International Journal of Mechanical Engineering and Technology (IJMET)
Volume 8, Issue 5, May 2017, pp. 75–88 Article ID: IJMET_08_05_009
Available online at http:// http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=8&IType=5
ISSN Print: 0976-6340 and ISSN Online: 0976-6359

 Scopus Indexed

NUMERICAL ANALYSIS OF HYBRID CARBON FIBER COMPOSITE SPECIMEN AND VALIDATION OF RESULTS

Nithin Prasad
M. Tech Machine Design Student, Department of Mechanical Engineering,
Sree Narayana Institute of Technology, Adoor, Kerala, India

Adarsh Ramachandran
Assistant Professor, Department of Mechanical Engineering,
Sree Narayana Institute of Technology, Adoor, Kerala, India

ABSTRACT

This paper focused on numerical study of hybrid composite specimens, which was experimentally done by preparing according to ASTM standards. Hybrid Composites are the formed by the combination of more than one reinforcement and matrix, which exibits very good properties compared to that of single reinforcement composites. These materials shows better flexibility when comparing with other composites. Hybrids consists of two types of fibers - a high modulus fiber and a low modulus fiber. The high-modulus fiber provides better stiffness and high load bearing capacity, whereas the low-modulus fiber provide more damage tolerance and reduces the material cost. The mechanical properties of a hybrid composite can be altered by varying volume ratio and stacking sequence of different plies.

The numerical study was conducted by developing specimen models in Solid Edge ST4 and later importing to ANSYS Workbench 15.0. The boundary conditions were applied and analysis results compared with that of experimental results to suggest its further applications.

Key words: Hybrid, Reinforcement, Matrix, High Modulus Fiber, Low Modulus Fiber.

Cite this Article: Nithin Prasad and Adarsh Ramachandran, Numerical Analysis of Hybrid Carbon Fiber Composite Specimen and Validation of Results, *International Journal of Mechanical Engineering and Technology*, 8(5), 2017, pp. 75-88.
http:// http://www.iaeme.com/ijmet/issues.asp?JType=IJMET&VType=8&IType=5

1. INTRODUCTION

Hybrid composites are more advanced composites forms as compared to that of conventional plain laminate composites. Hybrids can have several alternatives including more than one reinforcing phase and a single matrix phase, single reinforcing phase with multiple matrix phases and multiple reinforcing with multiple matrix phases. Based on the possible interactions connecting the inorganic and organic species, hybrid materials can be classified into two types – Class I and Class II hybrids. Class I hybrid materials exibits weak interactions between the two phases whereas Class II hybrid materials exibits strong chemical interactions between the components.

The hybrid composites were prepared by means of hand lay-up process, which is the easiest way in making a composite specimen. The reinforcements were selected on the basis of high modulus of elasticity, density as well as strength. Carbon fiber has the high modulus of elasticity, strength and low density compared to other selected materials and it was preferred as the base material. Other materials were arranged layer by layer over base material and matrix was applied to make a perfect bonding between them. Loading of 75 kgs was provided for 1-2 days and obtained material was cut into ASTM standard dimensions by means of a cutter.

In order to replace the existing stainless steel used in impact energy absorbers of automobiles and to minimize cost of manufacturing of composites, hybrids are prepared and experimental analysis was conducted. The analysis shows it has better properties than composites which are available in present day market and can be used in high loading conditions. In the present work an attempt has been made to validate those experimental results and to find out how much energy can be absorbed by the hybrid compared to conventional materials used in energy absorbers of automobiles.

2. INTRODUCTION TO ANSYS

Inorder to simulate all disciplines of physics, structural, fluid dynamics, heat transfer and electromagnetic interactions, ANSYS software is mostly used. It consists of pre-processor, solver and post-processor softwares for numerical analysis purpose. The pre-processor software engine is used to model geometry. The solver provides solutions corresponds to boundary conditions applied to the meshed geometry. Finally, pre-processor provides desired output. FEA is used mostly in mechanical and civil engineering designs. The FEM is a very useful tool for studying the behaviour of structures. In this method, the finite element model is created by dividing it into a number of finite elements, where each element is interconnected by nodes. Depending upon the behaviour and geometry of the structure being analyzed, the selection of elements for modelling the structure is done. The modelling pattern (mesh) for the finite element method, has a great role in the modelling process. The accuracy of ANSYS results depends upon the mesh size and no of elements considered. Even though the FE model does not exactly behave like the actual structure, sufficiently accurate results will be obtained for most practical applications.

The goal of meshing is to provide robust and easy to use meshing tools will simplify the mesh generation process. These tools will provide moderate to high degree of user control and have the benefit of being highly automated.

Advantages of FEA

- Visualization increases
- Design cycle time reduces
- No. of prototypes reduces
- Testing reduces
- Optimum design

The process of performing ANSYS can be broken down into three main steps.

