EXPERIMENTAL INVESTIGATION AND NONLINEAR ANALYSIS OF POLYMER MEMBERS SUBJECTED TO UNIAXIAL TENSION

Akram Shaker Mahmood
Civil Engineering Department
University of Anbar

Arz Yahya Rzayeg
Mechanical Engineering Department
University of Anbar

ABSTRACT

An experimental investigation as well as nonlinear analysis is carried out in this paper to study the behavior of polymer members (Epoxy & Polyester) under direct tension. The ANSYS model accounts for nonlinear phenomenon, such as, Tension Softening Material (TSM) and Enhanced Multilinear Isotropic Softening (EMIS) models. The polymer specimens are modeled using PLANE82 element – eight node plane element – eight node plane element, which is capable of simulating the failure behavior of polymer material members. The intention of this paper is thereby to discuss the proposed softening models to validate the complete Stress-Strain and Load-Deflection response of prismatic specimens subjected to uniaxial tension. The outcomes from the verifications of both modeling techniques have shown good agreement with the experimental results obtained from literature.

KEYWORDS: Nonlinear Material, Finite Element, Epoxy, Polyester, Tension Softening, and Enhanced Multilinear Isotropic Softening.

1.INTRODUCTION

Most polymers behave as viscoelastic at any condition and at a wide range of temperatures such as, whereas some of the engineering materials exhibit viscoelasticity, such as most metals, only when temperature increases to permit material nonlinearity.

Most materials which have viscoelastic properties are solved linearly if they show small deformations, otherwise, they are solved by the non-linear solution.

In metals and non-metals materials, the time-dependent response is required to analyze problems in important areas of engineering such as the problem of solid propellant rocket fuels, the problems of turbine blade, and soil mechanics which require analysis taking into account viscoelastic , viscoplastic or creep response under varying time, temperature and loads histories. In fact, the activity in this field is primarily due to the large-scale development and utilization of polymeric materials. Many of these newly - developed materials exhibit mechanical response characteristics which are outside the scope of such theories of mechanical behavior as elasticity and viscosity Thus, the need for a more general theory is quite apparent.

The basic objective of the present paper is to present a mathematical model suitable for the nonlinear analysis of polymer members under monotonically increasing loads. This model incorporates the material nonlinearity due to the deformation in material particles.

The main obstacle to finite element analysis of polymer members is the difficultly in characterizing the material properties [1], [2]. Much effort has been spent in search of a realistic model to predict nature of polymer (Epoxy & Polyester) material for which proper modeling of such members is a challenging task. Despite the great advances achieved in the fields of plasticity, damage theory and fracture mechanics, among others, an unique and complete constitutive model for polymer material structures is still lacking.

2.EXPERIMENTAL WORK

2.1 Manufacturing Procedures

Fig (1) shows all parts of the mold which is used to prepare the specimen; these are:

1. The upper glass: It is a piece of glass of dimensions (200×200×6mm)
2. The lower glass: It is exactly the same as the upper glass.
3. The rest: It is the part on which the upper and the lower glass rested.
4. The arm: This is used to apply a suitable pressure on the surface of the upper glass.
5. The wax: It is used to paint the upper and the lower glass to avoid the specimen being adhered to the mold.

2.2 Preparation of Material for Manufacturing

The preparation process included mixing of the resin polyester with (0.2- 0.5 % polyester) cobalt and (1.5-2 %polyester) hardener, or mixing the resin epoxy, in addition to performing the necessary measurements.

2.3 Description of Tensile Test

The length and width of the tension coupons used in both the experimental and numerical evaluations are in accordance with ASTM Standard D3039 [3]. The specimen is (110mm) long, (20mm) wide and (3.5mm) thick. The tabs were (37.5mm) long, (20mm) wide, (3.5mm) thick as shown in Fig (2) with a taper angle of 20° smoothly curved. The set of dimensions are the effective length of specimen Le, the tentative length L, and width of the specimen W [4].

