IOP Conference Series: Materials Science and Engineering

PAPER • OPEN ACCESS

Effect of low velocity impact on hybrid composite pipes orientated with different helix angles – numerical method

To cite this article: V Vijaya Rajan *et al* 2021 *IOP Conf. Ser.: Mater. Sci. Eng.* **1145** 012061

IOP Conf. Ser.: Mater. Sci. Eng. **1145** (2021) 012180 https://doi.org/10.1088/1757-899X/1145/1/012180

Retraction

Retraction: Effect of low velocity impact on hybrid composite pipes orientated with different helix angles – numerical method (*IOP Conf. Ser.: Mater. Sci. Eng.* **1145** 012061)

Published 23 February 2022

This article (and all articles in the proceedings volume relating to the same conference) has been retracted by IOP Publishing following an extensive investigation in line with the COPE guidelines. This investigation has uncovered evidence of systematic manipulation of the publication process and considerable citation manipulation.

IOP Publishing respectfully requests that readers consider all work within this volume potentially unreliable, as the volume has not been through a credible peer review process.

IOP Publishing regrets that our usual quality checks did not identify these issues before publication, and have since put additional measures in place to try to prevent these issues from reoccurring. IOP Publishing wishes to credit anonymous whistleblowers and the Problematic Paper Screener [1] for bringing some of the above issues to our attention, prompting us to investigate further.

[1] Cabanac G, Labbé C and Magazinov A 2021 arXiv:2107.06751v1

Retraction published: 23 February 2022

Effect of low velocity impact on hybrid composite pipes orientated with different helix angles – numerical method

V Vijaya Rajan[1], D Elancheziyan[1], P Amarnath[1], G L Arumparithy[1], M Shanmugam[2] and V Santhanam[2]
[1]Department of Mechanical engineering, Saveetha Engineering College, Chennai, India
[2]Department of Mechatronics Engineering, Rajalakshmi Engineering College, Chennai, India
vijayarajan@saveetha.ac.in

Abstract. In this study Finite Element Analysis (FEA) model is developed to understand the effect of low velocity impact on hybrid (Carbon/Glass/Epoxy) composite pipes which were wound using the filament winding technique in two different diameters. The results were compared with the existing experimental results and further validated FEA model was used to understand the effect of varying thickness and change of helix angle in the impact using ABAQUS tool. In this FEA design, composite pipe was modelled with the orientation of (CG90/CG60) for varying thicknesses (2.67mm, 4mm, 5.3mm and 6mm) and helix angles with variations as 15˚, 30˚, 45˚, 60˚, 75˚, and 90˚ respectively. Hybrid pipe model was subjected to three different levels of impact energy namely as 20 J, 25 J and 30 J. Effect of impact studied in terms of force, deflection and energy absorbed by the specimen. The possibility of increasing in the energy absorption through increasing thickness and withstanding high impact forces and impact resistance decreasing with increase in the helix angle of orientation namely, as 15˚, 30˚, 45˚, 60˚, 75˚, and 90˚ respectively was seen.

1. Introduction

The use of composite pipes in different fields like oil firms to transmit oil and in automobile industries etc., is seen. This is due to their superior properties like strength to weight ratio, high corrosion resistance property, good fatigue strength and good burst strength. In some cases they may subjected to impact of various kinds in a working area during maintenance and fitting operations. Composite pipes are usually wound using of the filament winding method with some orientation. The strength of these composite pipes depends upon various parameters like thickness, orientation of fibres, impact energy, mass of the plunger and materials etc.

[1] made experimental study of the impact behaviour of filament wound composite pipes and the results were validated using the numerical method. [2] studied axial crush behaviour in a conical frusta in low velocity. Conical frusta was made by E-Glass/epoxy over wrapped aluminium and the results were compared with these from the finite element analysis. [3] did a experimental and numerical study of the impact analysis on productive layer which was made by geofoam structures in terms of energy absorption for reducing the effect of impact on steel pipes and composite pipes. [4] developed a numerical model for GFRE pipes in ANSYS/LS-DYNA and studied the effect of impact in different energy levels for two different thicknesses and the FE results were correlated with experiment results.