1. Pre-processing
2. Solver
3. Post-processing

Pre-processing

This step is most important one in numerical analysis. Any modelling software can be used for geometric modelling and the model can be shifted to other simulation softwares for analysis purpose. Later mesh generation (It is the process of subdividing a region to be modelled into a set of small elements) is employed. In general, geometric arrangement of elements and nodes in a finite element model is defined by a mesh network. Nodes will represent points where parameters such as displacements are calculated. Elements are bounded and localized mass and stiffness properties of the model are defined by means of nodes. Also, elements can be defined by mesh number, by which references are to be made corresponding to deflections, stresses at specific model location. The common mesh elements used in ANSYS solver is triangular, tetrahedral and brick.

Solver

User has to work hard, while computer acts as the solver to do the job during pre-processing. User has to click on solve icon & wait for the results. Internal software carries out all the calculations including matrix formations, inversion, and multiplication & provide solutions for the required parameters.

Eg: Displacement & then find strain & stress for static analysis.

Post-processing

This is the final step in which the ANSYS results are analyzed. However, the real ANSYS simulation values are frequently found to provide accurate predictions of quantities such as find displacement and stresses. Post processing is for viewing results, verifying the results, to arrive at conclusions to think about what steps could be taken to further improvement of the design.

Assumptions

- Software to be used for ANSYS 15.0
- Model is simplified for FEA.
- Meshing size is limited to computer compatibilities for better processing.
- Static analysis is considered.

- Material used for analysis is orthotropic.

3. NUMERICAL ANALYSIS

3.1. Static Analysis

The static analysis is the primary step employed for the numerical study of the hybrid composites. The steps followed as follows.

1. Create a solid model of specimen in Solid Edge ST4. Dimensions are same as that of experimental test specimen. Save the model in IGES format.
2. Add hybrid materials in ANSYS 15.0 Workbench under new category by using experimentally obtained values.
3. Open static structural module for static analysis in ANSYS 15.0 Workbench and import the created 3D model into it.
4. Select the required hybrid materials for analysis purpose and assign each one at a time.
5. Create a mesh convergence study of specimen and select suitable mesh for further analysis.
6. Apply boundary conditions as that of experimental approach.
7. Run the static analysis by providing required parameters.
8. Obtain the analysis results.

Meshing

Meshing is the process in which the geometry is divided into elements and nodes. Along with material properties, the meshes are used to represent the stiffness and mass distribution of the structure mathematically. The mesh has been generated automatically. The default element size is determined based on several factors including the overall model size, other topology proximity, curvature of body, and the complexity of the model.

- Type of meshing: 3D
- Type of elements: Automatic

Boundary Conditions

For the static analysis of the specimen, certain boundary conditions need to be mentioned. Boundary conditions include which parts needs to be fixed; how much load should be provided, torque, speed, etc. The considered boundary conditions considered here were as follows:

Tensile Load Test

- One end of the specimen model was fixed.
- Maximum load obtained from experimental analysis was provided at other end.

Flexure Load Test

- Model was fixed by means of two - pin arrangement.
- Maximum load was provided on top surface by means of another pin.

Considered Load Conditions

- Here hybrids were preferred for numerical analysis.
- Maximum loads obtained during tensile and flexure load tests were considered for analysis.
- Static analysis was done by mesh convergence study.

Table 3.1.1 Applied Load Conditions

TENSILE LOAD TEST	
Carbon – Kenaf Hybrid	Carbon – E Glass Hybrid
10060.587 N	9536.978 N
FLEXURE LOAD TEST	
Carbon – Kenaf Hybrid	Carbon – E Glass Hybrid
555.15 N	339.554 N

For static analysis of specimen using hybrid composites, the mechanical properties of materials should be introduced to ANSYS. The properties are obtained from experimental analysis of the specimen according to ASTM standards. These properties should be added to ANSYS under the name of new material. Now this material can be applied to any model for further numerical analysis purposes.

Table 3.1.2 Considered Mechanical Properties

PROPERTIES		**HYBRID COMPOSITES**	
		CARBON - KENAF	**CARBON – E GLASS**
Density (kg/m^3)		1650	1470
Young's Modulus (MPa)	E_x	214191.2	25882.41
	E_y	3421.65	4116.33
	E_z	3421.65	4116.33
Poisson's ratio	μ_{xy}	0.267	0.273
	μ_{yz}	0.319	0.355
	μ_{xz}	0.319	0.355
Shear Modulus (MPa)	G_{xy}	84526.91	10165.91
	G_{yz}	12970.6	1518.94
	G_{xz}	12970.6	1553.33

