Abstract-- Composite material is a combination of two or more materials on a macroscopic scale to form a useful material. Composite find tremendous application in marine, automobile, sports & recreation equipments etc. Hence a quantum of research is going on in order to find the improvements in existing combination of composites. One of the most promising composite in the current scenario is Carbon Fabric Reinforced Polymers (CFRP). The validation of a composite under varied flaw condition is also an important aspect and Finite element approach is one of the methods for such process.

In the current work an attempt has been made to evaluate the various stresses near the crack field using FEA approach. ANSYS 13 Version is used for modelling & analysis. The composite with three volume fractions & various crack configuration & orientation are chosen. From the study of Analytical & Numerical results it can be seen that the line crack is more prone to failure than any other crack configuration. The Analysis carried out on Elliptical crack with axis orientation along and across the direction of loading shows that crack oriented along the direction of loading fails at early levels than other orientation. It can be seen that Numerical results are in agreement with Analytical Results.

Keywords-- Composite Material, Finite Element Analysis, Fracture Mechanics, Stress Analysis

I. INTRODUCTION

Composite material is a combination of two or more materials on a macroscopic scale to form a useful material. Different materials can be combined on a macroscopic scale such as in alloying but the resulting material is macroscopically homogeneous. The advantage of composite material is that they usually exhibit the best qualities of their constituents. There are three different types of composites, which are commonly used in the present day engineering. They are:

- Fibrous composites; which consists of fibers in a matrix
- Laminated composites; which consists of layers of various metals
- Particulate composites; which are composed of particles in a matrix
One of the most common ways of expressing the strength and stiffness of a material is a ratio of either of the quantities to the density i.e. weight per unit volume. Such an index does not include the cost to achieve strength or stiffness, but cost comparisons might not be valid by themselves since many factors influence cost.

II. FRACTURE MECHANICS

Fracture mechanics deals with the study of how a crack or flaw in a structure propagates under applied loads. Cracks and flaws occur in many structures and components, sometimes leading to disastrous results. The engineering field of fracture mechanics was established to develop a basic understanding of such crack propagation problems.

To find stress field in the vicinity of the crack tip, it is important to realize that general state of deformation at the crack tip can be considered in terms of three basic modes of deformation identified in terms of the mode of crack surface displacement. The three basic modes are generally referred as Mode I, Mode II, and Mode III. The present problem is associated with Mode I type failure. Mode I type failure refers to a pure opening mode in which the crack tip opens symmetrically with respect to their position under the load, known as opening mode.

Classifications of Fracture problems

Plane stress problem

A class of common engineering problems involving stresses in a thin plate or on the free surface of a structural element, such as the surfaces of thin-walled pressure vessels under external or internal pressure, the free surfaces of shafts in torsion and beams under transverse load, has one principal stress that is much smaller than the other two. By assuming that this small principal stress is zero, the three-dimensional stress state can be reduced to two dimensions. Since the remaining two principal stresses lie in a plane, these simplified 2D problems are called plane stress problems.

Plane strain problem

Some common engineering problems such as a dam subjected to water loading, a tunnel under external pressure, a pipe under internal pressure, and a cylindrical roller bearing compressed by force in a diametral plane, have significant strain only in a plane; that is, the strain in one direction is much less than the strain in the two other orthogonal directions. If small enough, the smallest strain can be ignored and the part is said to experience plane strain.

 Modes of Failure

To find stress field in the vicinity of the crack tip, it is important to realize that general state of deformation at the crack tip can be considered in terms of three basic modes of deformation identified in terms of the mode of crack surface displacement. The three basic modes are generally referred as Mode I, Mode II, and Mode III as shown in figure: 3

Classifications of Fracture problems

Plane stress problem

A class of common engineering problems involving stresses in a thin plate or on the free surface of a structural element, such as the surfaces of thin-walled pressure vessels under external or internal pressure, the free surfaces of shafts in torsion and beams under transverse load, has one principal stress that is much smaller than the other two. By assuming that this small principal stress is zero, the three-dimensional stress state can be reduced to two dimensions. Since the remaining two principal stresses lie in a plane, these simplified 2D problems are called plane stress problems.

Plane strain problem

Some common engineering problems such as a dam subjected to water loading, a tunnel under external pressure, a pipe under internal pressure, and a cylindrical roller bearing compressed by force in a diametral plane, have significant strain only in a plane; that is, the strain in one direction is much less than the strain in the two other orthogonal directions. If small enough, the smallest strain can be ignored and the part is said to experience plane strain.

