Nonlinear Finite Element Analysis of Polymer Members

Akram Shaker Mahmood
Arz Y Rzayeg

Available at: https://works.bepress.com/akram_mahmood/4/
Nonlinear Finite Element Analysis of Polymer Members SubJECTED to Uniaxial Tension

Akram Shaker Mahmood
Civil Engineering Department
Engineering College – Anabr University

Arz Yahya Rzayeg
Mechanical Engineering Department

Abstract:
An investigation study is made for verify the behavior of polymer members (Epoxy & Polyester). The results of nonlinear analysis are illustrated with experimental work results. The ANSYS model accounts for nonlinear material 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 -, 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 to provide good agreement with the experimental results obtained from literature.

الخلاصة:
تُعنى هذه الدراسة في معرفة سلوك وصرف الأعضاء البوليميرية مثل (الإيروكسي و البولي أستر). لقد استخرجت نتائج التحليل بالعناصر المحددة ذات سلوك اللاخطي مع نتائج عملية. نموذج برنامج ANSYS قد استخدم في معرفة السلوك اللاخطي في المادة مطابقاً نماذج تحليلية مثل نموذج استيرخاء شد المادة (TSM) ونموذج الاستيرخاء المتعدد المتطابق الضمني (EMIS). تم تمثيل عينات الأعضاء البوليميرية بعنصر PLANE82 ذو ثمانية عقد الذي كان فعالاً في تمثيل الفشل للأعضاء البوليميرية. إن الغرض من هذا البحث هو معرفة مدى قدرة النموذج الاستيرخاء المقترح في استراخة علاقة الإجهاد-الانفعال و علاقة الحمل-الانحراف، لعينات تحت قوى شد محورية. فورنت نتائج البحث بنتائج عملية مختبرية أخذت من بحوث سابقة ووجدت توافق جيد بين تلك النتائج.

Key Words: Nonlinear material, Finite element, Epoxy, Polyester, Tension Softening, and Enhanced Multilinear Isotropic Softening.
Introduction:

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

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

In metals and non-metals materials, the time-dependent response presents analysis 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 or 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 present paper is to present a mathematical model suitable for the non linear analysis of polymer members under monotonically increasing loads. This method incorporates the material nonlinearity due to the deformation in material particles.

The main obstacle to finite element analysis of polymer members is difficulty in characterizing the material properties [1] and [2]. Much effort has been spent in search of a realistic model to predict nature of polymer (Epoxy & Polyester) material, 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.
Experimental Work:

Manufacturing Procedures:

These procedures can be classified into three stages:

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

1. The upper glass: It is a piece of glass of (20cm) length, (20cm) width, and (0.6cm) thick.

2. The lower glass: It is exactly the same as the upper glass.

3. The rest: 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.

*Fig (1) The mold components*
Preparation of Material for Manufacturing

The preparation process includes mixing the resin polyester with (0.2-0.5% polyester) cobalt and (1.5-2% polyester) hardener, or mixing the resin epoxy. Performing the necessary measurements.

Description of Tension 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 110 mm long, 20 mm wide and 3.5 mm thick. The tabs were 37.5 mm long, 20 mm wide, 3.5 mm thick with a taper angle of 20° smoothly curved.

Specimen geometry is shown in Fig (2). The set of dimensions defines this specimen. These are the effective length of specimen $L_e$, the tentative length $L$, and width of the specimen $W$ [4].

![Fig (2) Tensile specimen geometry](image)
**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 member 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.
Experimental Results:

In this work, two types of resins are used. To explain the results obtained from experimental tests. Table (1) illustrates the properties of resins. Fig (4) shows the photos of the specimens used in this work. Table (2) illustrates the different responses of deflection between polyester and epoxy resin for the same type and with the same load.

Table (1): Experimental mechanical properties of the resin used in present work [5]

<table>
<thead>
<tr>
<th>Resin type</th>
<th>E (Gpa)</th>
<th>G (Gpa)</th>
<th>K(Gpa)</th>
<th>( \nu )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyester</td>
<td>1.0602</td>
<td>0.384</td>
<td>1.478</td>
<td>0.38</td>
</tr>
<tr>
<td>Epoxy</td>
<td>0.333</td>
<td>0.127</td>
<td>0.295</td>
<td>0.312</td>
</tr>
</tbody>
</table>

Table (2): The composite responses to deflection [5]

<table>
<thead>
<tr>
<th>Resin type</th>
<th>Load apply (gm)</th>
<th>Deflection (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyester</td>
<td>914</td>
<td>2.29</td>
</tr>
<tr>
<td>Epoxy</td>
<td>914</td>
<td>3.36</td>
</tr>
</tbody>
</table>

Fig (4)
(a) Epoxy before test.
(b) Epoxy after test
(c) Polyester before test
(d) Polyester after test
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 consist of a 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.

DETERMINISTIC FINITE ELEMENT ANALYSIS

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 a 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).

Fig.(5) The PLANE 82 Element Simulation
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 ($U_y = 0$), 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.

Analysis Results:

The linear and nonlinear finite element solutions are done using ANSYS 10.0, the small displacement static and line search nonlinear option are used. A $20^{th}$ are number of sub-steps with $10^{-4}$ value of tolerance. The results of nonlinear solution iterations for every sub-steps are shown in fig.(6).

POLYESTER TEST:

From experimental results, a load-deflection relation is shown in fig.(7), including the solution of linear and nonlinear finite element analysis. Good agreements of calculated and experimental results. Finite element solutions (linear & nonlinear) at initial have approximately same values, but the nonlinear solution curve has more accurate results compared with experimental curve results. According the polymer member is behaved as elastic nonlinear material, according to the tension softening model issued in finite element analysis. Fig.(8), shows the stress-strain relation ship of experimental results and finite element solutions. The polyester members are
tested, and modeled using finite element method have good verification in this paper. The contour results of stress in linear solution is shown in fig.(9).

Fig.(7) Load-Deflection relation of Polyester specimen

Fig.(8) Strain Stress Relation of Polyester specimen

Fig. (9) Stress in polyester using ANSYS contour results for linear FE solution
**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 relation ship of load-deflection. Good agreements are shown, and nonlinear solution have 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 agreements are shown. The Contour results using ANSYS for nonlinear FE solution of stress and strain are shown in fig.(12).

![Fig(10). Load-Deflection relation of Epoxy resin member](image1)

![Fig.(11) Strain-Stress relation of Epoxy resin member](image2)

![Fig. (12) Contour results using ANSYS for nonlinear FE solution](image3)
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 load of polymer member strength 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:


