ISSN 2347 - 3983
Volume 8. No. 5, May 2020
International Journal of Emerging Trends in Engineering Research
Available Online at http://www.warse.org/IJETER/static/pdf/file/ijeter110852020.pdf
https://doi.org/10.30534/ijeter/2020/110852020

Explicit Dynamic Analysis of Industrial Helmet Manufactured Using Bamboo and Vakka Fiber Composite

Somasundaram A[1], Anish Kumar M[2], Ashok R[3], Bhoopathi D[4]
[1]Kongu Engineering College, India, somas@kongu.ac.in
[2]Kongu Engineering College, India, anishkumarm.16mech@kongu.edu
[3]Kongu Engineering College, India, ashokr.16mech@kongu.edu
[4]Kongu Engineering College, India, bhoopathid.16mech@kongu.edu

ABSTRACT

Today, safety of workers in workplaces is a major concern and hence the Occupational Safety and Health Administration has made the usage of helmet in industrial sites as mandatory which in turn protect the workers from head injuries. The work presented in this paper considers the standard design of industrial helmet used and the same geometry is used in the analysis of helmet shell fabricated using bamboo and vakka fiber reinforced composite. The current analysis focuses on the impact stress on helmet by Finite Element Analysis through dynamic stimulation method using ANSYS software. The model has been designed in CATIA V5 and meshed with required element size. After modeling is done, the analyses have been done using ANSYS. The use of hybrid composite material in this application have increased properties such as strength and hardness of the helmet proposed. The results obtained from analysis have been displayed for various boundary conditions.

Keywords: Bamboo fiber, Boundary conditions, Displacement, Dynamic stimulation, Finite Element Analysis (FEA), Hybrid composite, Occupational Safety and Health Administration (OSHA), Stress, Vakka fiber

1. INTRODUCTION

Adhering to the norms of using Personal Protective Equipments (PPE) is essential in industrial applications to protect the workers from hazards. Helmet is a type of protective device worn on the head to guard against injury and protect the head of the user by absorbing mechanical impact energy if any and defending from hazards. The volume and weight of the helmet is also an important issue as greater volume and weight ratio increases the injury risk for users head directly and neck indirectly. This research focus on the implementation of Finite Element Analysis in the development of helmet shell used in industrial safety applications when helmets are made by hybrid natural fiber composite techniques. The use of natural fiber reinforced composite for helmet and other technical application is considered due to its high strength to weight ratio and hardness.

Anil kumar K et al. [1] conducted a study on mould flow analysis on helmet using proE and concluded that Nylon 4-6 plastic is better compared to ABS plastics when used in the manufacturing of helmets in injection moulding process. Smith et al. [2] carried out 3D Finite Element Analysis of helmet elements and concluded that the usage of liner in helmet increased the energy absorbing capacity and the liners are manufactured from various material. Gilchirst et al. [3] found that the design is inadequate for side, front and back protection through mathematical model. Thomas et al. [4] carried out impact analysis on Glass Fiber Reinforced Polymer (GFRP) composite helmet using ANSYS software and concluded that reinforced composites have better mechanical properties than conventional materials. Wazerya et al. [5] developed a composite using E-glass fiber with varying percentage of fiber which influenced the mechanical properties such as tensile strength, hardness and flexural bending strength. Divakar H et al. [6] carried out the tensile and compression properties of ABS laminates with multi-layer glass fiber. Addition of glass fiber increased the tensile and compression properties and decreased the bending strength of ABS. Elanchezhian et al. [7] discussed the mechanical behavior like tensile, flexural and impact strength of the GFRP and CFRP laminates. Orientation of fiber in the laminates has played a vital role in mechanical properties of GFRP and CFRP. The tensile and flexural strength of CFRP is high than GFRP whereas, impact strength of GFRP is high compare to CFRP. Ismaeel, L.M.A et al. [8] discussed about the assumptions made for analyzing discontinuous random orientation fiber reinforced composite.

