Analysis of Laminated Composite Pipe Based in the Finite Element Method

Anju Verma[1], Apurba Mandal[2], Dungali Sreehari[3]

[1, 2, 3]National Institute of Technology, Uttarakhand, Srinagar Garhwal, 246174, India
vermaanju899[at]gmail.com

Abstract: ***A series of analysis has been done on damage and failure of carbon reinforced epoxy composite pipe made by filament winding process subjected to external pressure. The study is based on three models of composite pipe with varying the number of layers and winding angle. The models were based on ± 60_4, ± 60_6, ± 60_8, to analyze the effect of increasing the laminations on the composite pipe and the models based on varying the angles ± 45_4, ± 55_4, ± $65_4$to analyze the effect of varying the winding angle. ANSYS ACP is used in present studies to make the finite element model of the laminated composite pipe. Based on the FE result, it was observed that the stacking sequence play a vital role in improving the strength of the composite pipe. As the d/t ratio is less the composite pipe failed by buckling and as the ratio increase the failure is due to the delamination taking place in the composite. For the same winding angle, if the number of layers is increased, the strength of the composite increase and total deformation of the composite decreases. Optimum winding angle was 55^0 for the laminated composites.***

Keywords: ANSYS ACP, filament winding, winding angle, Inverse Reserve Factor (IRF)

1. Introduction

Composite materials find application in various fields like automobile, marine, aerospace, etc. due to its high strength to weight ratio[1]. In addition to the reduction in weight composites also have the advantage of improved manufacturing quality good surface finish, corrosion resistance, and dent resistance [2]. Laminated composite pipe is widely used in many engineering applications. Study of laminated composite pipes is carried out finding the optimum winding angle, stacking sequence, thickness diameter and other parameters for the loading conditions to get the best configuration [3].Composite pipes are utilized in remote operation vehicle(ROV) to screen risers, which present a constraint in the payload of about 150 kg (in water). In this manner, there is an interest to create lightweight structures which can sustain high external pressure which are corrosion resistance and it can also replace the metallic, ceramic material sometimes. To understand the best design under different loading conditions to get the failure mechanism and load-bearing capacity of the structure the finite element study is found in the literature.

Fabrication of composite materials is being traditionally done by filament winding method. Round shape components like pressure vessels are fabricated by this method as it produces a high strength, high stiffness, and good surface finish products. As the process is highly automated the production time is the less and the overall cost of the products is also reduced [4].

Analysis of composite pipes plays an important role as it depends on many factors in case of filament winding e.g. , the position of fibers, automation of the process, the volume fraction of fibers and the materials of the fiber [5]. When the compressive loading is applied the laminated tubes got deformed and when the load reaches its threshold value, it gets ruptured which could result in dangerous situations. If the thickness of the pipe is less than the failure is caused by buckling which is the phenomenon in which the pressure displacement graph shows sudden change. As the pipe thickness increased the failure is mainly caused due to delaminations which are caused by the adverse environmental conditions, improper transportations, and manufacturing defects [6]. Finite element analysis is a highly considered tool to reduce the trials used to get the best dimensions, stacking sequence, winding angle. There are many composite failure tools to predict the failure load and last ply failure is used in many kinds of literature to predict the failure load. Grove et. al. studied on the optimum winding angle on the filament wounded structures which concluded that the 55^0 is the best angle for the round structures or the pressure vessels[7].

In the present study, work is done based on epoxy carbon fiber composite pipe development in ANSYS 19 ACP module. The study is done by varying the angle and the number of layers subjected to external pressure and Composite failure theories based on Tsai-Wu is used to predict the failure load according to the inverse reserve factor (IRF).

2. Quadratic failure criteria

For fiber reinforced composite quadratic failure criteria is mainly applied. It combines both the stress and strain component into a second-degree polynomial which can be combined to get a plane stresses [8].

$$f = F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + F_{66}\tau_{12}^2 + 2F_{12}\sigma_1\sigma_2 + F_1\sigma_1 + F_2\sigma_2 \quad (1)$$

X_t= Tensile strength in direction of the fiber
X_c = Compressive strength in direction of the fiber
Y_t= Tensile strength in the transverse direction to fiber
Y_c = Compressive strength in the transverse direction of the fiber
S= Shear strength in the same plane
F_{11}, F_{22}, F_{66}, F_1, andF_2denote the material strength in principle material direction and F_{12} and these coefficient is calculated on the basis of failure theories.