[5] developed the FEA model and results were compared with the experimental and optimisation algorithms applied to optimise the material required. [6] investigated the progressive damage behaviour of hybrid composite plates using the impact test. It was validated by FEA. M.A. Badie, E. Mahdi, and A.M.S. [7] studied the effect of fibre orientation angle and stacking sequence in torsional stiffness in an automotive drive shaft which was made hybrid composite and studied the fatigue model by FEA. [8] took up the study of the impact effect on woven fabric laminate by experimental and by numerical analysis for the matrix strength and for the bonding properties between yarn and matrix. [9] studied the mechanical behaviours of carbon fibre composites using the tensile test and numerical analysis. [10] conducted multi axial cyclic load test on glass/epoxy composite tubes which were wound with different helix angles. Results showed the different winding angle giving a good performance in loading of different kinds. [11] work into the analysis of the mechanical properties of okra fibre using tensile test and results were correlated with the FEA, The properties of kenaf and banana fibre were found by FEA. S. [12] developed a numerical model in ANSYS and studied the effect of the impact and flexural behaviour on glass fibre reinforced composites. [13] did a study of the effect of impact analysis on ultra-lightweight cement composite pipes using experimental and by numerical methods which was developed in LS-DYNA. The results predict were close to pipe-in-pipe composite specimens. [14] developed a quasi-static model to understand the effect of interface angle, play orientation with respect to fixed angle and ply grouping in impact analysis. Results showed all factors having a notable effect on impact and damage propagation. [15] were prepared filament winded hybrid composite pipes with various diameter and low velocity impact test were conducted at various energy level, results shows that increasing thickness could increase the energy absorption rate of the composites. [16] did a study of the effect of stacking sequence on flexural properties of glass/carbon hybrid composites using experimental and by numerical methods which was done by Ansys ACP. The results shows that hybrid composites gives a better flexural strength than the individual composites. [17] were developed a new auto-generated geometry-based discrete model (AGDM) to study the progressive failure and damage evaluation in composites.

In this work FEA model was developed to understand the effect of low velocity impact on hybrid composite pipes at different energy levels on two specimens of different thickness and the results were correlated to the experimental work. Different helix angled FEA models were used to quantify the effect of low velocity impact on carbon/glass/epoxy hybrid composite pipes.

2. Finite Element Analysis – Model development

FEA model of Hybrid composite (Carbon/Glass/Epoxy) pipe was modelled using ABAQUS tool. ABAQUS 6.10 can model components using various input methods. The model could also be imported. There was a need to define the components as constituting a deformable body or a discrete rigid body or analytically rigid body. There was no need to apply material properties of the components were defined as rigid one. The Composite pipe was modelled as a 3D deformable shell part with 100 mm length and 50 mm internal diameter w ith interwoven (CG90/CG60) orientation. The thickness values of the specimen assigned in the section part of the pipe module was 4 mm and 6 mm. A plunger was modelled as a 3D discrete rigid shell part with 12.7 mm diameter.

2.1 Material Property, Meshing, and Boundary conditions

In ABAQUS 6.10 the properties of the materials can be defined in property module, select the material as homogeneous or the composite material. In this modelling properties of interwoven composite material model were given, which is given as orthotropic lamina. The properties of hybrid composite pipes were determined using the tensile test according to ASTM standard. Figure 1 shows the details of the tensile test conducted for the interwoven carbon/glass specimen. Calculations are given below,

Axial Young's Modulus, $E1 = \sigma$ axial / $\varepsilon 1 = 30193.47$ MPa

Lateral Young's Modulus, $E2 = \sigma$ lateral / $\varepsilon 2 = 7417.5$ MPa

Axial Poisson's ratio $\gamma 12 = \epsilon 2 / \epsilon 1 = 0.17$

Lateral Poisson's ratio, $\gamma 21 = E2\gamma 12/ E1 = 0.03$
Rigidity modulus, G12 = = 7444.67 MPa
Rigidity modulus, G23 = = 3600.72 Mpa