2.4 Tensile Testing Apparatus

The apparatus of tensile testing used in this study is shown in Fig (3). Experiments are carried out at the metal laboratory, in the Specialized Institute for Engineering Industries in Baghdad at laboratory temperature and with a load application speed equal to (0.2 mm/min), the tensile testing machine comprises the following:

1. Two movable members, each of which is carrying one grip.
2. Grips for holding the test specimen between the fixed member and the movable member.
3. Drive mechanism for imparting to the movable member a uniform, controlled velocity with respect to the stationary member.
4. Load indicator with suitable load indicating mechanism capable of showing the tensile load carried by the test specimen when held by the grips.
5. Extension indicator which presents a suitable instrument for determining the distance between two fixed points located within the gage length of the test specimen at any time during the test.

3. EXPERIMENTAL RESULTS

In this work, two types of resins are used the properties of which are shown in Table (1). Fig. (4) Shows photos of the specimens used in this work. Table (2) illustrates the different responses of deflection of both polyester and epoxy resin for the same type and amount of fiber and with the same load and time**.**

4. MATERIAL MODEL

The finite element code ANSYS, Version 10.0, has been used. Its polymer material model consists of material model to predict the failure of elastic materials, applied to a two dimensional PLANE 82 element. The material is capable of plastic deformation. The tension softening is determined by failure surface. Tensile failure is defined by the maximum tensile stress criterion, a tension cutoff. Unless plastic deformation is taken into account, the two material behaviors are linear and nonlinear elastic until failure.

5. DETERMINISTIC FINITE ELEMENT ANALYSIS

5.1 Model Overview

A 2-D solid model of the actual test specimen is built using ANSYS 10.0 software package [6] using the PLANE 82 element. PLANE 82 is an 8-node structural plane element designed to model plane stress invariants [7]. The element is defined by 8 nodes having two degrees of freedom per node: translations in the nodal x, and y directions. The X-direction corresponds to the longitudinal direction of the test specimen and the loading direction. The Y-direction corresponds to the lateral direction of the test specimen. Convergence studies are conducted to evaluate the proper mesh size for the problem at hand. It is determined that (72) plane elements are required as shown in Fig. (5).

5.2 Model Restraints and Load Application

The model is composed of two different components: tabs, and adhesive polymeric matrix material. The bottom tabs are restrained on their outer flat area in the y-direction (Uy = 0), whereas the upper tabs were subjected to uniform displacement applied to the outer flat area. A uniform pressure is applied on the flat tabs area to simulate the actual experimental grip pressure.

6. ANALYSIS RESULTS

The linear and nonlinear finite element solutions are done using ANSYS 10.0, the small displacement static and line search nonlinear options are used. A 20^{th} is number of sub-steps with 10^{-4} value of tolerance. The results of nonlinear solution iterations for every sub-steps

7. POLYESTER TEST

From experimental results, a load-extension relation is shown in Fig. (7), including the solution of linear and nonlinear finite element analysis. Good agreements of numerical and experimental results. Finite element solutions (linear & nonlinear) initially have approximately the same values, but the nonlinear solution is closer to the experimental results. Accordingly the polymer member behaves as elastic nonlinear material, according to the tension softening model issued in finite element analysis. Fig. (8), shows the stress-strain relationship of experimental results and finite element solutions. The polyester members are tested and modeled using finite element method an shown good verification in this paper. The contour results of stress in linear solution are shown in Fig. (9).

8. EPOXY TEST

In epoxy test, the two epoxy resin members are tested in laboratory, and the finite element solutions are done for linear and nonlinear behavior. Fig. (10) Shows the relationship of load- extension. Good agreement is shown, and nonlinear solution has close results with experimental outcomes. A stress-strain relation is explained in Fig. (11), the nonlinear materials finite elements are done and compared with experimental test.

Good agreement is shown. The Contour results using ANSYS for nonlinear FE solution of stress and strain are shown in Fig. (12).

9. CONCLUSIONS

Based on the comparison of the predicted results of polymer (epoxy and polyester) members with corresponding experimental data, the following conclusions are drawn:

1. The predicted strength of polymer member at various stages is found to be in good agreement with test data, especially for non-linear finite element solution.
2. The proposed model predicted slightly softer results in non linear material of the load deflection response of polymer members.
3. From all validated test cases, the tension softening material (TSM) model proposed herein has been proved to be able in determining the complete stress- strain response for specimen of various geometries.
4. Finite element models constructed in ANSYS V.10 using the dedicated element (2D plane element) have accurately captured the tension response of these systems up to failure.
5. It is expected that the modeling strategy for finite element analysis proposed in this study is useful for analyzing polymer members.

