Aluminium Lined, Carbon Composite Overwrapped Pressure Vessel

Yunusa Mujaheed\textsuperscript{a}, Zu Lei, Chen Dan and Tapa Arnauld Robert

School of Materials Sciences and Engineering, Wuhan University of Technology, Wuhan 430070, China

\textsuperscript{a}Corresponding author: Ekamuyaheeda@yahoo.com

Abstract. A numerical analysis of filament-reinforced internally pressurized cylindrical vessel with over-wrapped metallic liner is presented, the study deals with simulation of aluminium liner pressure vessel overwrapping by composite T800/epoxy was selected, ABAQUS Software was use in which 20 layers of composite were analysed. The stress state of the pressure vessel under the internal pressure of 33MPa was analysed, and the bursting strength was predicted to be 92.3MPa. MATLAB software was used to simulate the winding process, the 5 axis numerical control winding machine was used for the actual winding, and the fibres were arranged in a regular and uniform way, and the sliding phenomenon was not occurred. After solidifying, the bursting strength was 85.4MPa, and the difference between the actual value and the predicted value was 6.9%. The experimental results were in good agreement with the simulation results.

1. Introduction

Composites pressure vessels are widely used for many applications. Their light weight is a major benefit for portable uses, such as medical gases for health care applications using vessels volumes in the range 1-5L. On the other hand, their high mechanical strength allows storing very high pressure gas, which make the particular suitable for transportation of large quantity from a production to distribution site, using vessel volumes higher than 150L. The combination of these two advantages lead to become a standard technology for 700 bar hydrogen (CGH2) or 200 bar natural gas (CNG) vehicle using on board storages 50-150 L.

For all these applications, it paramount to ensure the structural integrity of the vessels during their entire lifetime, mechanical damage such as impacts or drops is one of the most feared [1]; as it is well known that composite materials are very sensitive to transverse impacts [2].

Pressure vessels have long been manufactured by filament winding process. In this process, bands of resin impregnated fibre are wound over a cylindrical mandrel using a computer control fibre placement machine. Fibre application to the metallic mandrel is usually accomplished by a transverse feed head over a rotating surface, capable of achieving any desired winding orientation. Traditionally, composite analysis techniques rely on the assumption of plane stress, and therefore they are only applicable to thin laminates. Originally, many of the classical thin shell theories are based on Kirchhoff-Love hypotheses [3]. Composite vessel has been studied in this present work. The composite vessel is made of inside aluminium liner overwrapping with carbon fibre/Epoxcomposite. The analysis demonstrated that for thin-wall and
thick-wall single-layered cylinders, the optimum winding angle is in the range of 54°-57°. Xu et al. [4] utilized finite element analysis to predict burst pressure of the composite storage vessel. In this present research attempt have been made to:

- Determine the burst pressure and the maximum shell displacement for asymmetric fiber orientations of composite vessel.
- Determine optimum winding angle according to diverse failure criteria.

2. Theoretical Studies

2.1. Optimum Winding Angle

Netting analysis is one of the most popular analytical techniques used for investigating the behaviour of multi-layered composite materials, especially for filament wound pressure vessels. The main assumption in netting analysis is that all loads are carried by the fibres neglecting the stiffness of the matrix and internal pressure subjected to the vessel produces a hoop-to-axial-stress ratio of 2:1 [5] Figure 1 indicates the thin-wall cylindrical vessel with thickness of “t” and length of ‘L’ which is subjected into the internal pressure “p”. In order to calculate the axial (longitudinal) and hoop stresses at the cylinder with the radius of “R”, Eq. (1) can be utilized [6]:

\[ N_\theta = \frac{PR}{2}, N_\phi = PR \]  

(1)

Axial and hoop stress can be calculated as in Eq. (2)

\[ \sigma_{\text{axial}} = \frac{N_\phi}{t} = \frac{PR}{2t}, \sigma_{\text{hoop}} = \frac{N_\theta}{t} = \frac{PR}{t} \]  

(2)

Axial and hoop stresses can be calculated according to equilibrium across the cut section too as Eq. (3) and (4)

\[ PL(2R) = 2\sigma_{\text{hoop}}Lt \rightarrow \sigma_{\text{hoop}} = \frac{PR}{t} \]  

(3)

\[ P\pi r^2 = \sigma_{\text{axial}}(2\pi Rt) \rightarrow \sigma_{\text{axial}} = \frac{PR}{2t} \]  

(4)