Figure 1. Tensile Test with strain meter

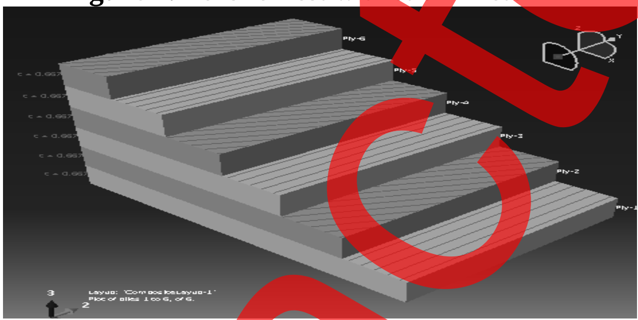

Figure 2. Composite layup model with orientation

Discrete rigid plunger part has no property, but mass value was added to the reference point of the plunger. 6 layer Composite layup was created and its respective orientation values were material properties were assigned to it. Figure 2 shows the composite pipe layup model with the relative material orientation thickness and co-ordinate system for each layer. Similarly for 6 layers with different orientations composite layup model was created with their respective properties. Global co-ordinate system was used for the purpose of orientation. Figure 3 shows the Modelling of composite pipe.

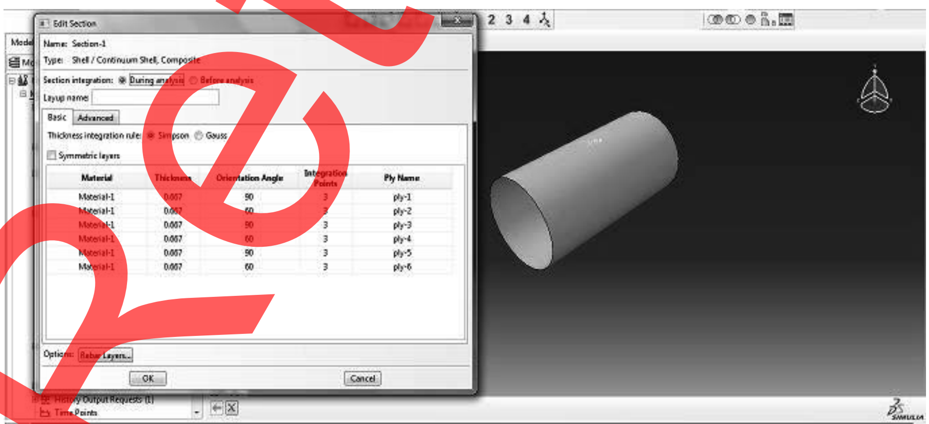

Figure 3. Modelling of Composite Pipe

The composite pipe model, plunger models were assembled by instances with dependent mesh condition. The height of the plunger was adjusted using the translational movement. Two individual surfaces were created for each part. Figure 4 shows the assembled model of the composite and the plunger.

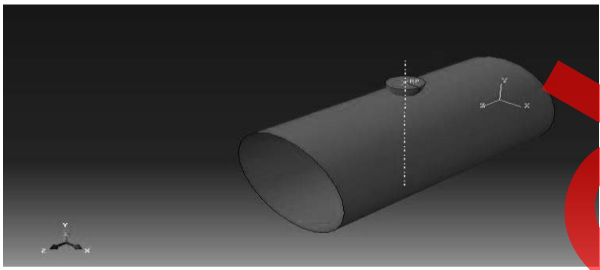

Figure 4. Assembled model of composite pipe and plunger

In ABAQUS meshing could be done by various type of elements and sizes of element feeding. Accuracy in the results of the jobs depends upon the both element type and size. Composite pipe was meshed using an S4R element. S4R is a 4 noded shell element which has 6 degrees of freedom per node. The plunger was meshed using a R3D4 element. R3D4 is a 4 node 3D bilinear rigid quadrilateral element. Figure 5 shows the meshed model.