Tsai Wu failure theory:

$$F_{11} = \frac{1}{X_t X_c}, F_1 = \frac{1}{X_t} - \frac{1}{X_c}, F_{22} = \frac{1}{Y_t Y_c}, F_2 = \frac{1}{Y_t} - \frac{1}{Y_c}, F_{66} = \frac{1}{S^2}, \quad (2)$$

$2F_{12} = XY$, *default*-1 (3)

So the equation can be given by

Regarding plane stress-strain, Inverse reserve factor is calculated as follows for Tsai Wu failure criteria [9].

$$f_{th} = \left(\frac{\sigma_1}{X}\right)^2 + \left(\frac{\sigma_2}{Y}\right)^2 - \frac{\sigma_1 \sigma_2}{X^2} + \left(\frac{\tau}{S}\right)^2 \quad (4)$$

Where ***X*** and ***Y*** signifies ultimate normal stresses in a direction along and perpendicular to fiber and ***S***is ultimate shear stress.

3. Formulation of the design problem

Objective: To design a composite pipe and predict its burst pressure

Constraints: $f_{th} \leq 1$, IRF ≤1 no failure

Variables: Number of layers, winding angle

4. Material Data

Carbon fiber epoxy is chosen as the material for structural analysis of composite pipe. Carbon fiber is mostly chosen as the reinforcement in filament wounded structures as it has good strength, and stiffness, also good fatigue resistance. Mechanical properties of the carbon fiber epoxy are taken from the ANSYS library which is given in Table 1.

Table 1: Properties of epoxy carbon material used in the design.

Properties	*Dimensionality*	*Values*
Density	g/cm^3	1.49
Young' modulus in X direction	MPa	12100
Young' modulus in Y direction	MPa	8600
Young' modulus in Z direction	MPa	8600
Poission's Ratio XY	-	0.27
Poission's Ratio YZ	-	0.4
Poission's Ratio XZ	-	027
Shear modulus XY	MPa	4700
Shear modulus YZ	MPa	3100
Shear modulus XZ	MPa	4700

5. Finite element development of composite pipe

The composite pipe was developed in ANSYS Workbench 19 software tool using HP workstation, the processor of Intel Core i7, 16GB RAM, 1TB ROM and windows 8.1Pro. Modeling of the composite was done in ACP composite modeling module and its analyses were done in a structural module.

5.1 Model development

Commercial software ANSYS is used to build a mechanical model of the composite pipe. The geometry of the pipe is shown in table 2 [8].

Table 2: Dimensions of the pipe

Length of the pipe	400 mm
Internal diameter of the pipe	100 mm
Thickness of layers	0.4 mm

Fig 1 shows the fibers orientations used in the numerical modeling which is exemplified by the arrows. These directions are ±60, ±45, ±55, ±65 which are along the longitudinal axis of the shaft. $\pm 60_4$, $\pm 60_6$, $\pm 60_8$these models were used to study the effect of increasing the number of layups and$\pm 45_4$, $\pm 55_4$, $\pm 65_4$ were used to study the effect of winding angle when subjected to external pressure.