REFERENCES

[1] Chen, W. F. " Constitutive Equations for Engineering Materials – Volume: Plasticity and Modeling", Elsevier Science B.V., 1994.

[2] Barbosa, A. F., and Ribeiro, G. O. " Analysis of Reinforced Concrete Structures Using ANSYS Nonlinear Concrete Model", Computational Mechanics, CIMNE, Barcelona, Spain, 1998.

[3] ASTM Standard D 3039/D 3039M-00, Standard Test Method for Tensile Properties of Polymer Matrix Composite Materials. Annual Book of ASTM Standards. American Society of Testing Materials, West Conshohocken, 2000.

[4] Todo, M. "Effective of Strain Rate on Tensile Fracture Behavior of Fiber Reinforced Polyamide Composites", ICCM-11 Conference, Gold Coast, Australia, July, 1997.

[5] Rzayeg, A. Y. " Linear Viscoelastic Behavior of Composite Beams", M.Sc. Thesis, College of Engineering, ,University of Anbar, (2005).

[6] ANSYS Theory Reference, ANSYS, Inc. (2006).

[7] Zienkiewicz , O. C., and Taylor, R. L. "The finite element method: The Basis –Volume 1", 5th Edition, McGraw-Hill, London, (2000).

Table (1): Mechanical Properties of the Resin used in Present Work [5]

Resin type	**Young modulus E (GPa) Experimental**	**Shear modulus G (GPa) Experimental**	**Bulk modulus K(GPa) Calculated**	**Poissions ratio** u **Calculated**
Polyester	1.0602	0.384	1.478	0.38
Epoxy	0.333	0.127	0.295	0.312

Table (2): The Composite Responses to Deflection

Resin type	**Applied Load (gm)**	**Deflection (mm)**
Polyester	714	1.79
Epoxy	714	2.62

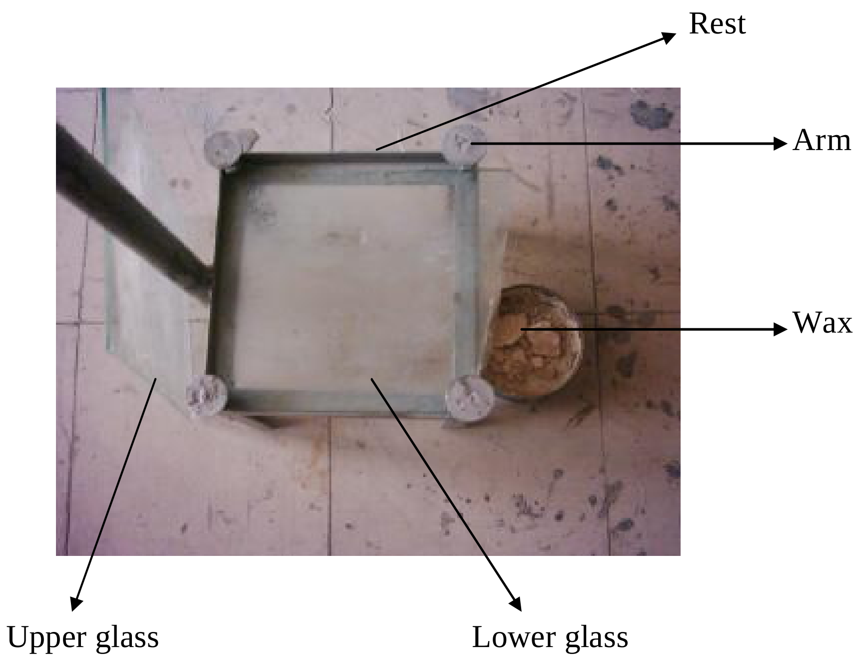
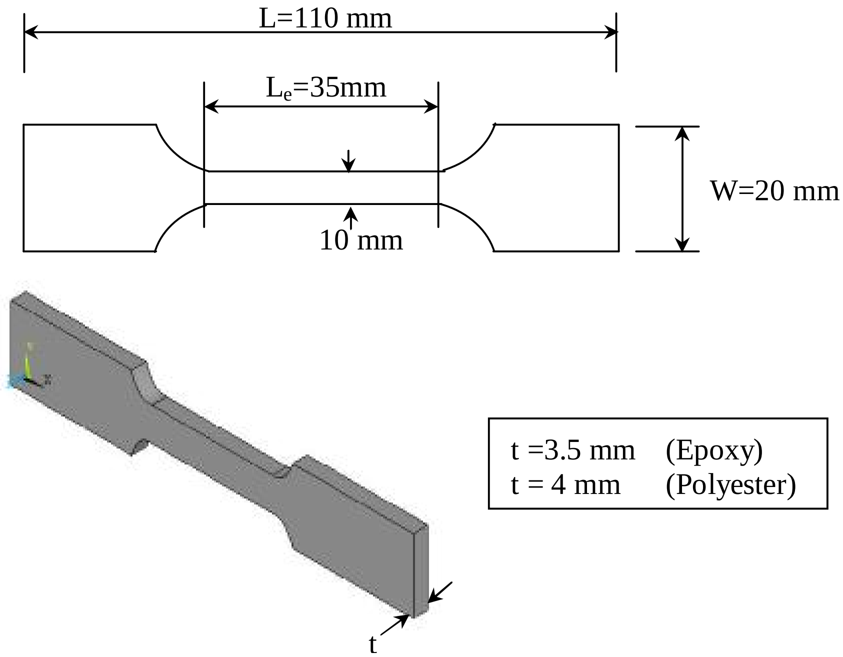


