FEM INVESTIGATION OF THE FIBER REINFORCED COMPOSITE PRESSURE TUBE UNDER IMPACT LOADING

Nihat AKKUS1,a, Garip GENC2,b, Ersin TOPTAS1,c

1Marmara University, Faculty of Technical Education, Istanbul, Turkey
2Marmara University, Vocational School of Technical Sciences, Istanbul, Turkey
a nakkus@marmara.edu.tr, bgenc@marmara.edu.tr, c etoptas@marmara.edu.tr

ABSTRACT

The effect of impact loading on Carbon Fiber Reinforced Plastic (CFRP) composite tube without and with high internal pressure has been investigated by FEM simulation in the present study. Composite tube which has an Aluminum cylinder and wound by carbon fiber was simulated by using 3-D model. MARC-Mentat commercial code has been selected as a computational tool. The geometry of the cylinder has been generated using Mentat pre-post interface software. The material properties of the Aluminum 6061T liner of and TORAY T-700SC + Epoxy composite layers have been included in the simulations as an isotropic and orthographic, respectively. The dome region of the vessel has not been included in the simulations. The maximum displacement of the composite part which was just under impactor has been divided into several steps during the simulations to check the progress of the impactor damage. The model of the simulation has been created by considering the real experiment which has been conducted by other researcher. The results of the simulations show that some zones under the impactor damage would face compressions stresses rather that tensional deformation, which is generally believed.

Keywords: Pressure vessel, Composite, FEM, Ballistic, Tube, Impact loading

1. Introduction

Resistance to high velocity impact is an important requirement for high performance structural materials. Even though, polymer matrix composites are characterized by high specific stiffness and high specific strength, they are susceptible to impact loading. For the effective use of such materials in structural applications, their behavior under high velocity impact should be clearly understood. For this reason this topic is the interest of researchers.

Naik et al. [1] presented investigation on the ballistic impact behavior of two-dimensional woven fabric. Tan et al. [2] presented an FE model of woven fabric that reflects the orthotropic properties of the fabric, the viscoelastic nature of the yarns, the crimping of the yarns, the sliding contact between yarns and yarn breakage using an assembly of viscoelastic bar elements. Duan et al. [3] presented a finite element analysis to study the influence of friction during ballistic impact of a rigid sphere onto a square fabric panel that was firmly clamped along its four edges and projectile-fabric friction and yarn–yarn friction were investigated. Fawaz et al. [4] presented an effective methodology for the optimum design of two-component armours using Florences model and a new hybrid solving technique. Übeyli et al. [5] presented that the ballistic performance of steel against 7.62 mm armor piercing projectiles. Lopresto et al. [6] investigated that low-velocity impact tests were carried out on stitched carbon fiber-reinforced plastic laminates of various thicknesses, whose behavior was studied with reference to the overall force–displacement curve, first failure load, penetration, indentation and damage extent. Duan et al. [7] studied a 3-D finite element analysis model using LS-DYNA to simulate the transverse impact of a rigid right circular cylinder onto a square patch of plain-woven Kevlar fabric. Mamalis et al. [8] studied that in the simulation works the LS-DYNA 3-D explicit finite element code is used to investigate the compressive properties and
crushing response of square carbon fiber reinforced plastic tubes subjected to static axial compression and impact testing. Abdullah et al. [9] investigated the high velocity impact response of a range of polypropylene-based fiber–metal laminate (FML) structures. Vaidya et al. [10] studied that the vibration response of composite sandwich plates composed of laminate face sheets and aluminum foam core was also studied under a free–free boundary condition and the vibration response (natural frequency and damping ratio) is reported as a function of impact to the sandwich plate. Caprino et al. [11] studied that four stitched graphite/epoxy laminates of different thicknesses were subjected to high-velocity impact tests. Two steel spheres, 12.7 and 20 mm in diameter, were used as bullets during the tests, carried out at two different speeds (65 and 129 m/s).

As seen above, most of the studies concentrated on low speed ballistic impact and quasi-static loading both of which is far from real situations and damages occurs under high velocity impact loading of cylinders. Some studies which are conducted under high speed used laminates instead of cylinders. A FE model has been proposed in the present study in which the speed of impactor can vary from low speed to high speed loading in Al + CFRP composite tube. The FE model has been compared to experiment in terms of Al and composite deformation which is happened after impactor made an indentation on tube. This was to understand the reliability of FE model.