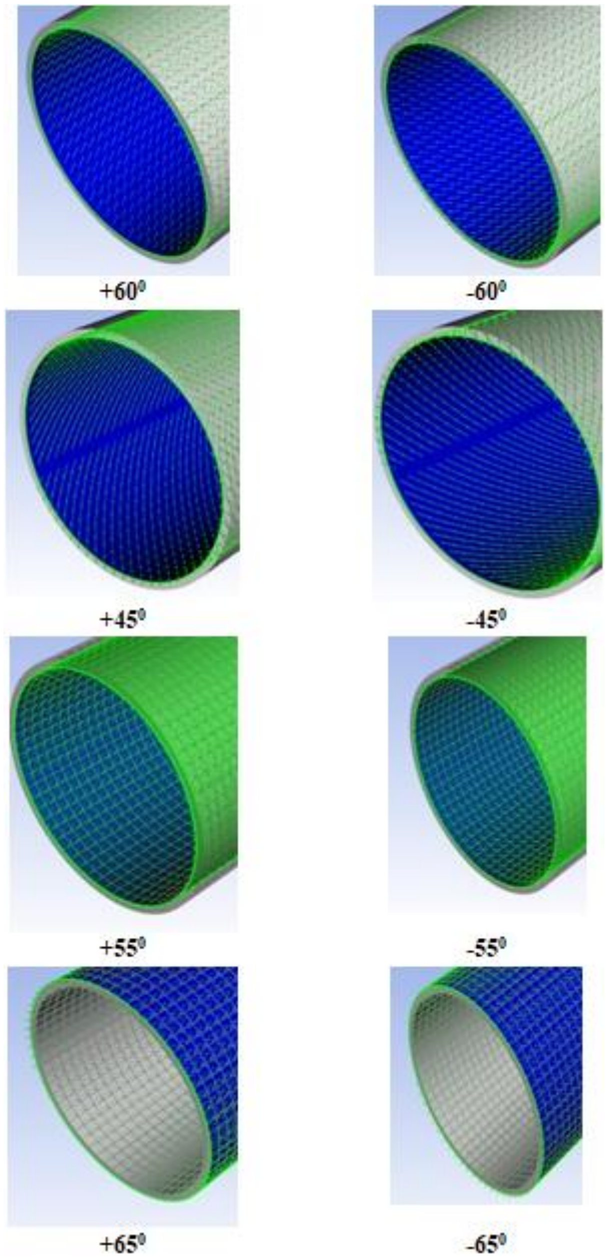

Figure 1: Depicting orientations of fibers used in the numerical modeling

In the meshed model, the quadrilateral element was used over the whole domain as shown in Fig.2. Meshing was done under user-controlled mesh in which 'extra fine' element size given to pipe. The complete mesh consists of 40320 elements and number of nodes 45927.

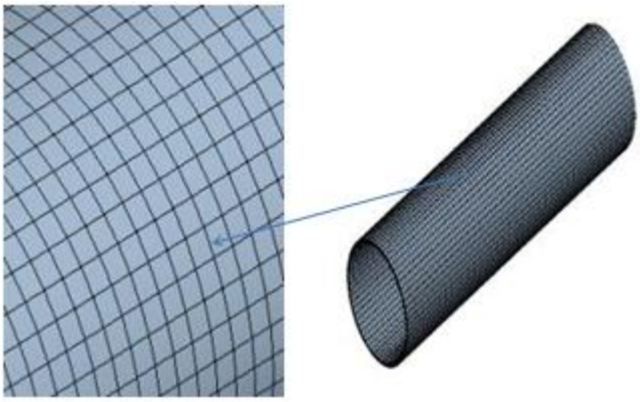

Figure 2: Meshed model of composite pipe

Stacking sequence of the models $\pm 60_4$and $\pm 45_4$are shown in Fig. 3 and all other models are modeled in the same manner by changing angle and number of layers. Regarding boundary conditions the pipe was fixed from both the ends and the pipe was subjected to external pressure.

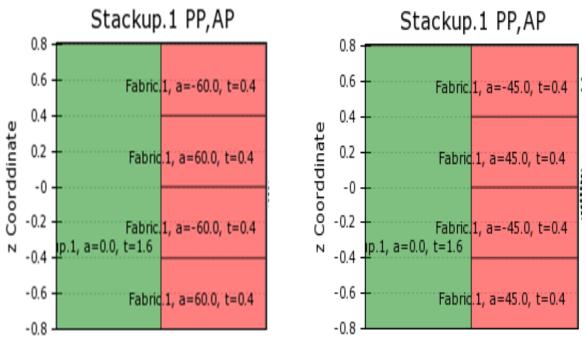

Figure 3: Stacking sequence of the model

5.2 Numerical Analysis

In this analysis of the last ply failure, if the last ply which is the outer ply got failed the whole structure is considered to be failed. IRF is determined by using the composite failure tool through the Tsai Wu theory. Randomly the pressure is applied to the model and its IRF is found out. IRF is the result type is an inverse margin to the safety factor. The failure load can be defined as the load value is divided by IRF. Failure is experienced when we have an IRF value greater than 1. If the value of IRF is less than 1 the applied load is to be increased to cause failure. Fig.4 represents the methodology used to determine the pressure at which the failure occurs.

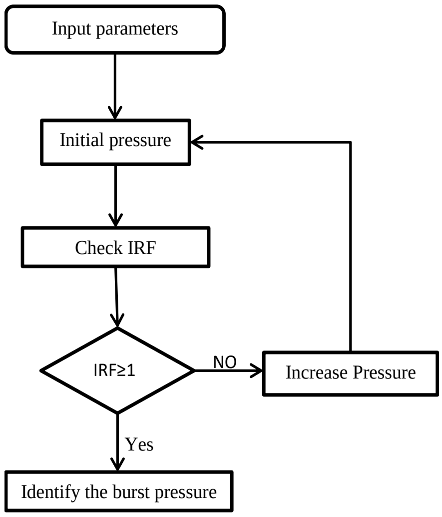

Figure 4: Flow chart of the methodology used to define the failure pressure

6. Numerical Results

6.1 Model verification

The model [90/$\pm 55_{12}$/90] of Humberto et al. [10] in terms of pressure and hoop displacement graph was validated. The present model of $\pm 60_8$was subjected to the same loading conditions as the Humberto model. Pressure and hoop displacement graph is plotted as shown in figure 5.