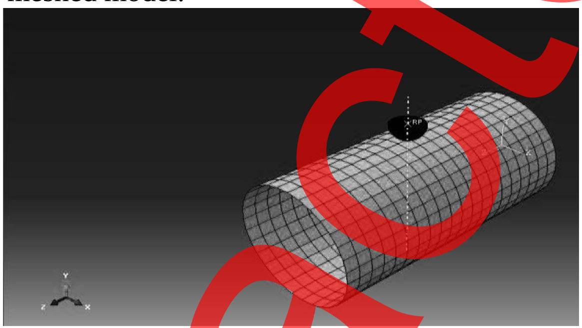

Figure 5. Meshed model

A dynamic/Explicit solver was selected with time intervals for doing the impact analysis. Mechanical tangential and normal hard contact interaction properties were used in the study. Interaction between composite pipes and plunger was done using surface-to-surface contact with the interaction properties. The Plunger surface acted as master surface and composite surface acted as a slave surface for the interaction. Interaction properties was used for defining the effect of the contact at the time of the impact between the master surface and the slave surface. Contact constraints were applied for the both the master and the slave surface. Figure 6 shows the steps and interaction property.

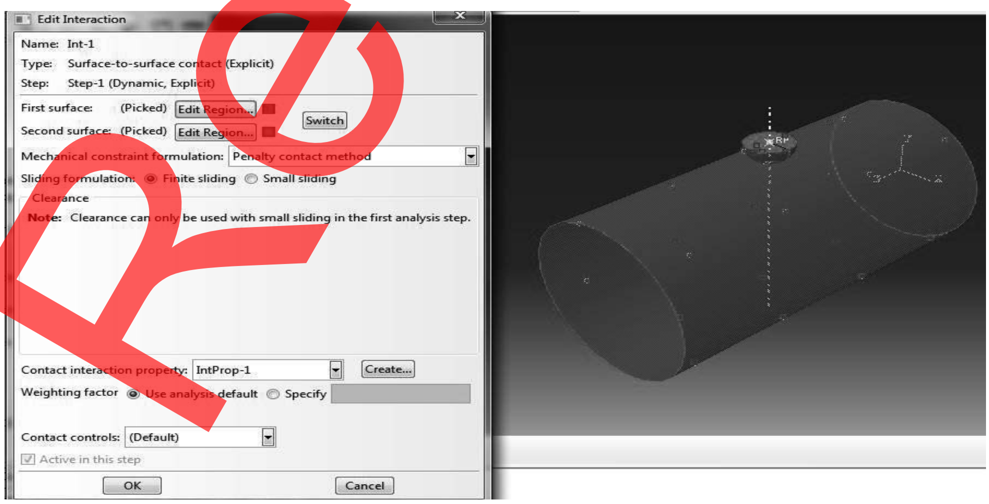

Figure 6. Steps and Interaction Property

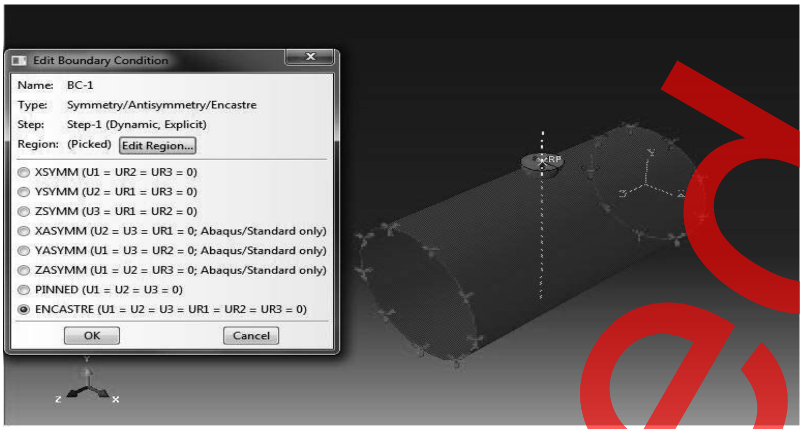

Figure 7. Boundary Conditions

Figure 7 shows the boundary conditions (BC) applied on the 100 mm composite pipe and the plunger. All degrees of freedom of the composite pipe were arrested to provide a fully constrained condition. This was applied on both the edges of composite pipe. Only a translational in y-direction alone was allowed for the plunger and all other displacement values were arrested. This was applied on the reference point of the plunger. Details of the BC's given below,

- Boundary condition for pipe – U1=U2=U3=UR1=UR2=UR3= 0
- Boundary condition 2 - U1=U3=UR1=UR2=UR3= 0
- Predefined field – initial velocity of plunger was applied in this region. This was applied on the reference point of the plunger in a negative direction.