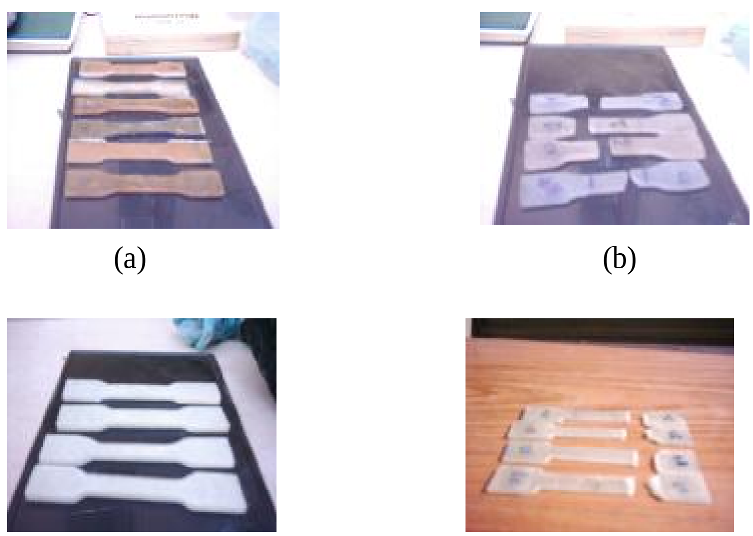
Figure (1): The Mold Components



t =3.5 mm	(Epoxy)
t = 4 mm	(Polyester)

Figure (2): Tensile Specimen Geometry

Figure (3): The Tensile test Apparatus.



(a) (b)

(c) (d)

Figure (4): (a) Epoxy before test. (b) Epoxy after test
(c) Polyester before test (d) Polyester after test