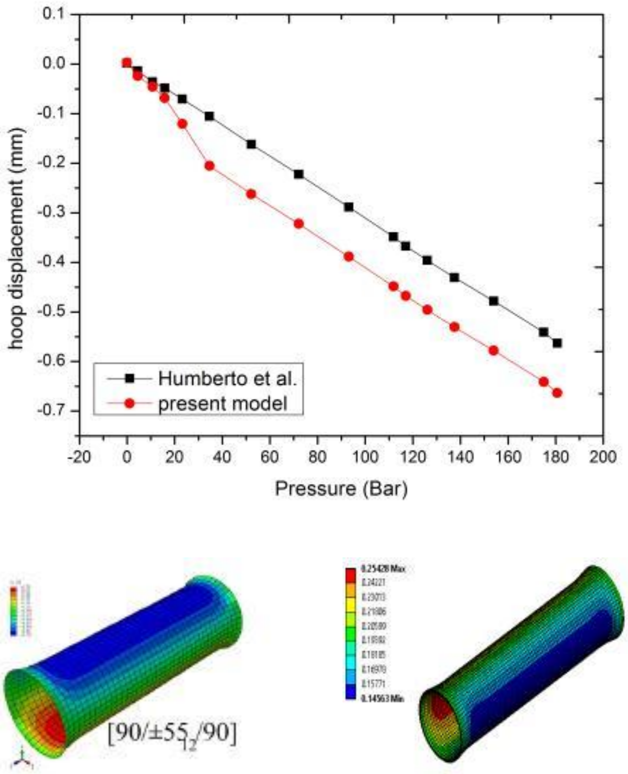

Figure 5: Pressure vs. deformation graph for the present model and Humberto model

From figure5 it is clear that the curve for both the models is approximately following the same nature. But there is more deformation in the present model because the number of layers is less in the present model also winding angle is $60^0$and the 55^0 is the optimum angle. The countours plot at

25 bar is shown above in which the minimum and maximum deformation of the pipe is at the same position.

6.2 Analysis based on increasing the number of layers

To study the effect of increasing the number of layers keeping the same winding angle three models were studied. The composite pipe was fixed from both the end and the pressure on the external surface is applied. Fig.6shows the pressure and deformation curve for the three models. The deformation varies linearly with the pressure for all the three models. After certain deformation taking place the pipe gets failed.

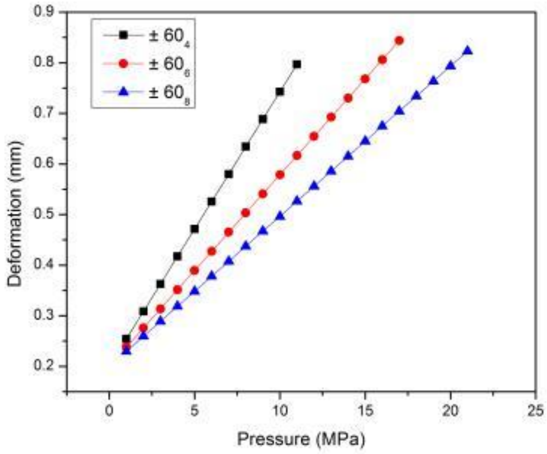

Figure 6: Depicting the pressure vs. deformation for the models with different number of layer