3. Results and validation of FEA model

The developed FEA model was correlated to the existing experimental work [15]. Energy absorption of specimen obtained from FEA model was used for comparison of the results. Figure 8 shows results obtained from experimental and from numerical model for 4 mm thickness specimen energy absorption value at 30 J for the purpose of validation.

This correlation validates the results of experimental [15] and numerical models at 30 J of the impact energy for the specimen with 4 mm thickness. The results for peek force and deformation were also correlated to the numerical model. The error percentage was seen as less than 7% of the overall results. The results of numerical and experimental models are given below,

Table 1. Comparison of Impact test results – FEA & Experimental

Orientation	Impact energy	Peak Force (N)		% of Error	Peak Energy (J)		% of Error
		FEA	Experimental		FEA	Experimental	
$(CG90/CG60)_3$	20J	4023.1	3781	6.01	19.58	18.24	6.84
	25J	4238.7	4062	4.17	24.48	24.05	1.76
	30J	4416.4	4358	1.32	29.36	28.93	1.46

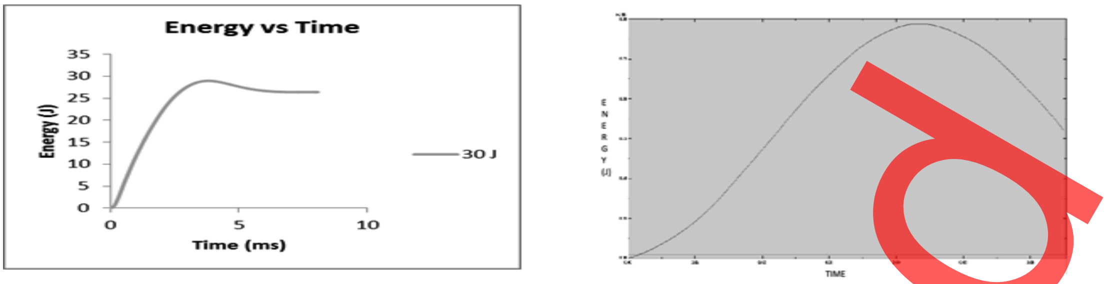

Figure 8. Experimental Result and FEA Result for 4 mm thickness specimen at 30 J Impact energy

3.1 Results and discussion

The validated FEA model used for quantifying the effect of low velocity impact on the varying helix angled modelled, for each orientation, maximum conduct force, displacement and maximum energy values are shown in Table 2. The maximum conduct force of specimen was seen decreasing with increase in the helix angle. The specimen with 15˚ helix angle has good load carrying capacity and good impact resistance capacity and displacement values were seen increasing with increase in the helix angle, showing the stiffness of the specimen decreasing with increasing helix angle.

Figure 9 shows the variations in force with respect to time at 30 J impact energy level for varying orientations. Decrease in the force values was seen with increasing helix angles. The maximum values reached 4485 N for 15˚ helix angle specimen but, when the helix angles were increased, the force values came to 4264 N for 90˚ angle specimen. Figure 10 shows the Peak Force Values for varying orientations at 30 J.

