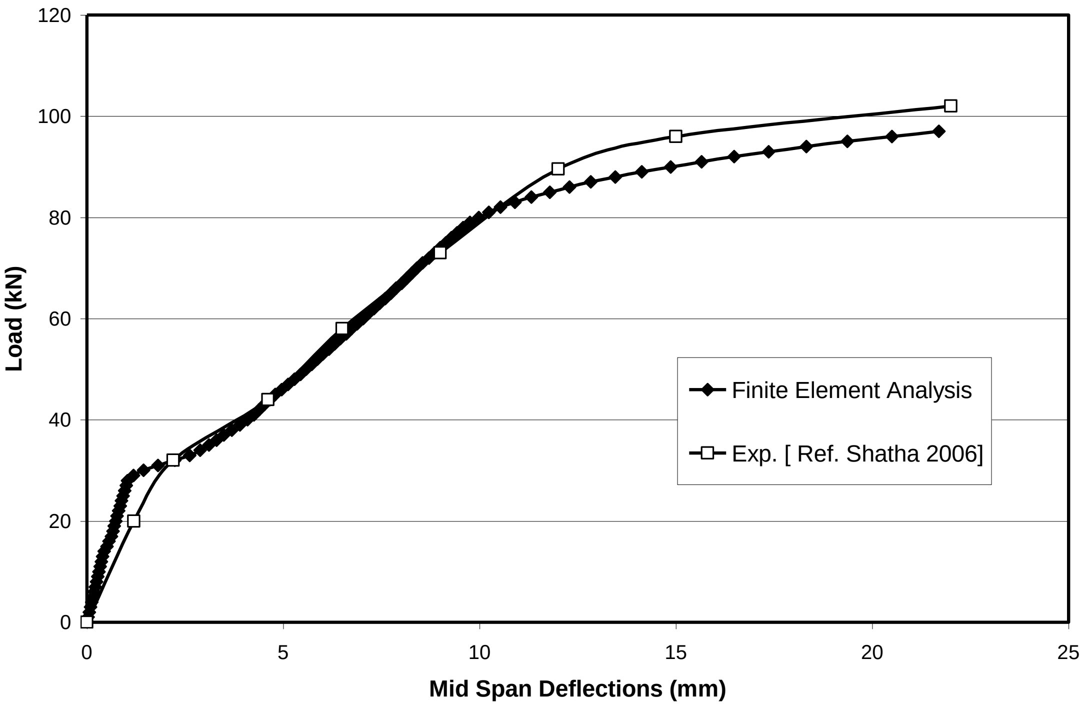


Fig.(6) Load-Deflection Curve of 2HY16 beam

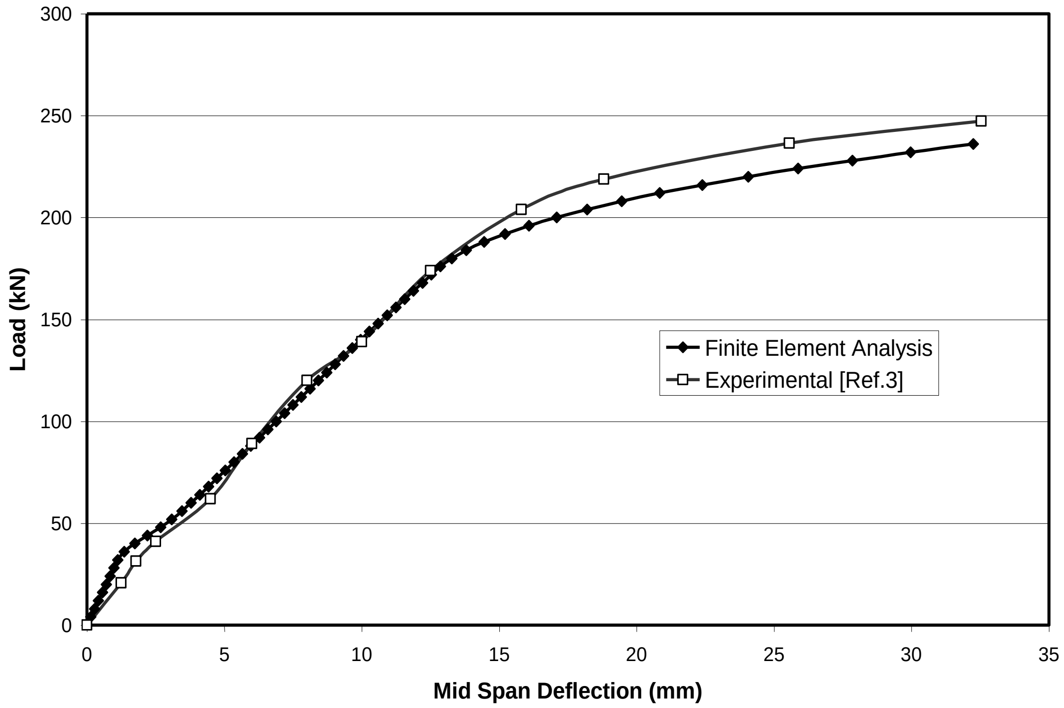


Fig.(7) Load-Deflection Curve of 2HY25 beam

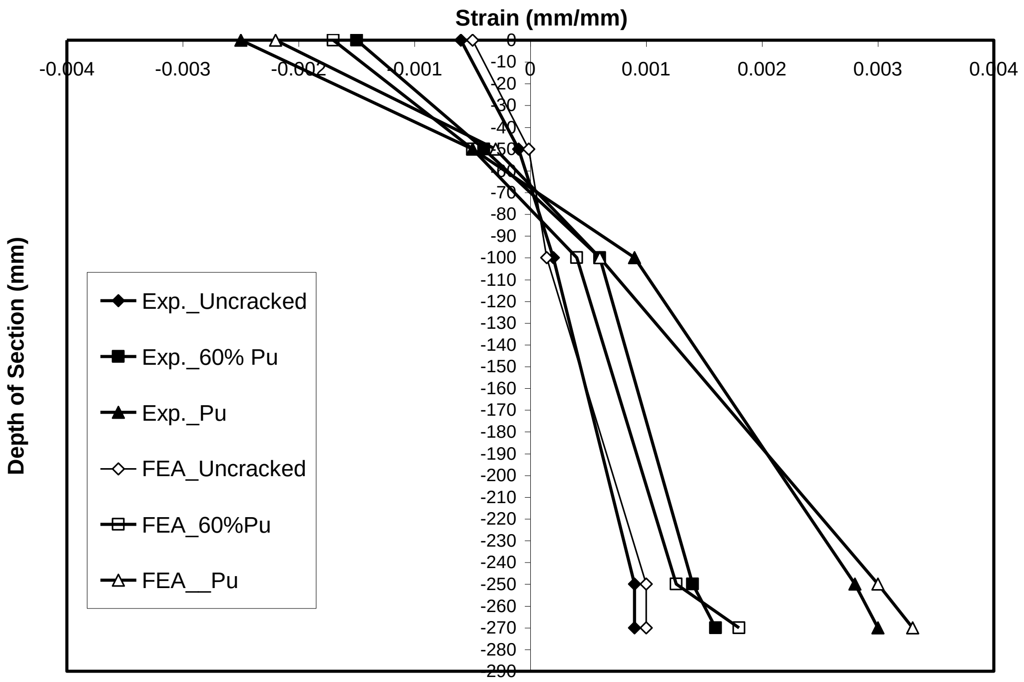


Fig.(8) Strain along mid span section depth of 2HY16 beam