![Figure 1. Thin cylindrical shell](image-url)
Optimum winding angle can be estimated by using ultimate tensile strength along with applying netting analysis respect to 2:1 hoop-to-axial stress ratio as it is shown at Eq.(5) and (6), as it is shown in figure.

\[ N_\phi = \sigma_u t \sin^2 \alpha, N_\theta = \sigma_u t \cos^2 \alpha \]  
\[ \frac{N_\phi}{N_\theta} = \tan^2 \alpha = 2 \rightarrow \alpha = \arctan(\sqrt{2}) = 54.7^\circ \]  

**Figure 2.** Schematic diagram of the aluminium lining structure

### 3. Simulation Details

The considered pressure vessel is the vertical one designed for low capacity applications. The vessel is sketched with 505 mm length, 166 mm diameter at the centre point and the wall thickness of (2.4-6) mm base on the geometry. The material of this cylindrical vessel consists of the aluminium alloy inside layer reinforced with composite material T800/epoxy of 20 layers. Table 1 and Table 2 indicates material property of CFRP and aluminum alloy respectively.

| Table 1. Basic mechanical properties of selected composite systems |
|---------------------------------------------------------------|
| Strength(Mpa) | Modulus(Gpa) | Poisson ratio (\(\nu_{12}\)) |
|---------------|--------------|---------------------|
| X_t = 2344.96  | E_xt = 144.89 | 0.33                |
| X_c = 1315.18  | E_xc = 138.04 |                    |
| Y_t = 58.07    | E_yt = 8.58  |                    |
| Y_c = 140.84   | E_yc = 8.18  |                    |
| S = 84.46      | G = 5.09     |                    |

**Table 2.** Mechanical properties of Al 6061 and CFRP [8]

|               | Density(kg/m\(^3\)) | E_t (Gpa) | E_x(Gpa) | \(\nu_{12}\) | G_{12}(Gpa) | t(Mpa) |
|---------------|----------------------|-----------|----------|-------------|-------------|--------|
| AL6061        | 2750                 | 70        | 70       | 0.3         | 27          | 600    |
| CFRP          | 1570                 | 135       | 8        | 0.27        | 3.8         |        |

### 4. Results and Discussion

#### 4.1. Matlab Results

During the winding process, the yarn starts from one end of the pole hole and passes through a complete loop back to a distance that is offset from the starting point by a yarn width; the lining is then filled through several complete cycles. This article uses the matlab software performs trajectory simulation on the yarn winding first based on mold geometry dimensions with MATLAB’s command box input header and geometric parameters of the barrel part are then build by code programming, geometric model of the lining. The equation establishes the geometric model of the winding trajectory.
Figure 3. Simulation diagram of the winding process considering the bandwidth. (a) Head winding process, (b) Overall winding process

When a complete cycle is over, the yarn is wound into the next stage. Figure 3 (a) intermediate graphics as shown, the webs start at position 1 and are arranged in a counter clockwise direction next to the hysteresis until position 5 is reached. From position 1 to position 5 the number of intermediate yarns, that is, the number of complete loops of the yarn evenly covered with the mold, and at this time the mold has been covered with two layers of overlapping fibres. The arrangement of the yarn after the winding is completed is shown in Figure 3 (a). The picture shows. As can be seen from the simulated perspective (b), the yarns are at positions 1, 2, 3, 4, 5 and 1', because the winding parameters and the number of tangent points are determined on the mold.' and 6', 5', 4', 3'There are overlaps at 2 the width of the aligned individual yarns in the circumferential direction is not necessarily an integer multiple of the circumference of the mold section. Fibre both the bandwidth and the die size are fixed, so this overlap is not effectively avoided.

4.2. Abaqus Result

Figure 4. Geometric model of the pressure vessel in ABAQUS

4.2.1. Finite Element Model. The finite element model of the composite pressure vessel is divided into two parts, the inner liner and the composite layer. First based on Figure 4. Shows the geometric model
of the lining in the ABAQUS software, considering the composite pressure studied in this paper the
symmetry of the force vessel in the structure, material and load, and the selection of the pressure
vessel to the whole 1/4 of the ring is limited met model.