3.2. Analysis Test results (Tensile and Flexure Load Tests)

3.2.1. Tensile Load Test Results

Carbon - Kenaf Hybrid Composite

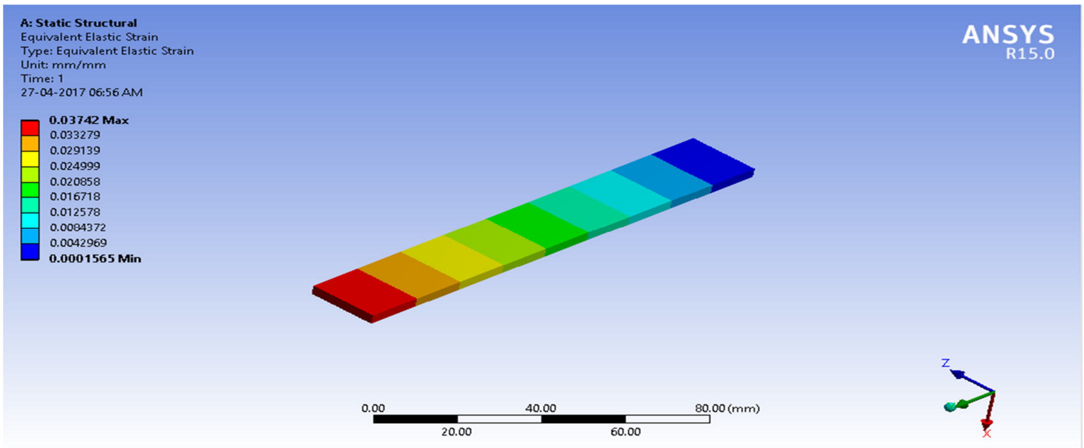

Figure 1 shows Equivalent Strain

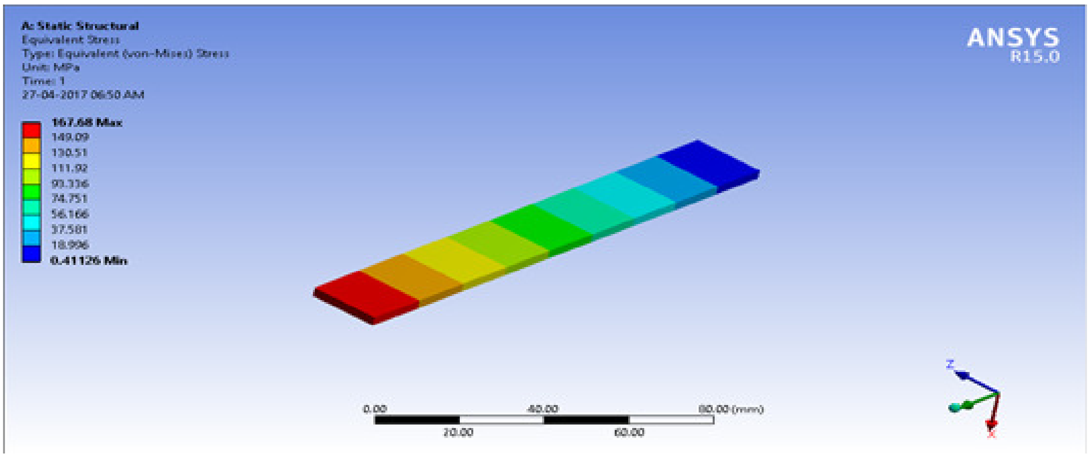

Figure 2 shows Von-Mises Stress

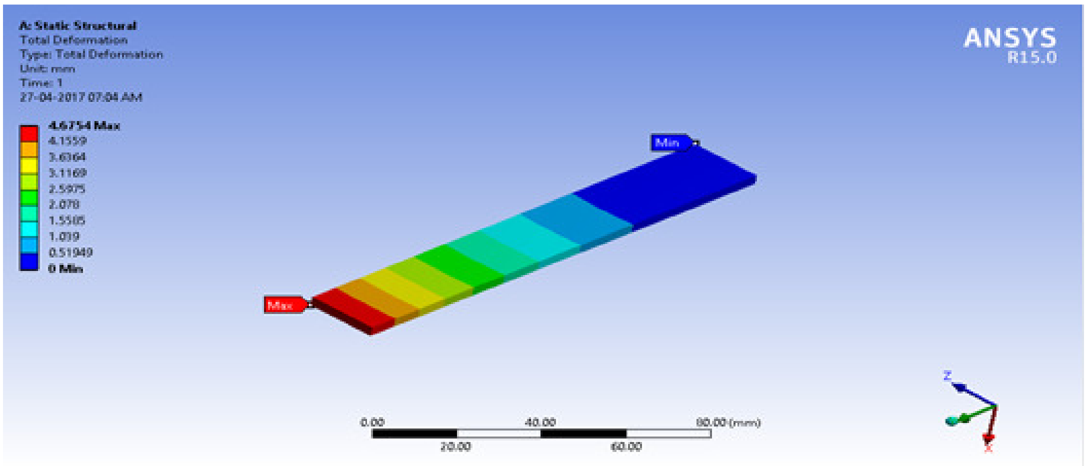

Figure 3 shows Total Deformation

Carbon – E glass Hybrid Composite