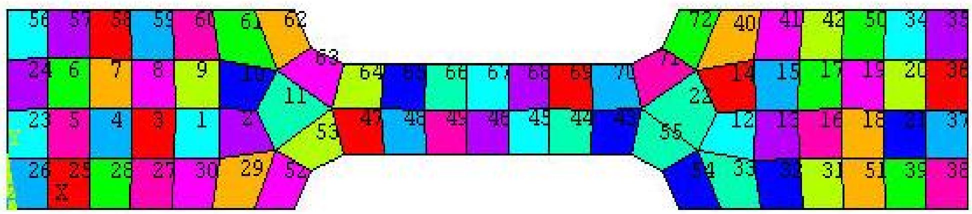


Figure (5): The PLANE 82 Element Simulations

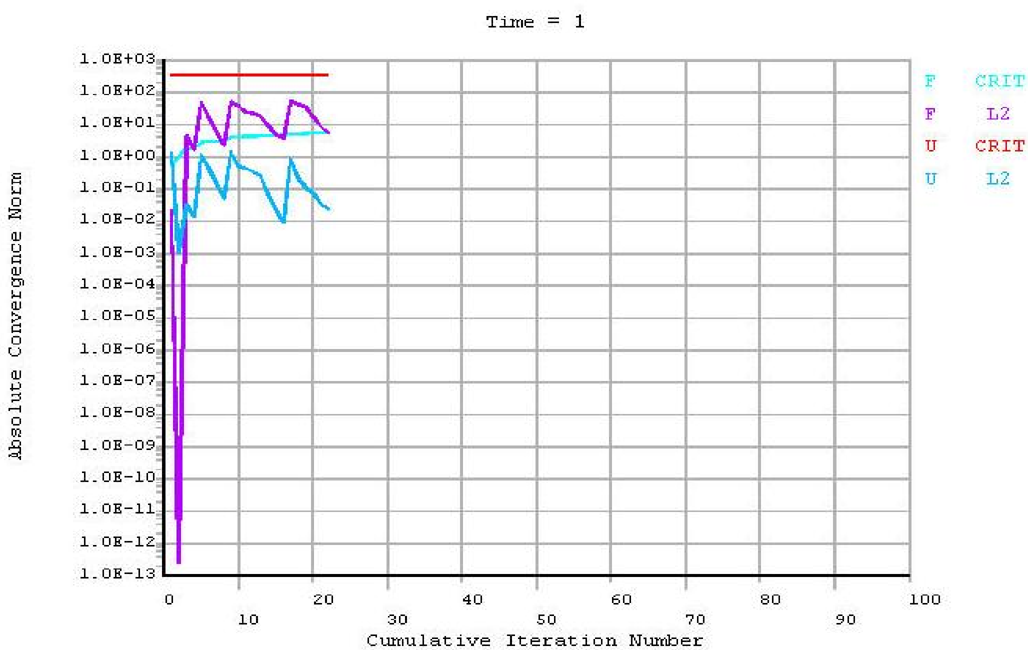


Figure (6): Results of Convergence for Nonlinear Solutions

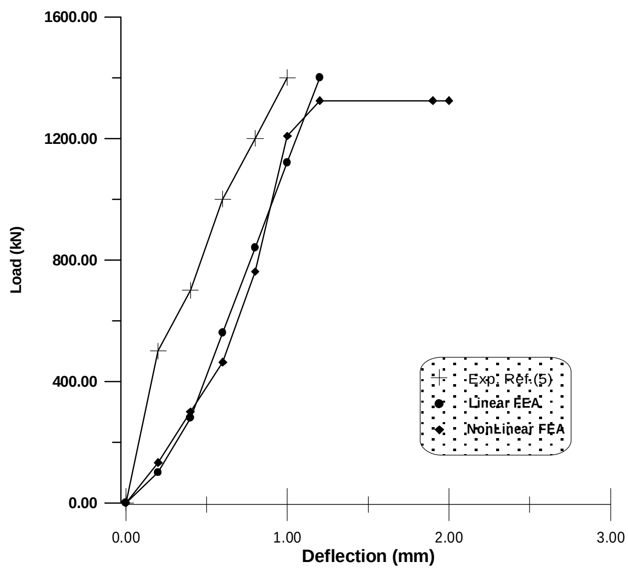