Historically the study of bones indicated that cranial bones are higher weight bearing bones in human body and these bones get injured commonly. Thurman et al. [9] found the compressive failure of dipole as 25.1 ± 13.3 MPa which lies between trabecular bone and cortical tables of skull. Dempster et al. [10] studied the importance of structural, material anisotropy in mechanical response of cranial bone and also discussed the mechanical significance of the various pattern in skull. The architectural features of the skull, their adequacy and limitations are analyzed in terms of their reaction to force systems and their proneness to fracture. Khan et al. [11] analyse the dynamic properties of aluminum and GFRP and CFRP composite ship propeller and concluded that composite propeller has more stiffness than aluminum propeller. SBVJ et al. [12] made found the tensile and compression strength of sisal fiber reinforced composite and conclude that it have high potential to withstand impact.

2. METHODS

The standard design of industrial helmet has been inherited. Geometrical values of Helmets with Indian Standards 2925 (1984) have been absorbed for the proposed analysis. In the dynamic impact analysis of industrial helmet ANSYS WORKBENCH 18.1 is used and the stages of simulation are explained in three steps:

Step 1: Preprocessing

The pre-processing, also called model preparation is often the most work intensive step of the FEA. The helmet disassembly into smaller parts, the finite (small, but finite) elements are done. This process is called meshing. Then a Computer Aided Design (CAD) model of helmet were created using CATIA V5 software with required dimensions as shown in Figure1.

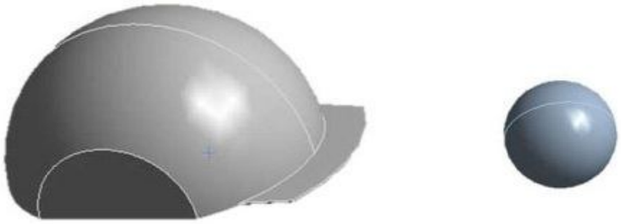


Figure 1: CAD model of Helmet and Steel Ball

This simulation considers the thickness of the shell as 5 mm and ball of diameter 4 inches (i.e 101.60 mm) with mass around 4kg. In this process ANSYS Workbench 18.1 software is used for making Finite Element Analysis. The models are brought in appropriate extension (*.stp, *.igs,) are imported into ANSYS 18.1 Workbench and the meshes were generated. The mesh is generated according to the design constraints and it can be of different elements according to the design such as 1D, 2D or 3D elements. For meshing the element size is kept as 10 mm, the minimum edge length as 4.98830 mm and the span angle is as course and these conditions are set in advance. In FEA, the meshes are the essential part for the complete process where the complex structural model is partitioned into small discrete elements. After mesh has been done it is noted that number of nodes in helmet model, ball is 11534 and 468 respectively. Helmet model is discreted into 33976 elements as shown in Figure.2

Figure 2: Meshed Helmet Model

Step 2: Processing

When the model is meshed into discrete elements the further step is called processing. At this phase, the elements are entrusted its material and also it's mechanical properties. Property values like material density, Young's Modulus, Poisson's Ratio, Ultimate and Yield Stresses are fed. These data is inherited from engineering material database in ANSYS 18.1. For new material (bamboo and vakka fiber composite) the properties are obtained from the experimental work and fed into ANSYS by adding new material as shown in Table 1 which has isotropic elastic properties.

Table 1: Material Properties

Properties	Composite Helmet	Steel ball	ABS
Youngs Modulus (GPa)	7.106	200	3.944
Shear Modulus (GPa)	2.548	79.615	0.854
Poisson Ratio	0.394	0.3	0.349
Density (g/cm^3)	1.275	7.85	1.04

In this phase, contact conditions between helmet and ball for dynamic impact analysis is also described as shown in Figure 3. The boundary conditions are added at this phase to the helmet design in terms of initial velocity, height and angle of impact of the object to be dropped. Finally processing velocity and number of intervals are assigned and the iterative process is ready to run the simulation. For dynamic impact analysis four processing velocities of ball were considered with respect to corresponding height. The impact conditions depend on the height of the object over the helmet that falls on the helmet. This analysis considers four such heights 5m, 20m, 46m and 80m by using formula V2=2gH (where g – gravitational acceleration 9.81m/s) the corresponding speeds are calculated as 10m/s, 20m/s, 30m/s and 40m/s respectively. After setting all these conditions, dynamic analysis iteration run is allowed in ANSYS.