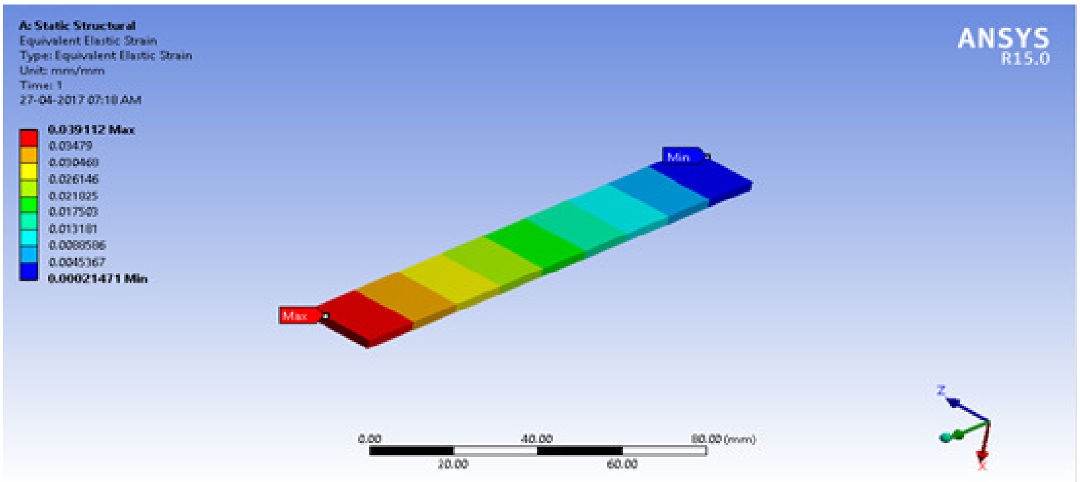

Figure 4 shows Equivalent Strain

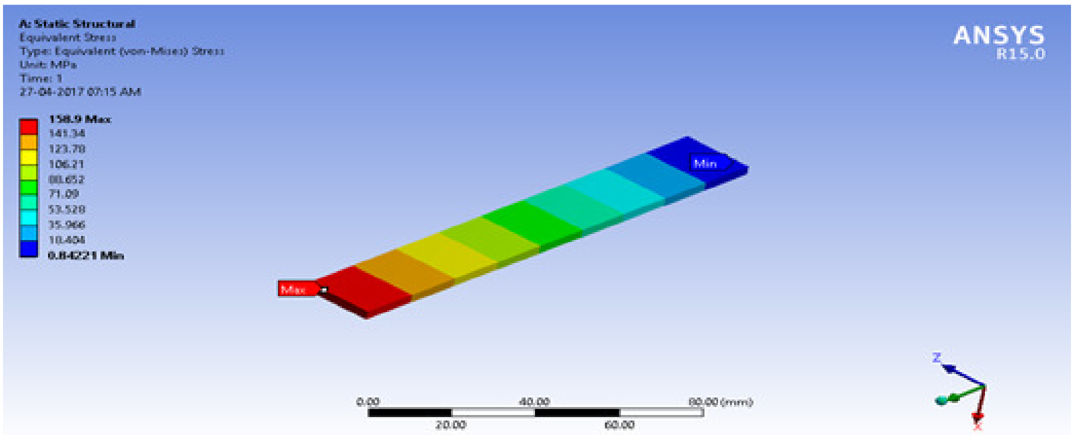

Figure 5 shows Von-Mises Stress

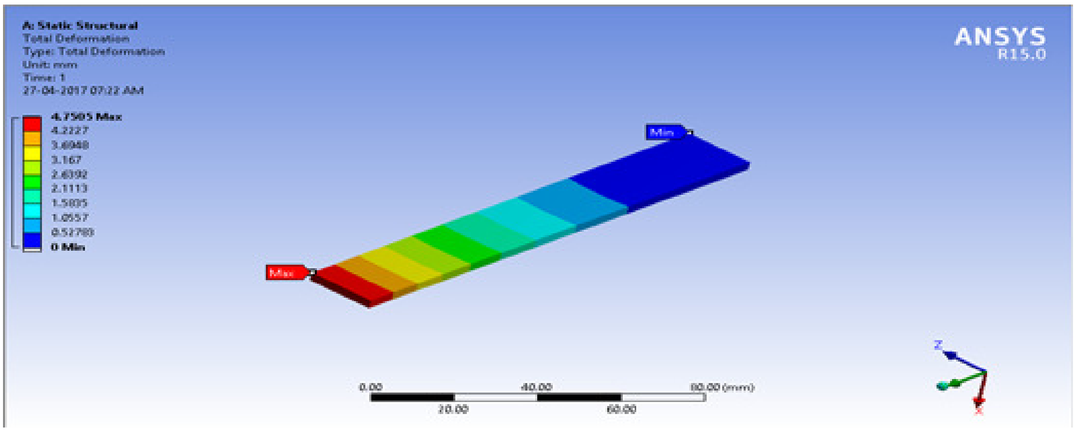

Figure 6 shows Total Deformation

The tensile load test results of the two hybrid materials Carbon-Kenaf and Carbon-E glass after numerical analysis is shown below.