2. Finite Element Modeling of Tube under Impact Loading

The Al + FRP composite tube under impact loading has been modeled using MSC Marc Mentat finite element simulation program [12, 13]. The platform for software was a desktop computer with 2 GHz speed with two processor and Windows XP operating system.

The size and materials properties of the tube were obtained from an earlier study made by Takekusa et al. [14]. The dimensions and the schematic drawing of composite tube is given in Fig. 1. The material properties of Carbon fiber and Aluminum tube are given in Table 1. Aluminum material was assumed as isotropic and carbon fiber + epoxy composite assumed as orthotropic material. The volume fraction of the fiber (Vf) to epoxy was 0.53. The dimensions of the Al-6061 tube are given in Fig. 1. The model was generated with Mentat pre-post processor. The geometry was created with 3960 solid element 6919 nodes. A 3-D solid element number-7 in Marc element library was chosen. Fiber orientation, thickness distribution, stacking sequence and number of layers were the parameters used to describe the composite structure.

![Fig. 1 Schematic drawing of composite tube](image_url)

The nodes at the one end of the tube were fixed along X axis and the nodes located at the other end of the tube were left free along X axis deformations. All the nodes which are located at both end of tube were fixed in Y and Z direction to make the boundary conditions are similar to that of experiments. The boundary conditions and the rigid ball which is simulating the impactor are shown in Fig. 2 (a). The deformed shape of tube after impact loading in simulation is also given in Fig. 2 (b).
The outer diameter of Al-tube is 100 mm, the thickness is 3 mm, and wounded carbon fiber reinforced polymer (CFRP) thickness 1.2 mm and length of tube 200 mm. Hoop winding was alone employed to cover Aluminum tube with carbon fiber composite and no other winding such as helical winding used. To validate the reliability of the FE model, the experimental parameters which is explained in reference [14] was inputted to the FEM simulation and the same depth of deformation on the composite and aluminum tube has been obtained. This indentation was caused by single-bounce impact tests which have been conducted on Al + CFRP composite cylinders in the same reference. Fig. 3 shows the comparison of the simulation with experiment with respect to the deformation depth.

It is very difficult to obtain different data during impact testing because of high speed of impactor and short testing time. High speed testing also needs more sophisticated and expensive equipment to obtain experimental data. Thus it is very useful to make simulation to see the stress and strain variation of composite tube once the reliability of the FE model is understood.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>A6061-O</td>
<td>68.6</td>
<td>&lt; 110</td>
<td>&lt; 145</td>
<td>&gt; 16</td>
</tr>
<tr>
<td>T-700-24K</td>
<td>230</td>
<td>-</td>
<td>4900</td>
<td>2.1</td>
</tr>
</tbody>
</table>

The outer diameter of Al-tube is 100 mm, the thickness is 3 mm, and wounded carbon fiber reinforced polymer (CFRP) thickness 1.2 mm and length of tube 200 mm. Hoop winding was alone employed to cover Aluminum tube with carbon fiber composite and no other winding such as helical winding used. To validate the reliability of the FE model, the experimental parameters which is explained in reference [14] was inputted to the FEM simulation and the same depth of deformation on the composite and aluminum tube has been obtained. This indentation was caused by single-bounce impact tests which have been conducted on Al + CFRP composite cylinders in the same reference. Fig. 3 shows the comparison of the simulation with experiment with respect to the deformation depth.

It is very difficult to obtain different data during impact testing because of high speed of impactor and short testing time. High speed testing also needs more sophisticated and expensive equipment to obtain experimental data. Thus it is very useful to make simulation to see the stress and strain variation of composite tube once the reliability of the FE model is understood.

![Fig. 2](image)

**Fig. 2** (a) The boundary conditions and (b) the deformed shape in FEM simulation

**Table 1.** Material properties of aluminum liner and fiber material

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>A6061-O</td>
<td>68.6</td>
<td>&lt; 110</td>
<td>&lt; 145</td>
<td>&gt; 16</td>
</tr>
<tr>
<td>T-700-24K</td>
<td>230</td>
<td>-</td>
<td>4900</td>
<td>2.1</td>
</tr>
</tbody>
</table>

The outer diameter of Al-tube is 100 mm, the thickness is 3 mm, and wounded carbon fiber reinforced polymer (CFRP) thickness 1.2 mm and length of tube 200 mm. Hoop winding was alone employed to cover Aluminum tube with carbon fiber composite and no other winding such as helical winding used. To validate the reliability of the FE model, the experimental parameters which is explained in reference [14] was inputted to the FEM simulation and the same depth of deformation on the composite and aluminum tube has been obtained. This indentation was caused by single-bounce impact tests which have been conducted on Al + CFRP composite cylinders in the same reference. Fig. 3 shows the comparison of the simulation with experiment with respect to the deformation depth.