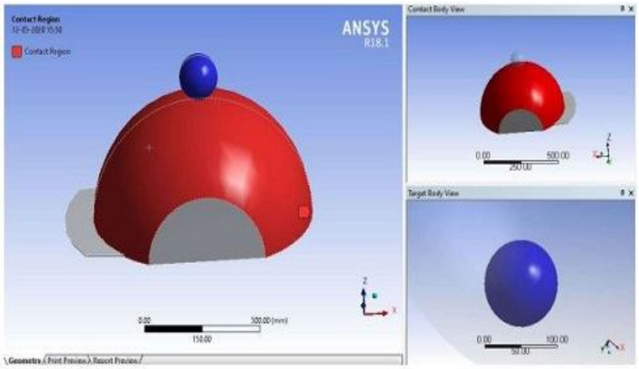


Figure 3: Contact region between helmet and steel ball

Step 3: Post Processing

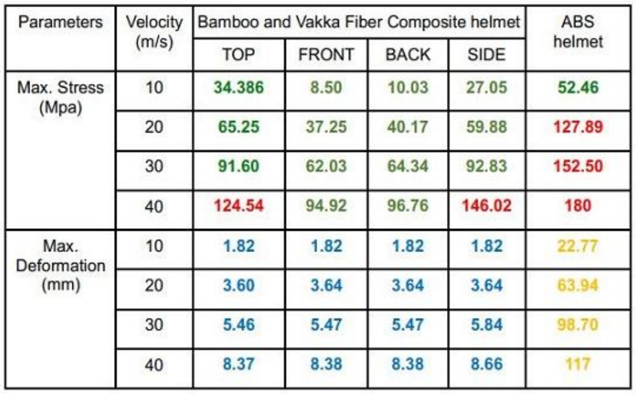
In this processing the highly detailed and complex outputs of Finite Element Method (FEM) calculations are transformed into a format that is easily understood by the user. The post-processing outputs are used for functionality checks on the FEM model and for reporting results. By completing the above procedures, the next step in the post processing part is to interpret the results obtained in the simulation. The stress and displacement plot of the assembled body are got at the end of this process.

At last, it can be noted that these models do not have the ability to produce fracture, but quite assume that the material will undergo continues deformation. Thus, it offers only qualitative assessment of the helmet which protect skull. Additionally, actual models ought to be created, that embody simulating fracture which might be useful to helmet manufactures. This needs far more detailed material models, which are not accessible and are beyond the scope of this work.

3. RESULT AND DISCUSSION

The aim of this analysis is to see how the use of bamboo and vakka fiber composite in industrial helmets can reduce the chances of serious damage to human skull and protect the workers from hazard. The results of the analysis are discussed below in Table 2.

Table 2: FEM analysis result comparison of ABS and Fabricated composite



Parameters	Velocity (m/s)	Bamboo and Vakka Fiber Composite helmet				ABS helmet
		TOP	FRONT	BACK	SIDE	
Max. Stress (Mpa)	10	34.386	8.50	10.03	27.05	52.46
	20	65.25	37.25	40.17	59.88	127.89
	30	91.60	62.03	64.34	92.83	152.50
	40	124.54	94.92	96.76	146.02	180
Max. Deformation (mm)	10	1.82	1.82	1.82	1.82	22.77
	20	3.60	3.64	3.64	3.64	63.94
	30	5.46	5.47	5.47	5.84	98.70
	40	8.37	8.38	8.38	8.66	117