Table: 3.2.1 Numerical results for Tensile Load Test

ANSYS PARAMETERS	**MATERIALS USED**					
	Carbon – Kenaf Hybrid			**Carbon – E Glass Hybrid**		
MESH SIZE	COARSE	MEDIUM	FINE	COARSE	MEDIUM	FINE
EQUIVALENT STRAIN	0.03699	0.037205	0.03742	0.038158	0.038397	0.039112
VON-MISES STRESS (MPa)	161.67	163.33	167.68	153.33	155.0	158.9
TOTAL DEFORMATIO N (mm)	4.6518	4.6636	4.6754	4.7241	4.7373	4.7505
NODES	683	2198	8865	683	2198	8865
ELEMENTS	80	280	1496	80	280	1496

3.2.2. Flexure Load Test Results

Carbon - Kenaf Hybrid Composite

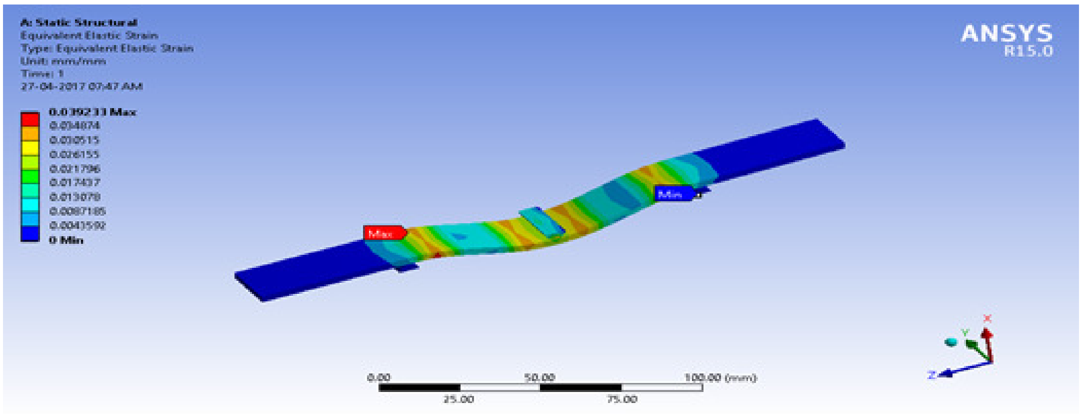

Figure 7 shows Equivalent Strain

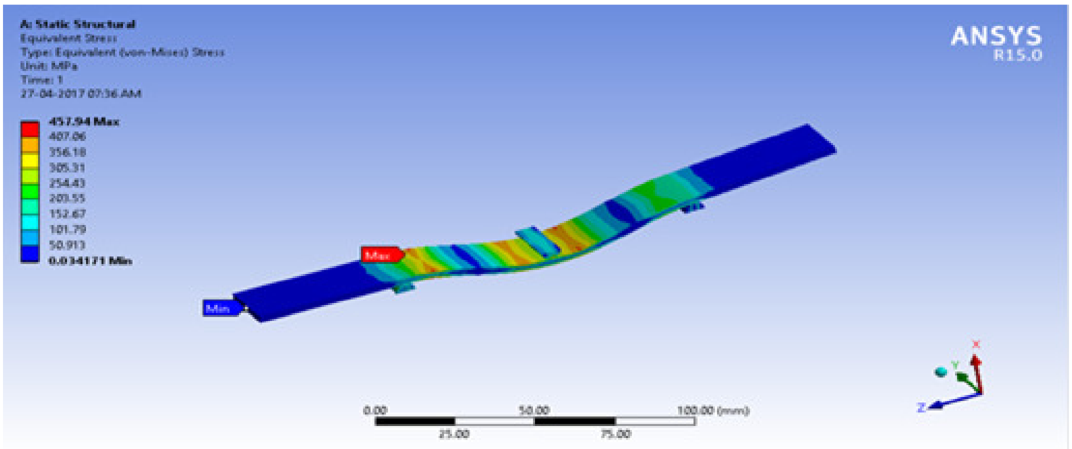

Figure 8 shows Von-Mises Stress

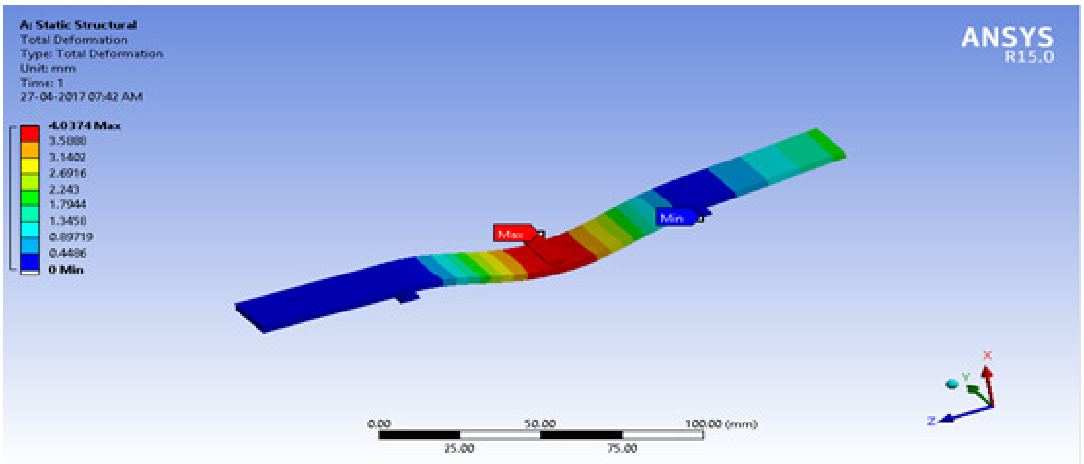

Figure 9 shows Total Deformation

Carbon – E glass Hybrid Composite

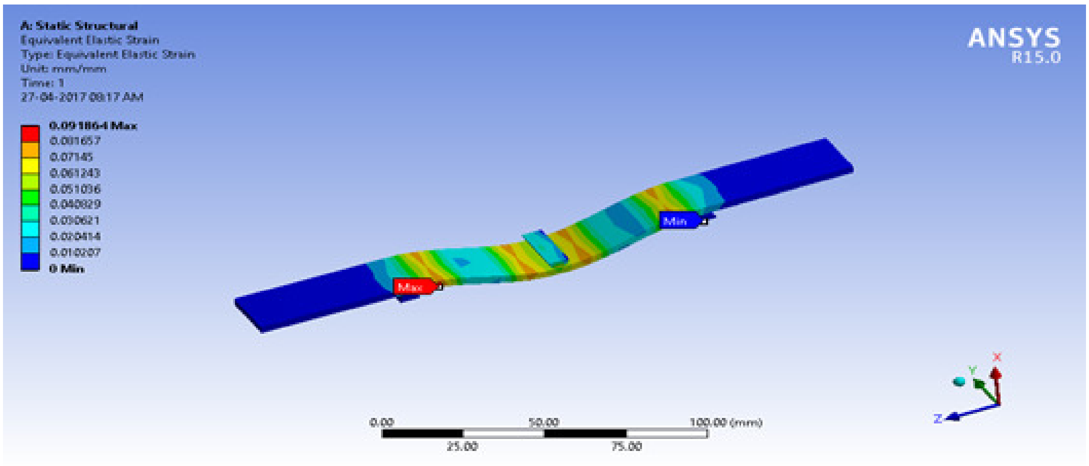

Figure 10 shows Equivalent Strain

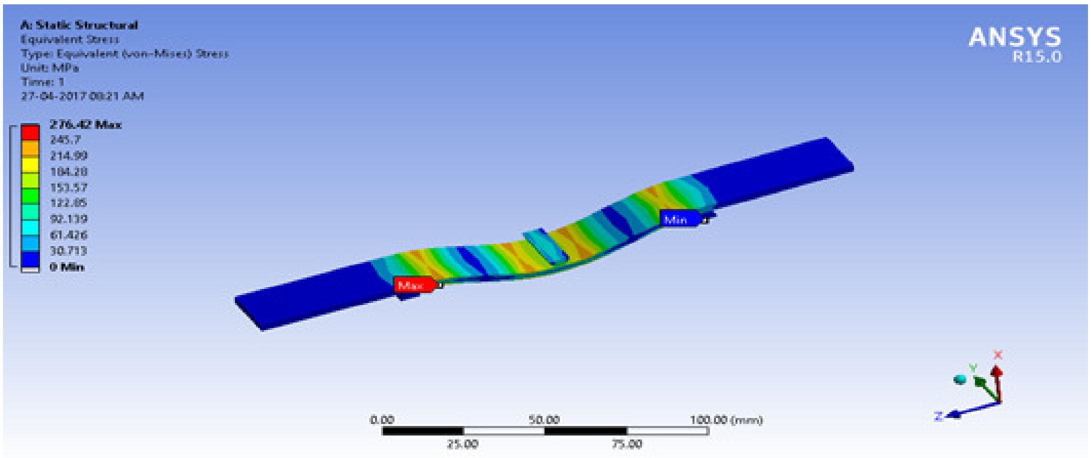

Figure 11 shows Von-Mises Stress

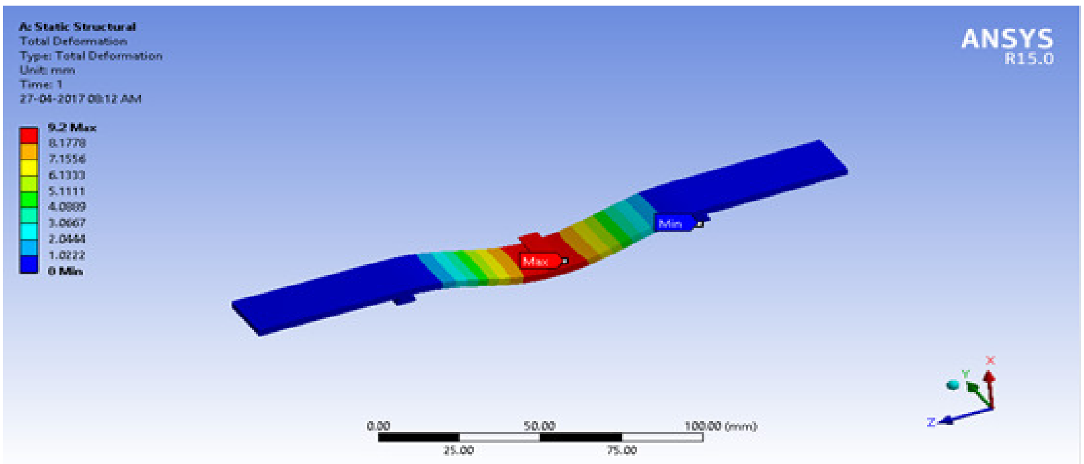

Figure 12 shows Total Deformation

Table 3.2.2 Numerical Results for Bending Load Test

ANSYS PARAMETERS	MATERIALS USED					
	Carbon – Kenaf Hybrid			Carbon – E Glass Hybrid		
MESH	COARSE	MEDIUM	FINE	COARSE	MEDIUM	FINE