The winding method of the pressure vessel is spirally and circumferentially wound in order, and the
layering method of the barrel section is set for \([11.0^0/98.9^0/11.0^0/98.9^0/11.0^0/98.9^0/11.0^0/98.9^0/11.0^0/98.9^0/11.0^0/98.9^0]\), the header section has a total of
10 layers. Composite material layup, its single layer thickness and the barrel section includes spiral
and ring direction total 20-layer composite layup, wherein the circumferentially wound single layer
has a thickness of 0.15 mm, which is slightly smaller than the spiral wound layer thickness. The
paving structure of the pressure vessel barrel section is shown in Figure 5.

The S4R shell element is selected as the basic unit of the composite layer, and the lining adopts the
C3D8R continuous entity unit. S4R is a linear quadrilateral curved shell unit and it is also a general
type of shell element, it is very adaptable, well it can be used for both the simulation of thick shell
problems and the simulation of thin shell problems. This unit can be defined, the direction of the local
material usually using the finite element method. The anisotropic material parameters of the composite
layer are defined according to the direction of the local material. So the unit can be used as a finite
element of the composite layer. The basic unit of analysis C3D8R is a 3D solid element and is also a
reduced integration unit with 8 Nodes, suitable for computational analysis of large deformations.
Applying the unit can guarantee the accuracy of a certain grid. Accurately solve the problem of large
distortion of the grid.

When meshing, you need to consider both the accuracy of the calculation and the efficiency of the
calculation in the pressure vessel model. The density of the grid is reasonably set in different parts,
appropriate meshing in areas where the calculated data changes greatly. The density is tighter, and the
area where the data changes little is calculated to appropriately reduce the mesh density.
4.2. Calculation Results and Analysis. A symmetric constraint is applied to the symmetry plane of the finite element model to limit the displacement of the model in the axial direction on the end face of the pole hole, and a binding constraint is imposed between the composite layer and the liner. Taking into account the self-tightening process, the load step is first loaded to the pre-tightening pressure [9] and then unloaded. Reloaded into the container for blasting, analysing the stress state of the pressure vessel when the second load is 33 MPa.

Figure 8. Pressure vessel meshing

Figure 9. Fibre circumferential layer stress distribution

Figure 10. Fibre spiral layer stress distribution
Analysis of the stress cloud Figure 9 and Figure 10 show that the composite layer has a higher stress level in the barrel section, and the stress level of the circumferential layer of the barrel section is higher than the stress level of the spiral layer, and the maximum stress of the circumferential layer at 809.0MPa, the maximum spiral layer stress is 485.0MPa. This shows the stress of the spiral layer and the circumferential layer. The horizontal difference is large, because the hoop tensile stress is greater than the axial stress during the compression process, and the fibre side the direction of the elastic modulus is the largest, and the ring is subjected to a larger tensile load, so the circumferential stress level is the highest.

![Figure 11. Load and boundary condition of the whole model](image)

![Figure 12. Lined PEEQ distribution](image)

Analyse the stress cloud Figure 11 and Figure 12 the lining also has a high stress level in the barrel section. Stress concentrations are created at the transition between the head and the barrel. This is because the lining has been reinforced in the head section. From the junction of the barrel and the head to the top of the head, the thickness is gradually thicker, and the thickness of the top is 2.5 times that of the barrel due to the particularity of the head structure, is such that the stress level is lower relative
to the barrel section when subjected to a load. Simultaneously, since the composite layer is a spiral plus circumferential layup structure, a structure is created in the transition section between the barrel body and the pole hole. The discontinuity causes the inner liner to create a stress concentration in this portion. The equivalent plasticity of the liner body, the maximum value of the change (PEEQ) is 6.040e03, and the head segment PEEQ=0. This shows that the inner lining of the barrel section has been in the yielding state while the head portion is still in an elastic state. At this time, the composite layer in the barrel section bears pressure load. Therefore, under the pressure of 33MPa after the pressure vessel is self-tightening, the circumferential direction of the composite layer is the main bearing. The structure is loaded, and the self-tightening process can give full play to the structural advantages of the composite material.

4.2.3. Prediction of Blasting Pressure. In this paper, the maximum strain criterion is used to predict the burst pressure of the pressure vessel, and the pressure capacity under the maximum strain criterion. The criterion for bursting pressure is [10]

$$\varepsilon_r = \varepsilon_f$$

In the middle $\varepsilon_r$ -maximum strain in the direction of the fibre in the hoop layer $\varepsilon_f$ -strain limit. A large number of practices and studies have shown [11, 12], circumferential composites during the blasting of composite pressure vessels the fracture strain of the layer is about 85.4% of the strain at break of the pure fibre, and the strain at break of the known pure fibre is $0.018 \times 0.854 = 0.01537$. Calculate the second applied load $\times$ Therefore, the fracture strain of the circumferential composite layer is 0.01537 the hoop strain in the process is plotted and the hoop strain under different loads is plotted.