Stress equations for Mode-I type of failure near the crack tip are

\[ \sigma_x = \sigma \left( \frac{\pi}{2\lambda} \right) \cos \left( \frac{\theta}{2} \right) \left[ 1 - \sin \left( \frac{\theta}{2} \right) \sin \left( \frac{3\theta}{2} \right) \right] \]  

(1)

\[ \sigma_y = \sigma \left( \frac{\pi}{2\lambda} \right) \cos \left( \frac{\theta}{2} \right) \left[ 1 + \sin \left( \frac{\theta}{2} \right) \sin \left( \frac{3\theta}{2} \right) \right] \]  

(2)

\[ \tau_{xy} = \sigma \left( \frac{\pi}{2\lambda} \right) \sin \left( \frac{\theta}{2} \right) \cos \left( \frac{3\theta}{2} \right) \]  

(3)
III. FINITE ELEMENT METHOD

The finite element method has evolved as a specialized technique for the general numerical solution method applicable to a broad range of physical problems. In general, the finite element method is based on a theory whereby an original object is viewed as an assembly of discrete building blocks called elements. The application of the method involves dividing the body into an optimum number of blocks. These blocks are connected to each other at specified points known as nodes, forming a network called mesh. The number of elements is determined by two factors; the capability of the computer being used and the accuracy of the results.

The realistic use of finite element method for solving the problem involves pre-processing, analysis and post processing. The pre-processing involves the preparation of data such as nodal coordinates, element connectivity, boundary conditions, loading and material properties. The analysis stage involves stiffness generation, stiffness modification and solution of the equations resulting in the evaluation of nodal displacements and derived quantities such as stresses. The post processing deals with presentation of the results. A complete finite element analysis is a logical interaction of the three stages. The systematic development of input data file is presented in the chapter. A mesh generation procedure is developed to generate the element topology such as number of nodes, number of elements, nodal coordinate positions and global node numbers associated with each element.

Objectives Of The Present Paper

The main objective of the present paper is

- To study the effect of stress field around the different defects like line crack, circular hole, elliptical hole and oval in a composite system for different volume fractions.
- To study the behaviour of singularity and oscillatory nature of the stress field near the crack tip.
- To study the effect of stress fields around these defects front-end Finite element commercial software ANSYS is used to carry out parametric analysis for different volume fractions.

IV. PRESENT SCHEME OF INVESTIGATION

Main objective of the paper is to study the effect of stress field around the defect and stress at the matrix and fiber interface to check the debonding of the composite system. As a major part of paper a series of finite element analyses are to be carried out on different defects in composite structures subjected to uniaxial loading on a micro-mechanical scale. The following steps to be followed to carry out the analysis.

Modelling of a composite system for different defects

The details of the composite unit considered for the analysis is as shown in Figure: 4 in which the fibers are aligned parallel to the direction of loading, similar models with the different defects like line, circular, elliptical and oval defects are to be modelled for finite element analysis.

![Figure: 4 Composite system considered for FE Analysis with elliptical crack](Image)

V. FINITE ELEMENT ANALYSIS OF COMPOSITE STRUCTURE

The problem basically consists of analyzing a composite material system under uniaxial tensile loading, the dimensions being shown in Figure: 6. The initial analysis was carried out for a volume fraction of 20%. This problem is solved under plane stress condition assumption. Since the geometry and the boundary conditions are symmetric about the Y-axis, half portion of composite is modelled for plane stress analysis as shown in Figure: 6. Finite element meshing is created for the half model. The meshing consists of structural 6-noded triangular elements with mid nodes for simulating singular effect near the vicinity of crack tip and the rest of the model consists of structural 8-noded quadratic elements. The defect is modelled according to the dimension of the fiber. To obtain accurate results, linear mesh is introduced at the sensitive portion of the model that is at the crack tip region. Hence near the crack-tip area denser mesh is created. In this problem, three levels of meshing are used. Starting from coarser to a highly denser mesh towards the crack tip area. After meshing is completed, symmetric boundary conditions are applied. For denser mesh manually adaptive meshing is used in Figure: 7.
It can be seen transformation for sub-division-to-subdivision per element is done. Mesh is denser near the vicinity of the crack area and the areas that are farther from crack tip the mesh is coarser.

For each level of meshing to satisfy symmetry conditions adaptive meshing is done. Finally after completing meshing process all the nodes are merged and compacted. All the elements are also compacted. And at each step of merging, a boundary check is made which will show the boundary of model including crack. After completion of creating the FEM model and defining all its geometric properties, it should be converted into the form where it behaves like a real structural component. Its material property is to be defined. Following are material properties of the fiber and a matrix.

**Material properties**

The following properties of isotropic Carbon fiber and isotropic Epoxy materials are taken from values reported in the literature [3] and used in the current investigation.

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus, GPa</th>
<th>Poisson’s Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Carbon Fiber</td>
<td>395</td>
<td>0.13</td>
</tr>
<tr>
<td>Epoxy Matrix</td>
<td>3.7</td>
<td>0.35</td>
</tr>
</tbody>
</table>

**Boundary Conditions**

In this step, the configuration is applied with its necessary boundary conditions. The displacement and load boundary conditions applied are,

- Nodes lying at bottom of model are constrained in both X and Y directions.
- Nodes lying at top surface of models are coupled to move together in Y direction.
- Pressure of 50 N/m² is applied on the top surface of the model.
- The loading and boundary conditions with elements are depicted as shown in the figure: 7

**Summary of the Problem**

- Number of elements used : 20000
- Number of nodes used : 60609
- Number of degree of freedom : 1, 21,218
- Number of constrained degrees of freedom : 200
- Number of active degrees of freedom : 1, 21,018
- Number of active degrees of freedom : 1, 21,018

**Solving the problem**

To solve the problem follow the steps:

- Save the backup copy of the database to a named file.
- Start solution calculations
- Leave the solution processor

**Reviewing the Results**

A result from composite system analysis consists mainly of stresses, strains, displacements and plotting of graphs. This information reviewed in General post processor.

**VI. RESULTS AND DISCUSSION**

The result consist of two parts

- Stress contours
- Graphical distribution of stresses along the interface and crack tip
- Shear stress tables for different defects for 20% volume fraction

First set of results were drawn based on line crack with different volume fraction in this paper. Figure: 8 shows the stress distribution along the crack front for a line crack with different volume fractions in X - direction. Initially it is compressive in nature as it proceeds away from the crack it will be constant irrespective of the volume fraction.
Figure: 8 Normal stress ($\sigma_x$) along the crack tip in line crack for different volume fractions

Figure: 9 Normal stress ($\sigma_x$) along the crack tip in line crack for different volume fractions

Figure: 9 show the stress distribution in a line crack for different volume fractions along the crack front in $Y$-direction. It is observed that a maximum stress of 1329.2 MPa for 20% volume fraction and maximum stress were observed in the fiber because of the volume fraction of the fiber as the volume fraction increases stress near the crack tip goes on decreases and same trend were observed in the fiber.

To validate finite element software theoretical shear stress along the crack tip were determined based on fracture mechanics approach for Mode-I type of failure using equation 3 and figure:12 it indicates that shear stress is a function of angle $\theta$, radius $r$ and length of the crack ‘$a$’. If $\theta = 0$ shear stress along the crack tip is ZERO irrespective of the volume fraction and theoretical results were listed in the table 2 and finite element results for shear stress for different volume fractions along the crack tip is as shown in figure: 10. It is noticed that shear stress calculated using the Fracture mechanics approach were in good agreement with Finite element results. Results obtained from FEA are approximate and small variations were minimized by increasing the number of elements near the crack tip. Hence parametric analyses were carried out for different volume fractions and defects.

**TABLE II**

<table>
<thead>
<tr>
<th>Sl. No.</th>
<th>Line crack</th>
<th>Circular defect</th>
<th>Oval defect</th>
<th>Elliptical defect</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-0.08285</td>
<td>-0.07565</td>
<td>-0.03758</td>
<td>-0.18301</td>
</tr>
<tr>
<td>2</td>
<td>-0.12213</td>
<td>-0.12022</td>
<td>-0.12943</td>
<td>-0.03019</td>
</tr>
<tr>
<td>3</td>
<td>-0.16247</td>
<td>-0.16435</td>
<td>-0.16686</td>
<td>-0.1598</td>
</tr>
<tr>
<td>4</td>
<td>-0.18614</td>
<td>-0.18916</td>
<td>-0.18972</td>
<td>-0.18785</td>
</tr>
<tr>
<td>5</td>
<td>-0.19233</td>
<td>-0.19465</td>
<td>-0.19524</td>
<td>-0.19399</td>
</tr>
<tr>
<td>6</td>
<td>-0.18274</td>
<td>-0.18439</td>
<td>-0.185</td>
<td>-0.184</td>
</tr>
<tr>
<td>7</td>
<td>-0.15845</td>
<td>-0.15965</td>
<td>-0.16018</td>
<td>-0.15939</td>
</tr>
<tr>
<td>8</td>
<td>-0.12305</td>
<td>-0.12393</td>
<td>-0.12437</td>
<td>-0.12375</td>
</tr>
<tr>
<td>9</td>
<td>-0.07911</td>
<td>-0.07977</td>
<td>-0.08011</td>
<td>-0.07965</td>
</tr>
<tr>
<td>10</td>
<td>-0.02988</td>
<td>-0.03037</td>
<td>-0.03062</td>
<td>-0.03029</td>
</tr>
<tr>
<td>11</td>
<td>0.021301</td>
<td>0.020932</td>
<td>0.020769</td>
<td>0.020977</td>
</tr>
<tr>
<td>12</td>
<td>0.071028</td>
<td>0.070758</td>
<td>0.070676</td>
<td>0.070774</td>
</tr>
<tr>
<td>13</td>
<td>0.11613</td>
<td>0.11594</td>
<td>0.11593</td>
<td>0.11593</td>
</tr>
<tr>
<td>14</td>
<td>0.15324</td>
<td>0.15311</td>
<td>0.15317</td>
<td>0.15309</td>
</tr>
<tr>
<td>15</td>
<td>0.17926</td>
<td>0.17919</td>
<td>0.17929</td>
<td>0.17915</td>
</tr>
<tr>
<td>16</td>
<td>0.19196</td>
<td>0.19192</td>
<td>0.19205</td>
<td>0.19187</td>
</tr>
<tr>
<td>17</td>
<td>0.188</td>
<td>0.18799</td>
<td>0.18813</td>
<td>0.18794</td>
</tr>
<tr>
<td>18</td>
<td>0.16701</td>
<td>0.16701</td>
<td>0.16714</td>
<td>0.16696</td>
</tr>
<tr>
<td>19</td>
<td>0.12785</td>
<td>0.12786</td>
<td>0.12796</td>
<td>0.12782</td>
</tr>
<tr>
<td>20</td>
<td>0.071291</td>
<td>0.071296</td>
<td>0.071351</td>
<td>0.071276</td>
</tr>
<tr>
<td>21</td>
<td>0.000173</td>
<td>0.000173</td>
<td>0.000173</td>
<td>0.000173</td>
</tr>
</tbody>
</table>
TABLE II shows the distribution of shear stress near the crack tip for different defects for 20% volume fraction, irrespective of the defect shear stress along the crack front is Zero.

A Normal and shear stress contour plot for a line crack as is shown in figure: 11 and it is observed maximum stress near the crack tip. It is evident that if any defect present in a structure leads to discontinuity hence stresses near the discontinuity were maximum.

Similar analysis were carried for different volume fractions and defects and identified that Line crack in a composite is more serious compared to circular, elliptical and oval dislocations. Hence care should be taken during the manufacturing of the composite system.

Interface stress plays very important role, hence stress analysis along the interface in a composite system is very important to check whether the load is transferring from matrix to fiber. From the following stress graphs it is observed that stress is transferred from matrix to the fiber and stress is oscillatory singularity in nature if any voids present in the system within the elastic region. And shear stress is more oscillatory compared to normal stress.
VII. STRESS CALCULATIONS USING FRACTURE MECHANICS APPROACH

Equations listed in the section II were used to calculate the stress field near the crack tip. Details and position of the point at which stresses calculated are shown in figure 15.

![Figure 15 Position of point near the crack tip](image)

Let ‘a’ is length of crack it 10 mm, X and Y are the coordinates of the point 10 mm and 2.5 mm from the crack end, ‘r’ is the diagonal length and ‘θ’ is the angular position of the point. These parameters can be determined by using the following equations

\[ \theta = \tan^{-1} \frac{y}{x} \]
\[ r = \sqrt{x^2 + y^2} \]

<table>
<thead>
<tr>
<th>Stress</th>
<th>Theoretical MPa</th>
<th>FEM MPa</th>
<th>% Deviation</th>
</tr>
</thead>
<tbody>
<tr>
<td>Normal stress ( (\sigma_x) )</td>
<td>13.20</td>
<td>12.12</td>
<td>8.18</td>
</tr>
<tr>
<td>Normal stress ( (\sigma_y) )</td>
<td>14.43</td>
<td>14.40</td>
<td>0.21</td>
</tr>
<tr>
<td>Shear stress ( (\tau_{xy}) )</td>
<td>1.57</td>
<td>1.55</td>
<td>1.27</td>
</tr>
</tbody>
</table>

TABLE III
Theoretical and FEM results for line crack for 20% volume fraction

From the available literatures and results obtained from the finite element are compared and small variations in the results can be minimized with increase in the order of the polynomial and number of elements and hence stress analysis of composite system using traditional methods can be eliminated with the help of FEA software.

VIII. CONCLUSIONS

Composite materials are finding increasing application in the present day engineering design and development activities particular in aerospace, marine and automobile engineering field. This is because the offer very good attractive mechanical properties such as high strength to Wight ratio, high stiffness to weight ratio, higher thermal and wear resistance, corrosion resistance, etc.

The presence of defects in composite material will lead to deterioration in their mechanical properties, particularly their tensile strength and fatigue life. The stress field around the crack will be singular in nature. It has been observed by both analytical and computational investigation that the stress field very near the vicinity of the crack is oscillatory nature.

REFERENCES
[2] Lubos Nahlík, Zdeněk Knesl, Jan Klusák, Crack Initiation criteria for Singular Stress Concentrations, 2008,
[3] Rice and Johnson Influence of the crack tip model on results of the finite elements method
[9] Dowd et al., Influence of the crack tip model on results of the finite elements method, 1995