Table 2. Impact Test Results for varying orientations

Orientation	Stacking Sequence	Applied Impact energy	Peak Force (N)	Displacement (mm)	Peak Energy (J)
1	$(CG90/CG15)_3$	20J	4047.98	7.84	19.69
		25J	4333.68	8.84	24.62
		30J	4485.06	9.74	29.48
2	$(CG90/CG30)_3$	20J	4002.21	7.96	19.68
		25J	4277.55	8.97	24.60
		30J	4460.04	9.89	29.50
3	$(CG90/CG45)_3$	20J	4037.69	8.20	19.66
		25J	4267.10	9.24	24.57
		30J	4433.62	10.16	29.46
4	$(CG90/CG60)_3$	20J	4023.18	8.48	19.58
		25J	4238.77	9.54	24.48
		30J	4416.49	10.46	29.36

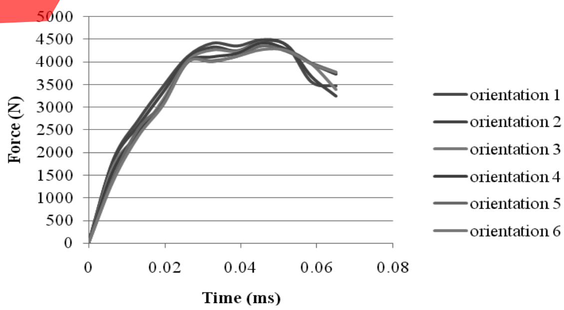

Figure 9. Time versus Force graph for varying orientations at 30 J impact energy level

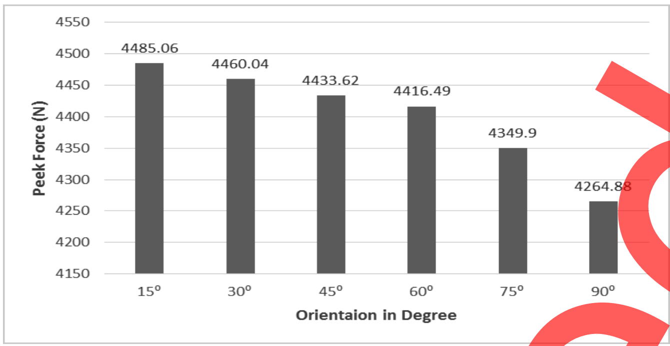

Figure 10. Peak Force Values for varying orientations at 30 J

Figure 11 shows the stress distribution pattern of the specimen for the impact on a 4 mm thickness specimen. It shows the occurrence of maximum contact force at the area of contact between impactor and specimen and the stress pattern distributed along the longitudinal direction.The results of the impact for varying thicknesses with orientation (CG90/CG60)3 are shown in Table 3. Internal diameter of the specimen fixed as 50 mm. Table 3 shows the results of the impact test for specimens with varying thicknesses namely, as 2.67 mm, 4 mm, 5.3 mm and 6.67 mm. Increase in the stiffness of the specimen from increase in the number of layer was seen, also reduction in the displacement and maximum energy values following an increase in the conduct of force values. There was an increase in the energy absorption of the material with increase in the number of layer.

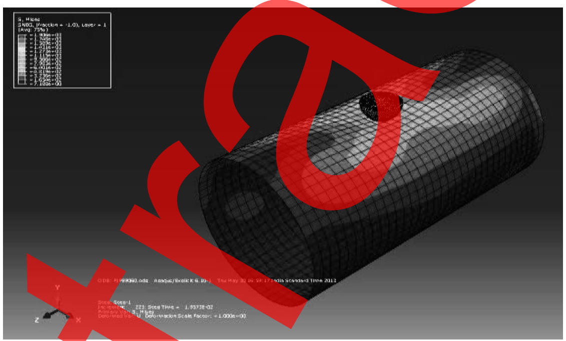

Figure 11. Stress Distributions at the time of impact

Table 3. Impact Test Results for Varying Thickness Specimens

Number of layers	Thickness (mm)	Applied Impact energy (J)	Peak Force (N)	Displacement (mm)	Peak Energy (J)
4	2.67	20	2669.47	13.05	19.84
		25	2676.24	14.36	24.82
		30	2751.27	15.45	29.74
6	4	20	4023.18	8.48	19.58
		25	4238.77	9.537	24.48
		30	4416.49	10.46	29.36
8	5.3	20	6108.56	5.95	19.31
		25	6603.75	6.71	24.13
		30	7105.81	7.39	28.9
10	6.67	20	6801.89	4.60	19.1
		25	7053.76	5.17	23.92
		30	7502.75	5.68	28.69