Figure 13. Maximum strain versus load curve of circumferential composite layer

It can be seen from Figure 13 that when the maximum strain of the circumferential composite layer is 0.01537, corresponding to this time the strength is 92MPa. The curve in the figure does not pass the origin, which is due to the self-tightening action of the pressure vessel, resulting in the load. When the charge is 0, there is still strain in the circumferential composite layer. Circumferential composite when the load is 92Mpa the strain of the layer is shown in Figure 14.
Therefore, according to the maximum strain criterion, when the second water pressure is applied to 92 MPa, the pressure vessel is blasted in the barrel section.

5. Conclusions
(1) By analysing the stacking rule of the non-geodetic winding of the yarn in the head section, the thickness and angle of the head part are calculated. (2) The geometric model of the composite material is established according to the thickness value and the winding angle of different parallel circles, and the geometric model of the pressure vessel is established by combining the geometrical dimensions of the aluminium liner. Then input the basic mechanical properties of the composite material, the elastoplastic stress-strain parameters of the aluminium lining, and establish the structural model of the composite pressure vessel. (3) Select the appropriate structural units, mesh the structural model, and after applying the constraint, analyse the stress-strain state of the pressure vessel when it is loaded to 33 MPa for the second time. The maximum strain criterion is used to predict the burst pressure of the pressure vessel. The results show that the pressure vessel blasts when the internal pressure is 92 MPa. In the previous paper, the matching experiments of fibre and resin materials were carried out, and the optimal system was selected to test the basic mechanical properties of the selected composite system. Then, the non-geodetic winding linear pattern of composite pressure vessel was studied. According to the stable winding and uniform filling of two process conditions, the suitable winding type was determined. The finite element model was established and the pressure vessel was simulated. The force state predicts the burst strength. In this paper, the pressure vessel is wound and molded by inputting the non-geodetic winding parameters established above, and the accuracy of the simulated line type is verified. The blasting test is performed on the cured pressure vessel to verify the linear design theory based on non-geodes and based on the correctness of the finite element strength analysis method.

References
[1] P. Blanc-Vannet, "<Burst pressure reduction of various thermoset composite pressure vessels after impact on the cylindrical part.pdf>," 2017.
[2] A. K. Agrawal S, "Impact damage on fibre reinforced polymer matrix composite - a review," J Compos Mater, vol. 48, pp. 317-32, 2014.
[3] G. C. Rosato DV, "Filament winding, its development, manufacture, applications and design.," ed: New York: Interscience, 1964.
[4] Z. J. Xu P, Liu P, "Finite element analysis of burst pressure of composite hydrogen storage
vessels," Materials & Design, vol. 30, pp. 2295-01, 2009.

[5] G. A. G. Evans J T, Evans J & Gibson A Composite angle ply laminates and netting analysis vol. 458. Proceedings of the Royal Society of London: Mathematical, Physical and Engineering Sciences, 2002.

[6] A. Onder, O. Sayman, T. Dogan, and N. Tarakcioglu, "Burst failure load of composite pressure vessels," Composite Structures, vol. 89, pp. 159-166, 2009.

[7] O. A., "First failure pressure of composite pressure vessels " Master of Science Thesis, Graduate School of Natural and Applied Sciences Dokuz Eylül University., 2007

[8] Z. Z. Yingjun W, Minqing S & Siro Z . "Finite element modeling of carbon fiber reinforced polymer pressure vessel," presented at the International Conference on Educational and Network Technology (ICENT) 2010.

[9] DOT-CFFC Standard, Carbon-Fiber, Aluminum Cylinders[S], 2007.

[10] C. S. Shu JC, Chang JB An enhanced analysis method for composite overwrapped pressure vessels[C]. USA: AIAA publishers,, 1995.

[11] W. X. XU Minglin, AN Mingkang, "Prediction of bursting pressure of UHMWPE filament wound aluminum-lined composite gas cylinder[J]." Fiber Composite Materials, vol. 28, pp. 19-22, 2011.

[12] Z. X. Chang Xinlong, Liu Xinguo, et al. , "Finite Element Analysis and Blasting Pressure Prediction of Composite Gas Cylinders[J]." Journal of Rocket Propulsion, vol. 34, pp. 27-31, 2008.