EQUIVALENT STRAIN	0.038877	0.038966	0.039233	0.090964	0.091324	0.091864
VON-MISES STRESS (MPa)	447.86	451.42	457.94	269.24	272.83	276.42
DEFORMATION (mm)	3.9312	3.9539	4.0374	9.131	9.154	9.2
NODES	2008	2311	4502	2008	2311	4502
ELEMENTS	898	1077	2056	898	1077	2056

3.3. Validation of Results

The experimental and numerical analysis of the test specimen was conducted. Inorder to check its accuracy, a validation study is required. For that, the results obtained from both analyses needs to be compared and to check its percentage variation. If the variation is within 15%, the considered parameters are right. Otherwise, there will be certain error in assumptions. The values considered for validation was of fine meshed ones, since it shows accurate results comparing the coarse and medium meshes. The considered no. of nodes and elements in fine mesh is high compared to other meshes and as the area under consideration reduces, the accuracy of results increases. Thus due to all this facts, fine mesh values are selected for validation purposes.

Table 3.3.1 Validation of tensile load test parameters

PARAMETERS	MATERIALS			
	Carbon – Kenaf Hybrid		Carbon – E glass Hybrid	
	Experimental	Numerical	Experimental	Numerical
Equivalent Strain	0.036	0.03742	0.03853	0.039112
