Numerical Analysis of Engine Hood

Ms Varimadugu Sandhya[1], MsYarramsetty Nagini[1] , Ms Venkata Sushma chinta[1], Mr V.Jaipal Reddy[1],MsJyothirmayi Narne*[1]

1Assistant Professor, Department of Mechanical Engineering
Chaitanya Bharathi Institute of Technology, Hyderabad-75, Telangana, India
** Corresponding author – N.Jyothirmayi,njyothirmayi_mech@cbit.ac.in*

Abstract- Weight reduction is one of the major factor in automobiles for improving the fuel efficiency. In this paper the conventional metal used for engine hood is replaced with the Fiber reinforced plastics (FRP). A three dimensional solid model of the engine hood is prepared. Mid surface of the solid model is generated and imported for analysis. The surface model is first checked for continuity as very small details are lost during importing. The material properties are defined and the mesh is generated. Boundary conditions and loads are applied. By varying the thickness numerical static analysis is performed and optimum value of the thickness for the surface is obtained.

Keywords Engine hood, FRP, Surface Model, Static analysis

INTRODUCTION

FRP composites are being promoted as materials for the century because of their superior properties like high strength-to-weight ratio, corrosion resistance, and excellent thermo mechanical properties.

The UD-GFRP composite laminates were tested for failure with drilled hole under uni-axial tensile testing by developing a finite element model [1].

The elastic properties under the effect of hygrothermal conditions on polymeric composite materials is investigated by developing a micromechanical degradation model [2].

The strength failure of the guyed tower was evaluated using the maximum strain, maximum stress theories, and Tsai Wu failure criterion and also the dynamic response at any location along tower height was calculated by using the conservative formula for simple scaling [3].

Fracture analysis conducted on Titanium alloys has revealed quasi-cleavage fracture with small symptoms of plastic shearing in the early part of cracking [4].

Between steel and fiber-reinforced composites the effects of fiber type and fiber orientation’s on the interface bonding is investigated for three different composites: glass fibers,carbon fibers/epoxy and Kevlar fibers/epoxy. These were tested under different orientations.To determine modes I and II fracture toughness the end-notched flexure and Double cantilever beam tests were utilized respectively. Results showed that the interface bonding between is significantly affected by composite fiber orientations and fiber types [5].

The Experimental and the numerical (finite element analysis) analysis using different geometric sections on buckling loading conditions of carbon fiber reinforced plastics (CFRP) layered composite plates is analysed.With position of hole in plateit is noted that the buckling load/unit length varies. [6].

METHODOLOGY

The purpose of the numerical analysis is to study the behavior of an actual engineering system. The analysis must be an accurate mathematical model of a physical prototype. The finite element model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system.

The three steps followed for the Static analysis are Model generation, apply the loads and obtain the Solution, Review the results

The term model generation means the process of defining the geometry configuration of the model's nodes and elements. A three dimensional model is built and mid surface is generated. The file is saved in a neutral format .IGES and imported to ANSYS.

Composites are difficult to model than an isotropic material, special care is needed while defining the properties and orientation of various layers. The material properties of the composite considered are Poisson's ratio=0.2, Young's Modulus = 9 x 10^{-5} N/mm^2, Rigidity Modulus= 3450 x 10^2 N/mm^2.

The two elements considered for analysis are Shell-63 and Shell-99. Mapped mesh is done on the model. A mapped mesh typically has a regular pattern. A line and area concatenation helps in getting mapped mesh. The meshed model is shown in Figure 1.

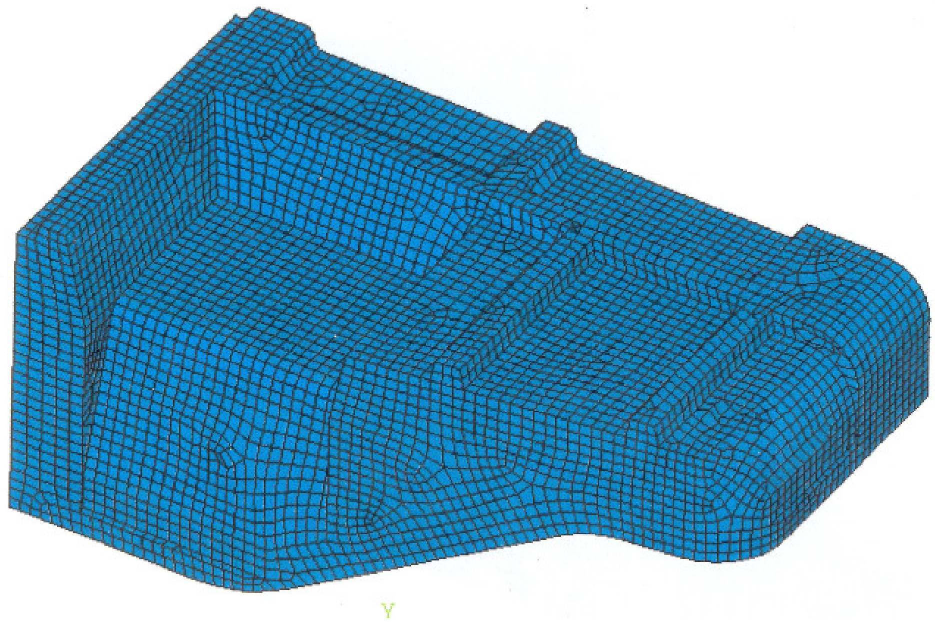

Figure 1: FE model of Engine Hood

The axis symmetric boundary conditions are considered. The axis symmetry is set on edge and the dof constraints are applied on the remaining model. A load of 240 kg is applied on the model.