In Time versus Force graphs, force values are seen increasing linearly for 2.67, 4, 5.3 mm thickness specimens but, for specimen with 6.67 mm thickness, the force increased to 7502 N. Experimental investigation showed the occurrence of a global failure at the impact area due to increase in the specimen force to 7502 N for 6.67 mm thickness. Experimental investigation showed the occurrence of the failure of 5.3 mm thickness specimens at 30 J impact. 2.67 mm thickness specimen got an impact by 30 J, the a complete failure of specimen occurred but failure in 4, 5.3, and 6.67 mm thickness specimens was localizeded when compared to specimen with 2.67 mm thickness. Figure 12, 13 and 14 represents the graphs.

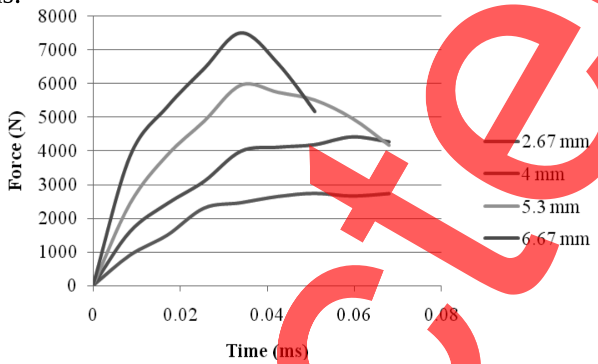

Figure 12. Time versus Force graph for varying thicknesses

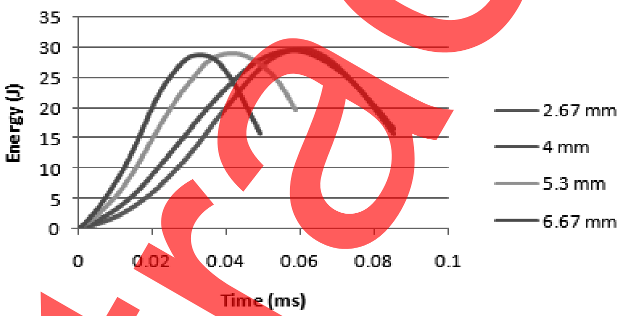

Figure 13. Time versus Energy graph for varying thicknesses

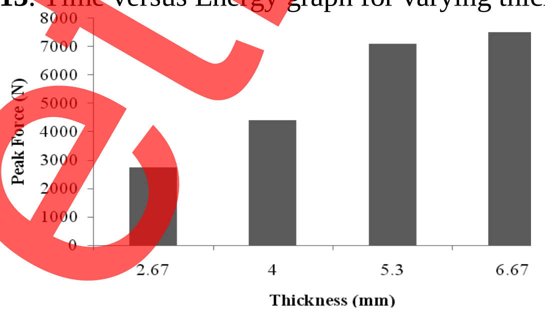

Figure 14. Peak Force values for varying thicknesses at 30 J

4.Conclusion

Low velocity impact analysis was done using the Finite element analysis method for varying orientation specimens and for varying thickness specimens with varying impact energy levels. The properties of interwoven material were found using the tensile test along with the strain gauges. The results were correlated to the experimental work and another model was used for quantifying the effect of thickness and orientation on impact. These results lead to the following conclusions,

- An increasing the helix angle reduces the impact resistance characteristics of the material and the stiffness of the material. Orientation with the helix 15˚ has a maximum peak force of 4485 N and reaches the maximum peak energy 29.48 J at 30 J impact energy level compare with orientations 30˚, 45˚, 60˚, 75˚ and 90˚.

- When helix angle increases from 15˚ to 30˚, 60˚, and 90˚ peak force gets reduced to 0.5%, 1.5% and 4.9 % respectively.
- Increase in thickness can increase the impact resistance characteristics of the material and increase the stiffness of the material. The 6.67 mm thickness specimen reaches the maximum force to 7502 N and peak energy 28.69 J at 30 J impact energy level.