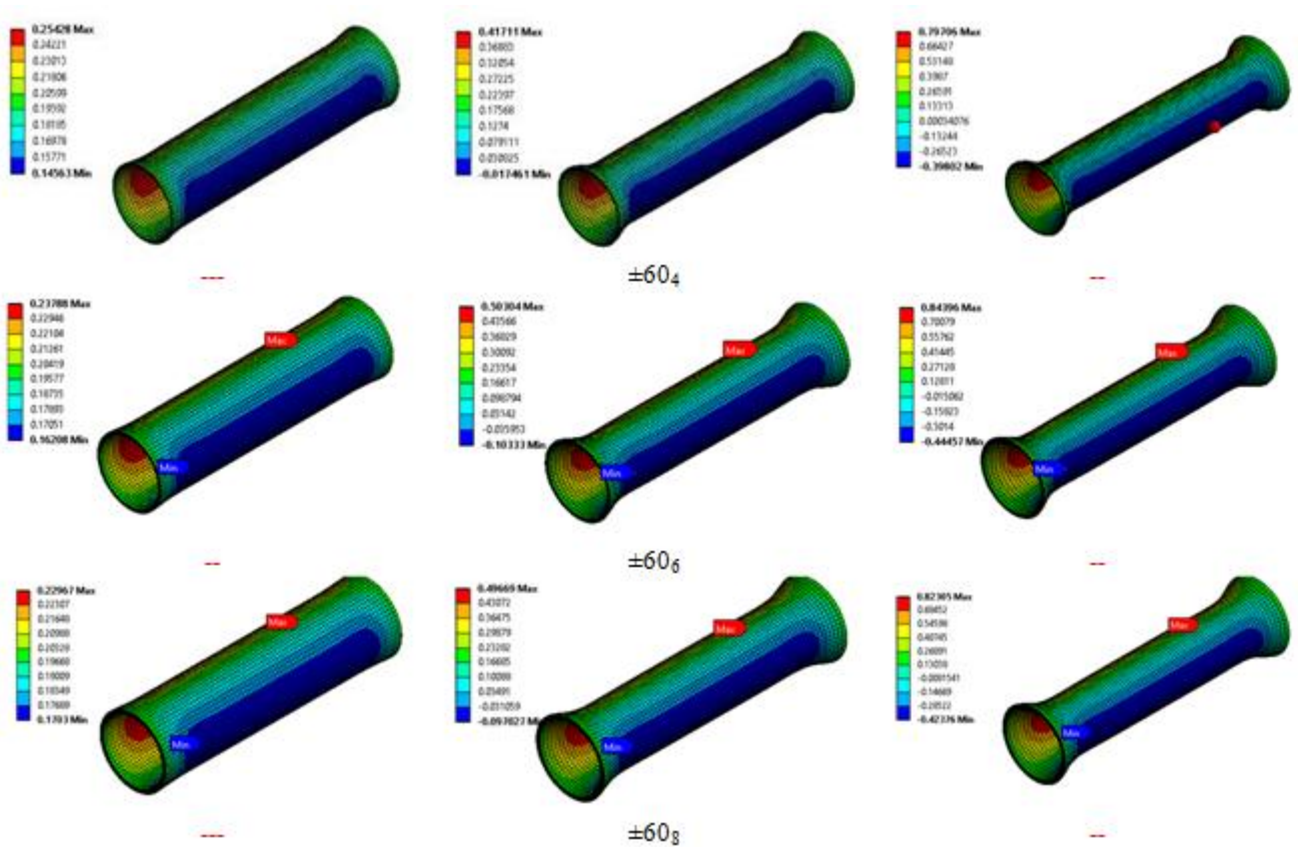

Figure 7: Contour plot of deformation at the initial intermediate and final level

From the contour plot Fig.7, it is clear that when we apply more number of layers the strength got increased but chances of delaminations increases as the d/t ratio increases [10]. With the increase in pressure, the buckling occurs at low pressure in the model having lesser layers as compared to the buckling in the more layers model.Fig.8 shows the stress-strain graph of the three models. The end point at which the failure take place indicates the point at which the value of IRF is equal to the or greater than 1. For the models $\pm60_4 \pm60_6$ $\pm60_8$the maximum failure pressure is 11 MPa, 17 MPa, 21MPa respectively.

The burst pressure for the maximum layer model is high but in composites we cannot increase the layers. As the layers or the thickness increases its chances of delaminations means the ply failure increases which is very disadvantageous for the filament wounded material. It could result in dangerous situations. So according to the application the stacking sequence, wind angle everything is taken into study. Fig8 shows the stress strain curve of the three models based on variations in the number of layers.

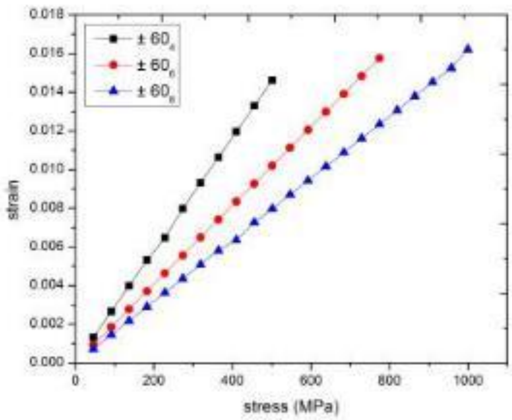

Figure 8: Depicting a stress-strain graph for the three model

The stress level for the model having more layups is showing that it can sustain more pressure. Stress and strain level is high for the model having 8 layups as compared to the other two models.