Von Mises Stress (MPa)	167.673	167.68	158.951	158.9
Total Deformation (mm)	4.74	4.6754	4.87	4.7505
% Variation (Strain)	-	3.79	-	1.58
% Variation (Stress)	-	<1	-	<1
% Variation (Deformation)	-	1.4	-	2.45

The validation results of tensile load test shows the variations of stress, strain and deformation were within the limit and so that the considered parameters were right and the materials can be preferred for further analysis. Fine mesh values were considered for validation of numerical results since fine mesh values always shows most accurate value.

Table 3.3.2 Validation of bending load test parameters

PARAMETERS	MATERIALS			
	Carbon – Kenaf Hybrid		Carbon – E glass Hybrid	
	Experimental	Numerical	Experimental	Numerical
Equivalent Strain	0.0394	0.039233	0.0921	0.091864
Von Mises Stress (MPa)	462.63	457.94	282.96	276.42
Total Deformation (mm)	3.94	4.0374	9.21	9.2
% Variation (Strain)	-	<1	-	<1
% Variation (Stress)	-	1.013	-	2.31
% Variation (Deformation)	-	2.47	-	<1

The validation results of bending load test also shows the variations were within the limit compared to experimental values. Fine mesh values were considered for validation of numerical results since fine mesh values always shows most accurate value.