It is very difficult to obtain different data during impact testing because of high speed of impactor and short testing time. High speed testing also needs more sophisticated and expensive equipment to obtain experimental data. Thus it is very useful to make simulation to see the stress and strain variation of composite tube once the reliability of the FE model is understood.

![Fig. 3](image)

**Fig. 3** Comparison of experiments with simulation in term of indentation depth. (a) The deformed shape after impactor hit composite tube [14] (b) The displacement vs step number in simulation.
3. Results of FEM Simulation

The effect of the surface damage on the strength of a filament wound an (FW) gas cylinder which subsequently causes loss of stiffness and strength resulting from accidental impact in fiber composite structures remains of much concern. Although a number of previous investigators have delineated the major features of the problem, important details remain uncertain. Although a number of studies have been carried out on impact of flat composite plates, relatively little work has been done on tubular geometries such as pressure vessels despite their usage in applications.

FEA provides a good approach to the analysis of composite cylinders under high internal pressure to understand how the stress distribution of cylinder changes. Fig. 4 shows displacement along tube longitudinal axis after indentation. All the displacements along longitudinal axis of tube with no internal pressure (continues-solid line) are always under minus direction. This means all the cylinder body is more affected when it is empty. But the ends of the cylinder are in plus area when cylinder has 50 MPa internal pressure. Comparing the length of the curves along longitudinal direction indicates that the cylinder with 50 MPa internal pressure has more longitudinal deformation. This is important how to approach and propose analytical solutions when composite cylinder under impact loading is considered. As it is well known the compression strength is considerably different than tensional strength in terms of fiber reinforced composites. Comparison of curves of CFRP and Al reveals that Aluminum tube has the same behavior like CFRP. This indicates that aluminum behaves like same as CFRP since the displacement are similar to each other. But, it should be considered that Al mechanical properties are quite different than CFRP, thus attentions should be paid when Al liners of composite high pressure vessels are subjected to fatigue life calculations.

Von Mises stress distribution of Al cylinder in terms of no internal pressure and with 50 MPa internal pressure is given in Fig. 5. The stress difference between tube end and the middle of tube where impactor deformations occurred is higher in low internal pressure and low in higher internal pressure. But the end of the tube has almost the same stress as impactor affected area. This should be carefully considered while the tube is designed.

Generally, designing the cylinders under internal pressure needs only tensional mechanical properties. Fig. 6 shows that compression and tension occurs at the same time during impact loading of cylinder. Considering that tensional and compression mechanical properties are quite different in orthographic materials This figure also reveals that the deformation immediately under the impactor is bending + tension in the longitudinal direction but become compression in the circumferential direction. Further away from the impactor towards to the end of cylinder, tensional stress become active in longitudinal and circumferential direction.

![Fig.4. Displacement on the FRP materials and Aluminum](image)
Fig. 5 Equivalent Von Mises Stress on the Aluminum materials

Fig. 6. Longitudinal stress distribution of cylinder with no pressure and 50 MPa internal pressure after impact

Fig. 7. Circumferential stresses in CFRP layer in the cylinder longitudinal direction

Circumferential stress distribution of CFRP composite material with 50 MPa internal pressure and without internal pressure is given in Fig. 7. This figure shows that compression stress occurs where the cylinder body was hit by impactor. It has been explained in reference [13] that some fiber breakage would occur depending on the speed of the impactor. This may cause stress redistribution of tube drastically and may result severe plastic deformation in Al tube central part. The effect of autofrettaged to make the Al tube fatigue life longer may become ineffective too.
4. Concluding remarks

A FE model to simulate the effect of impact loading on Al + CFRP composite cylinder with high internal pressure has been investigated in the present study. The reliability of the simulation model has been controlled by comparing the real experiment which has been conducted by other researcher the results of simulations revealed that some of the cylinder parts which is generally believed and designed to work under tensional stresses may under go compression stress under impact loading. Further studies will be conducted to understand the details of Al + CFRP composite cylinder under impact loading especially to predict the fatigue life of damaged high pressure vessels.

5. References