References

[1] Qiaoguo Wu, Xuedong Chen, Zhichao Fan, Yong Jiang, Defu Nie 2017 ,Experimental and numerical studies of impact on filament-wound composite cylinder,*Acta Mech. Solida Sin.* **30.5**, pp 540-549.

[2] M. Kathiresan, K. Manisekar 2016 ,Axial crush behaviours and energy absorption characteristics of aluminium and E-glass/epoxy over-wrapped aluminium conical frusta under low velocity impact loading,*Compos. Struct* **136**, pp 86–100

[3] Ozgür Anil, R. Tu_grul Erdem, Erkan Kantar 2015 ,Improving the impact behavior of pipes using geofoam layer for protection,*Int. J. Press. Vessel. Pip.* **132**, pp 52-64

[4] Rubaian F. Alshahrani, Nesar Merah, Shafique M.A. Khan, Yagoub Al-Nassar 2016 ,On impact-induced damage in glass fiber reinforced epoxy pipes ,*Int. J. Impact Eng.* **97**,pp 57–65

[5] M. Yong, L. Iannucci, B.G. Falzon 2010 ,Efficient modelling and optimisation of hybrid multilayered plates subject to ballistic impact,*Int. J. Impact Eng* **37**, pp 605–624

[6] Ercan Sevkat, Benjamin Liaw, Feridun Delale, Basavaraju B. Raju 2009 ,Drop-weight impact of plain-woven hybrid glass–graphite/toughened epoxy composites,*Compos. Part A* **40**, pp 1090–1110

[7] A. Haldorai and A. Ramu, Security and channel noise management in cognitive radio networks, Computers & Electrical Engineering, vol. **87**, p. 106784, Oct. 2020. doi:10.1016/j.compeleceng.2020.106784

[8] A. Haldorai and A. Ramu, Canonical Correlation Analysis Based Hyper Basis Feedforward Neural Network Classification for Urban Sustainability, Neural Processing Letters, Aug. 2020. doi:10.1007/s11063-020-10327-3

[9] Afroz Mehar G.M.Sayeed Ahmed G.Prassana Kumar, Md. Aaqib Rahman M.A.Qayum 2015 ,Experimental Investigation and FE Analysis of CFRP Composites,*Mater.Today: Proceedings* **2(4-5)** , pp 2831 – 2839

[10] Pranesh Krishnan, M.S. AbdulMajid, 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 *Mater. Des* **88**, pp 196–206

[11] Rakesh.Potluri, James Paul .K, Sd. Abdul kalam, P.Prasanthi 2017, Mechanical Properties Characterization of Okra Fiber Based Green Composites& Hybrid Laminates, *Mater.Today:: Proceedings* **4**, pp 2893–2902

[12] S. Irfan Sadaq, V.Suvarana Kumar, G. M. Sayeed Ahmed, Md. Irfan 2015 ,Experimental Investigation and Impact Analysis of GFRP Composite Laminates,*Mater. Today: Proceedings* **2(4-5)** , pp 2808 – 2816

[13] Yu Wang, Xudong Qian, J.Y. Richard Liew, Min-Hong Zhang 2014 ,Experimental behavior of cement filled pipe-in-pipe composite structures under transverse impact,*Int. J. Impact Eng* **72**, pp 1-16

[14] Edgar Fuoss, Paul V. Straznickya-f, Cheung Peon 1998 ,Effects of stacking sequence on the impact resistance laminates - Part 1: parametric study,*Compos. Struct* **41(1)** , pp 67-77

[15] V.Vijaya Rajan, R.Muruganandhan 2019 ,Impact analysis of carbon/glass/epoxy hybrid composite pipes,*IJRTE* **8**, pp 6002-6003

[16] Chensong Dong 2020 ,Flexural properties of symmetric carbon and glass fibre reinforced hybrid composite laminates,*Compos. Part C* **3** , p 100047

[17] Jiakun Liu, Stuart Leigh Phoenix 2020 ,An auto-generated geometry-based discrete finite element model for damage evolution in composite laminates with arbitrary stacking sequence,*Compos. Part C* **3**, p 100066