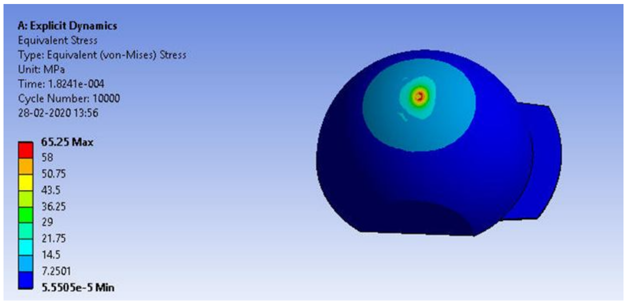
Under the circumstances considered, the bamboo and vakka fiber composite helmet showed efficacy when velocity of falling object is up-to 30 m/s since the maximum yield stress of human skull is about 100MPa whereas, the conventional material (ABS) helmet fails to withstand impact at 20 m/s. So, it summarized that bamboo and vakka fiber composite material gives more rigidity to the human skull from impact load.

4. CONCLUSION

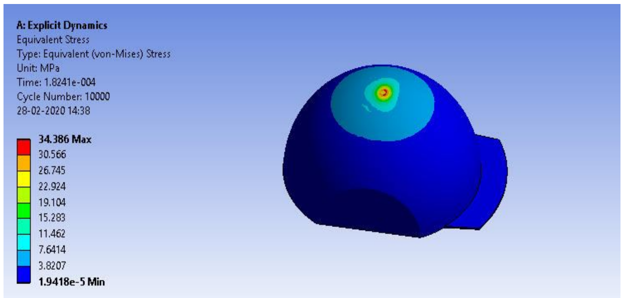
Explicit dynamic analysis is carried out on the existing designed helmet by using ANSYS software and it shows that bamboo and vakka fiber composite material have capability to replace the conventional material used for manufacturing the safety helmet and other applications. For obtaining more precise and accurate results, it is necessary to improve the refinement of boundary conditions, improve the quality of the meshes and allow the simulation process to go on for increased time period. All these things would improve the quality of results obtained.

APPENDIX

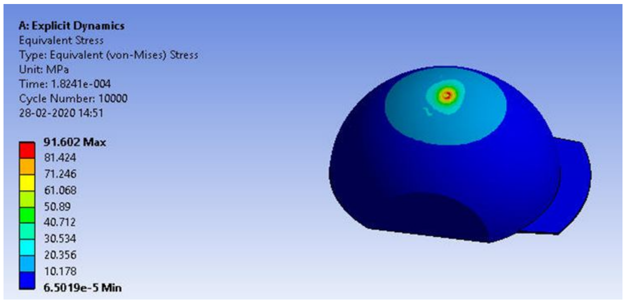
Von-Mises Stress plot of bamboo and vakka composite helmet for different velocities at which the steel ball hits the helmet