6.3 Analysis based on varying the angle keeping the same number of layers

Optimum winding angle for the filament wounded structure is 55^0[7]. In the present study, the study is based on keeping the number of layers same and varying angel three models are developed $\pm 45_4$, $\pm 55_4$, $\pm 65_4$. The burst pressure for the $\pm 55_4$model was maximum and compared to the other two models. According to the inverse reserve factor theory based on Tsai Wu composite failure theory the minimum pressure to cause the failure is calculated. The data is given in table 3.

Table 3: Burst pressure of the three models

Model	*Stress (MPa)*	*Burst Pressure (MPa)*
$\pm 45_4$	846.71	16
$\pm 55_4$	715.5	21
$\pm 65_4$	675.35	10

If we increase the winding angle the direction of fiber alignment for radial loading conditions, becomes close to loading directions that is close to $90^0$and tube strength got increased.

7. Conclusion

Based on the simulation of the laminated composite model with varying the winding angle and the number of layers, all the models were subjected to external pressure and failure analysis of the pipe was carried out on inverse reserve factor according to the Tsai Wu failure composite failure theory. For the models having different layups but same angle study shows that the deformation is less as the number of layups is high as compared to the models having less number of layers. Burst pressure was more in case of pipe having more number of layers. Study based on varying the winding angle by keeping the number of layers constant shows that the optimum angle of winding is 55^0. It can sustain more pressure prior to failure. As for the model $\pm 55_4$, the burst pressure was 21 MPa while the burst pressure for the other two models is less.

References

[1] J. Y. Han, H. Y. Jung, J. R. Cho, J.H. Choi, & W.B. Bae, (2008). Buckling analysis and test of composite shells under hydrostatic pressure. *Journal of materials processing technology*, *201*(1-3), 742-745.

[2] A. Zuraida, A.A. Khalid, &A.F.Ismail (2007). Performance of hybrid filament wound composite tubes subjected to quasi static indentation. *Materials & design*, *28*(1), 71-77.

[3] L.A.L. Martins, F.L. Bastian, & T.A. Netto, (2014). Reviewing some design issues for filament wound composite tubes. *Materials & Design*, *55*, 242-249.

[4] U. Wollina, M. Heide, W. Muller-Litz, D. Obenauf, & J. Ash (2003). Functional textiles in prevention of chronic wounds, wound healing and tissue engineering. *CurrProbl Dermatol*, *31*, 82-97.

[5] P. Krishnan, M.A. Majid, M. Afendi, A.G. Gibson, & H. F.A. Marzuki (2015). Effects of winding angle on the behaviour of glass/epoxy pipes under multiaxial cyclic loading. *Materials & Design*, *88*, 196-206.

[6] A. Tafreshi, (2006). Delamination buckling and postbuckling in composite cylindrical shells under combined axial compression and external pressure. *Composite structures*, *72*(4), 401-418.

[7] S. Grove (1999). Optimum fiber orientation in filament wound structures. *Journal of materials science letters*, *18*(15), 1203-1203. Grove, S. (1999). Optimum fiber orientation in filament wound structures. *Journal of materials science letters*, *18*(15), 1203-1203.

[8] Abdel-Ghany, Ahmed & Taha, Iman &Ebeid, Samy. (2016). Failure Prediction of Fiber Reinforced Polymer Pipes using FEA. International Journal of Engineering and Technical Research. 4. 115-120.

[9] A. Cherniaev& V. Komarov (2015). Multistep optimization of composite drive shaft subject to strength, buckling, vibration and manufacturing constraints. *Applied Composite Materials*, *22*(5), 475-487.

[10] J.H.S. Almeida Jr, M.L. Ribeiro, V. Tita, & S.C. Amico (2016). Damage and failure in carbon/epoxy filament wound composite tubes under external pressure: Experimental and numerical approaches. *Materials & Design*, *96*, 431-438.

Author Profile

Anju Verma received the B.Tech degrees in Mechanical Engineering from Ajay Kumar Garg Engineering College, Ghaziabad from UPTU University in 2014.