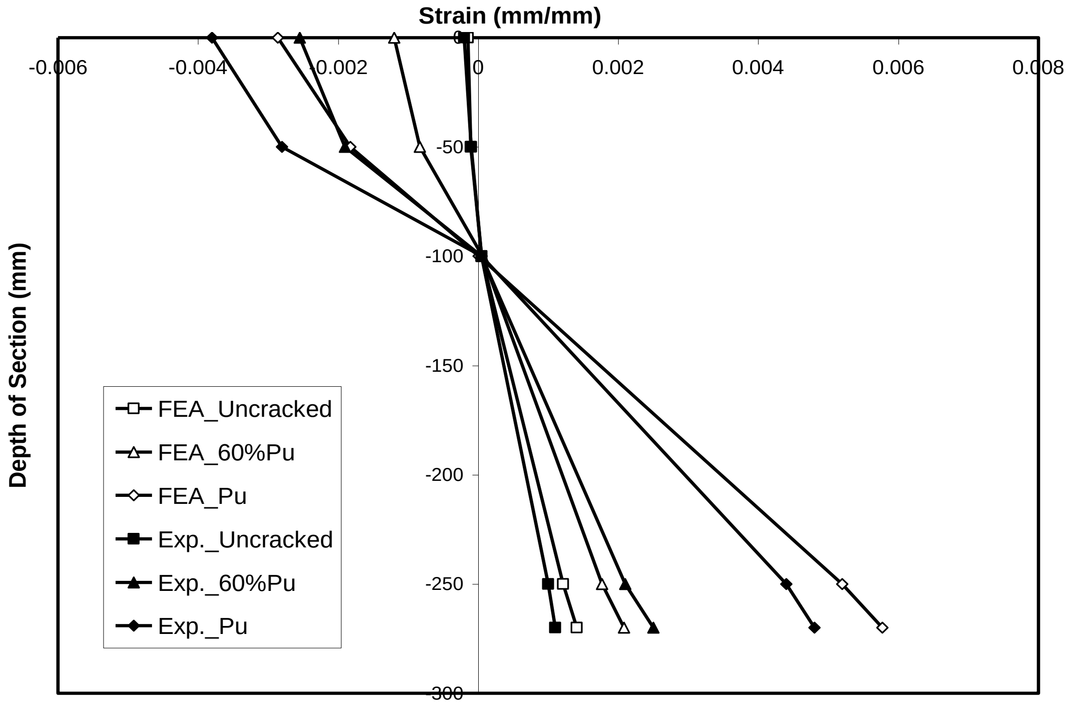
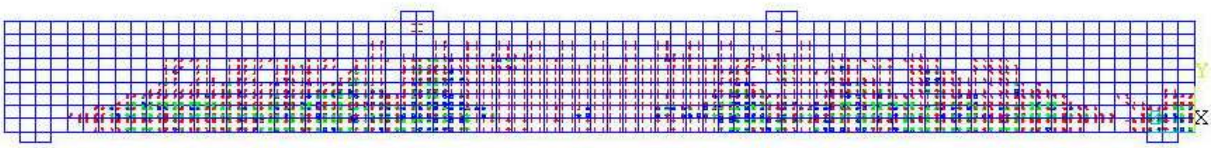


Fig.(9) Strain along mid span section depth of 2HY25 beam

(a): Experimental Crack Patterns



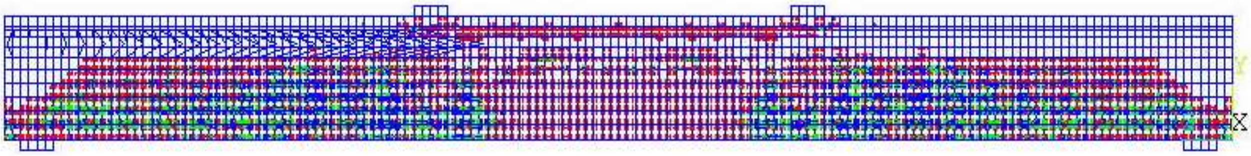
(b): FE Crack Patterns

Fig.(10) Crack patterns of the beam 2HY16 (Steel yielding Failure)

a. Experimental Photo

b. By Finite Element Analysis

(a): Experimental Crack Patterns



b): FE Crack Patterns

Fig.(11) Crack patterns of the beam 2HY25 (Compression Region failure)
a. Experimental Photo
b. By Finite Element Analysis