3.4. Energy Absorption Comparison with Conventional Materials

From the obtained values, it was found that among the hybrid materials, the better one was Carbon-Kenaf hybrid. It has better stress, strain properties than Carbon-E glass hybrid and can be used in energy absorbing applications. Inorder to check whether it can replace Steel or Aluminium, certain factors need to be considered. The energy absorption test considered was bending test.

Table 3.4.1 Comparison of materials based on energy absorption

MATERIALS	Density (kg/m^3)	Peak Load (N)	Strain Energy (mJ)	Deformation (mm)	SEA (J/kg)
Carbon Kenaf Hybrid	1650	555.15	20.129	7.4347	27.35
Carbon E glass Hybrid	1470	339.554	23.772	14.852	18.15
Stainless Steel	7850	460	0.45418	0.33222	2.90
Aluminium	2770	430	2.1135	1.1802	10.77

Specific Energy Absorption is found out by the equation, $E_S = \frac{E}{\rho\delta A_{mat}}$ where E is the strain energy obtained in mJ, δ is the deformation in mm obtained from ANSYS Workbench 15.0. and A is the area of specimen. The peak load for Stainless steel and Aluminium were obtained from ANSYS considering safety factor value as 1.

From the table, it was found that hybrids shows better energy absorption than Stainless steel and Aluminum. Also due to its less weight, the overall curb weight will be reduced and thus efficiency can be increased. Among hybrids, Carbon-Kenaf shows better property as an energy absorber as it has high specific energy absorption compared to other materials.

4. RESULTS AND DISCUSSIONS

From the experimental test results, for tensile load test, stress, strain and deformation values for both experimental and numerical results came similar. From flexure test too, stress, strain and strain energy was observed to be almost similar. From the strain energy absorption comparison results, it was found that hybrids shows very good properties than Structural steel and Aluminium in case of Energy absorption. Thus it was found that hybrids can use as energy absorbers than conventionally used materials. The better material among hybrids was found to be Carbon-Kenaf hybrid composite.

5. CONCLUSION

In the present work, numerical analysis was conducted on test specimen by introducing hybrid composites as materials inorder to validate experimental analysis results. The measured parameters were Equivalent Strain, Von-Mises Stress and Total Deformation. The results were validated and values obtained were found to be much similar. The energy absorption characteristics of conventional materials were compared with hybrid composites later and it was found that hybrids shows very high energy absorption properties compared to materials used as energy absorber in automobiles. Also it was found that Carbon Kenaf hybrid was better among hybrids as it has low density, higher load carrying capacity and SEA compared to other materials. SEA value varies upto 89% compared to Steel while 60% compared to Aluminium for Carbon Kenaf hybrid. Because of all these properties, this material can be preferred as a replacement for stainless steel used in automobiles as energy absorbers like front or rear bumpers as well as side intrusion beams.

REFERENCES

[1] Nithin Prasad, Adarsh Ramachandran, *Experimental Analysis of Hybrid Carbon Fiber Composite Specimen*, International Research Journal of Engineering and Technology (IRJET) Volume: 04, Issue: 4 April-2017.

[2] K. Veeraswamy, V. Venkata Sudheer Babu, *Design and Analysis of a Composite Beam for Side Impact Protection of a Car Door*, International Research Journal of Engineering and Technology (IRJET) Volume: 03, Issue: 2 Feb-2016.

[3] Harun Bayraktar, David Ehrlich, Jon Goering, Michael McClain, *3D Woven Composites for Energy Absorbing Applications*, 20th International Conference on Composite Materials, Copenhagen, 19-24th July 2015.

[4] Prashanth Turla, S. Sampath Kumar, P. Harshitha Reddy, K. Chandra Shekar, *Processing and Flexural Strength of Carbon Fiber and Glass Fiber Reinforced Epoxy-Matrix Hybrid Composite*, International Journal of Engineering Research and Technology (IRJET) Volume: 03, Issue: 4 April-2014.

[5] Shinji Ochi – *Mechanical properties of kenaf fibers and kenaf/PLA composites*, Science Direct Elsevier, Mechanics of Materials 446–452, Oct-2007.

[6] Analysis of Tensile Behavior Hybrid Carbon - Jute Fiber Reniforced Epoxy Composite, Sandeep. B, Dr. K.S Keerthi Prasad and Girish. T.R, International Journal of Mechanical Engineering and Technology. Volume 5, Issue 9, September (2014), pp. 51-55.

[7] Abhishek Singh, B. R. Phanikumar and Ram Prasad. Effect of Geogrid Reinforcement on Load Carrying Capacity of A Coarse Sand Bed, International Journal of Civil Engineering and Technology, 7(3), 2016, pp. 01–06.

[8] Mohd Fadzli Abdollah, Rafidah Hassan, *Preliminary Design of Side Door Impact Beam for Passenger Cars Using Aluminium*, Center of Advanced Research on Energy, University Technical Malaysia Melaka - 2007.