Figure (7): Load-Extension Relation of Polyester Specimen

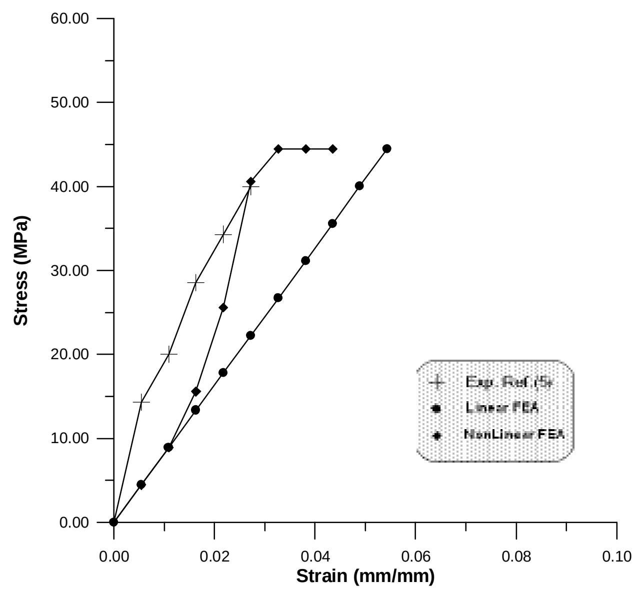


Figure (8): Stress Strain Relation of Polyester Specimen

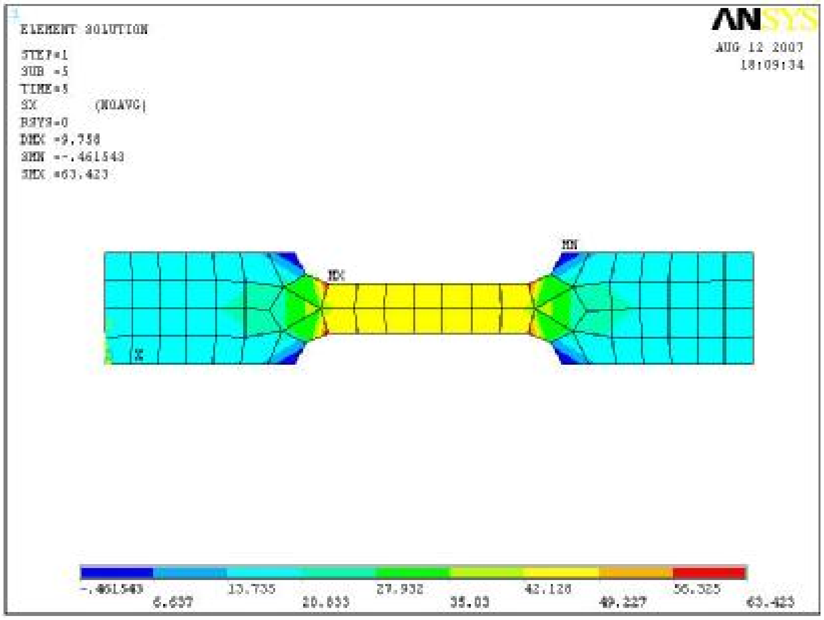


Figure (9): Stress in Polyester using ANSYS Contour Results
for Linear FE Solution