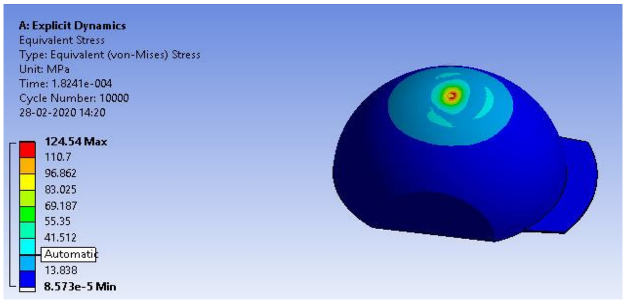
For 10 m/s



For 20 m/s

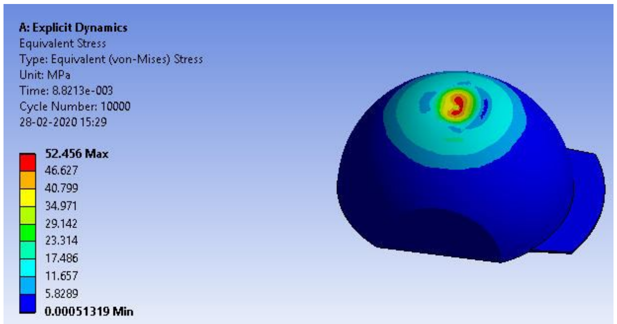


For 30 m/s

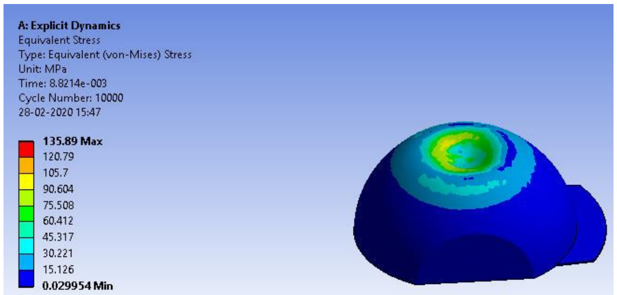


For 40 m/s

Von-Mises Stress plot of ABS helmet for different velocities at which the steel ball hits the helmet



For 10 m/s



For 20 m/s

REFERENCES

1. Anil Kumar, K. and Suresh Babu, Y., 2013. **Design and analysis of industrial helmet**. Int. J. Comput. Eng. Res, 3, pp.48-53.
2. Smith, T., Lenkeit, J. and Boughton, J., 2005. **Application of finite element analysis to helmet design.** In IUTAM Symposium on Impact Biomechanics: From Fundamental Insights to Applications (pp. 255-262). Springer, Dordrecht. https://doi.org/10.1007/1-4020-3796-1_26
3. Gilchrist, A. and Mills, N.J., 1987. **Construction site workers helmets**. Journal of occupational accidents, 9(3), pp.199-211. https://doi.org/10.1016/0376-6349(87)90012-5
4. Thomas, A., Suresh, P., Thilak, J.A.J. and Subramani, N., 2017. **Impact analysis on composite helmet by using frc and glass fiber by using ANSYS.** Stress, 31(38.3062), pp.43-2435.
5. El-Wazery, M.S., El-Elamy, M.I. and Zoalfakar, S.H., 2017. **Mechanical properties of glass fiber reinforced polyester composites**. International journal of applied science and engineering, 14(3), pp.121-131.
6. Divakar, H., Nagaraja, R., Puttaswamaiah, S. and Guruprasad, H.L., 2015. **Mechanical characterization of thermoplastic ABS/glass fibre reinforced polymer matrix composites**. Int J Eng Res Technol, 4, pp.1127-1131. https://doi.org/10.17577/IJERTV4IS051122
7. Elanchezhian, C., Ramnath, B.V. and Hemalatha, J., 2014. **Mechanical behaviour of glass and carbon fibre reinforced composites at varying strain rates and temperatures**. Procedia Materials Science, 6, pp.1405-1418. https://doi.org/10.1016/j.mspro.2014.07.120
8. Ismaeel, L.M.A., 2015. **Optimization and static stress analysis of hybrid fiber reinforced composite leaf spring**. Advances in Materials Science and Engineering, 2015. https://doi.org/10.1155/2015/374609
9. Thurman, D. and Guerrero, J., 1999. **Trends in hospitalization associated with traumatic brain injury**. Jama, 282(10), pp.954-957. https://doi.org/10.1001/jama.282.10.954
10. Dempster, W.T., 1967. **Correlation of types of cortical grain structure with architectural features of the human skull**. American Journal of Anatomy, 120(1), pp.7-31. https://doi.org/10.1002/aja.1001200103
11. Khan, M.A., Uddin, K.S. and Ahmed, B., 2013. **Design and Dynamic analysis on composite propeller of ship using FEA**. Internationl Journal of Advanced Trends in Computer Science and Engineering, 2013.
12. SBVJ, C.B., Arun, K. and Eswaraiah, B., 2015. **Fabrication and testing of natural (sisal) fibre reinforced polymer composites material**. International Journal of Engineering Technology and research, 3(5).