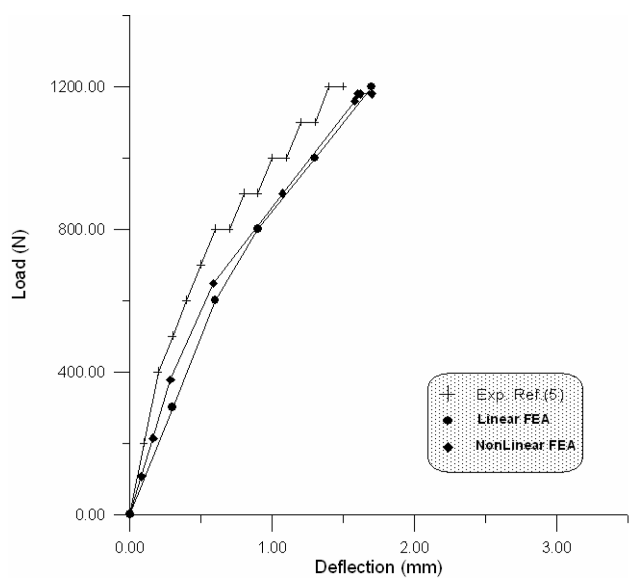


Figure (10): Load- Extension Relation of Epoxy Resin Member

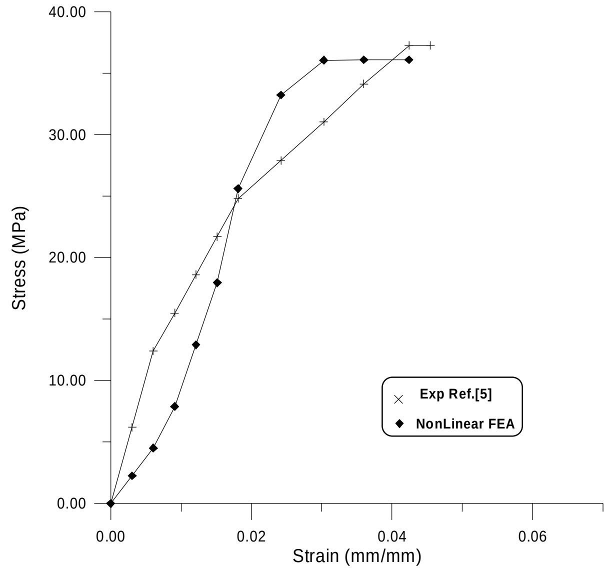
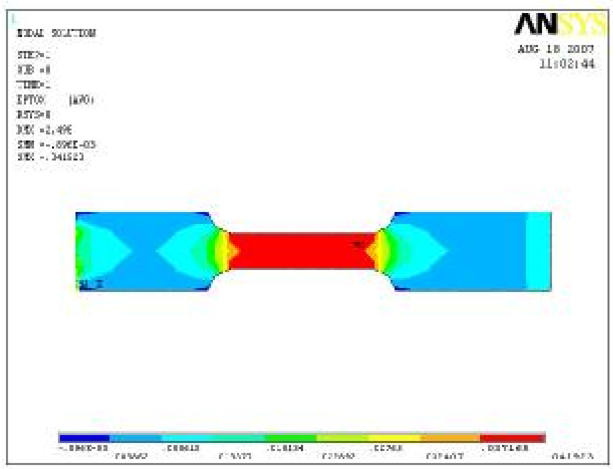
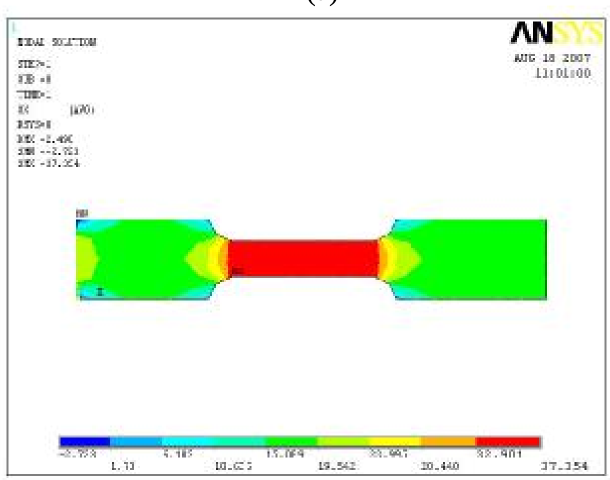


Figure (11): Stress Strain Relation of Epoxy Resin Member



(a)



(b)

Figure (12): Contour Results Using ANSYS for Nonlinear FE Solution
(a) Strain Plot (b) Stress Plot

التحليل اللاخطي باستخدام العناصر المحددة للأعضاء البوليميرية المعرضة لشد محوري

أكرم شاكر محمود
قسم الهندسة المدنية

أرز يحيى ارزيك
قسم الهندسة الميكانيكية

الخلاصة

تُعنى هذه الدراسة بمعرفة سلوك وتصرف الأعضاء البوليميرية مثل (الايبوكسي و البولي أستر) لقد استخرجت نتائج التحليل باستخدام طريقة العناصر المحددة ذات السلوك اللاخطي مع نتائج عملية. نموذج برنامج ANSYS اُستُخْدِم في معرفة السلوك اللاخطي في المادة مطابقاً نماذج تحليلية وهي نموذج استرخاء شد المادة (TSM) ونموذج الاسترخاء المتعدد المطبق الضمني (EMIS). تم تمثيل عينات الأعضاء البوليميرية بعنصر PLANE82 ذو ثمانية عقد الذي كان فعالاً في تمثيل الفشل للأعضاء البوليميرية. إن الغرض من هذه الدراسة هو معرفة مدى قدرة نموذج الاسترخاء المقترح في استخراج علاقة الإجهاد – الانفعال وعلاقة الحمل – الانحراف، لعينات معرضة لشد محوري باتجاه واحد . قورنت نتائج البحث مع نتائج عملية مختبرية أخذت من بحوث سابقة ووجد إن هناك توافق بين تلك النتائج.