Considering the yield strength of the composite material the minimum thickness considered is 3.75 mm. The analysis is done for different thicknesses ranging from 3.75, 3.9,4, and 4.2 mm respectively for shell element.

RESULTS AND CONCLUSIONS

The post processing of the numerical analysis is done and the contour plots are extracted for the model. The Displacement contours in X,Y and Z directions are shown in the figure 2, figure 3 and figure 4.

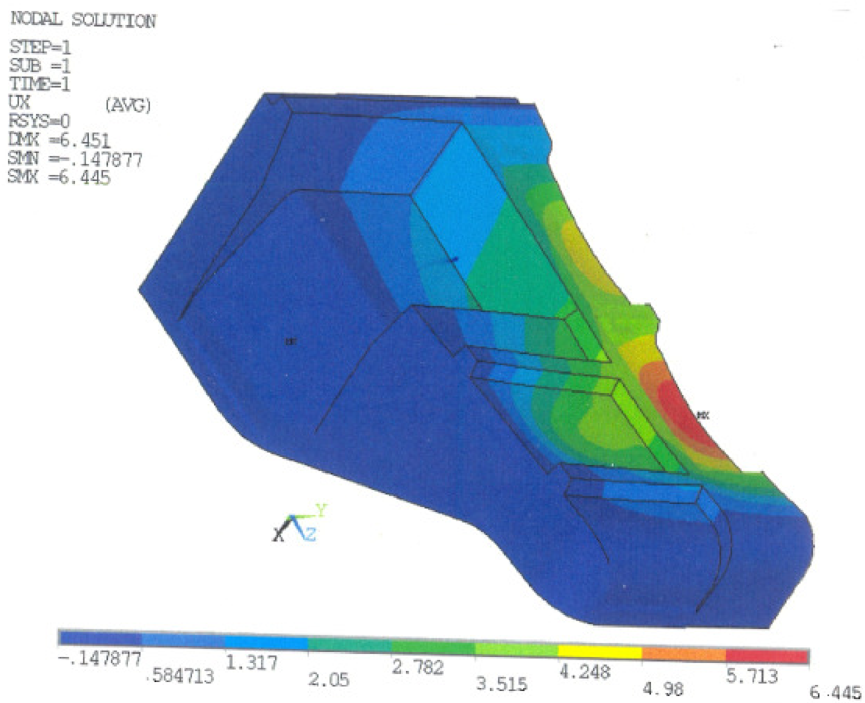

Figure 2: Displacement in X Direction

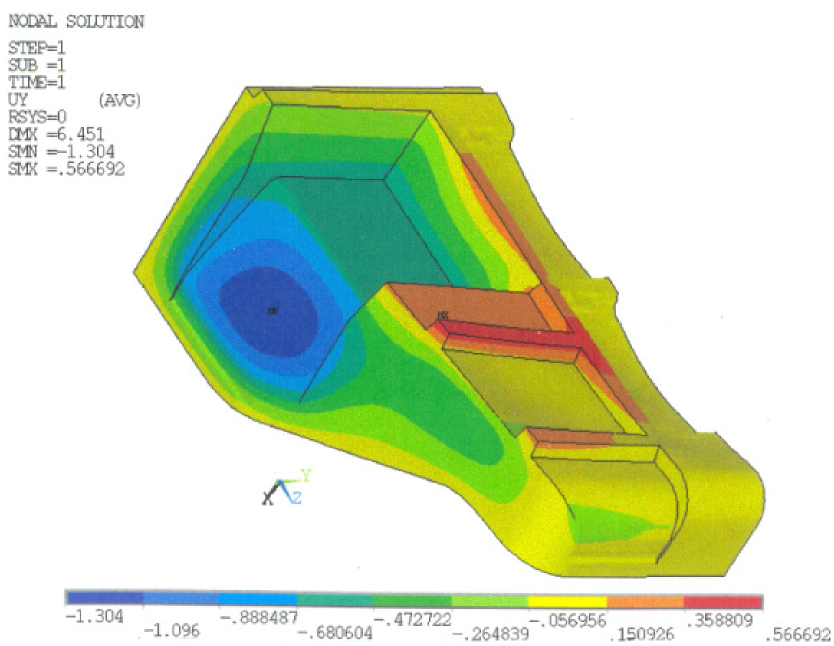

Figure 3: Displacement in Y Direction

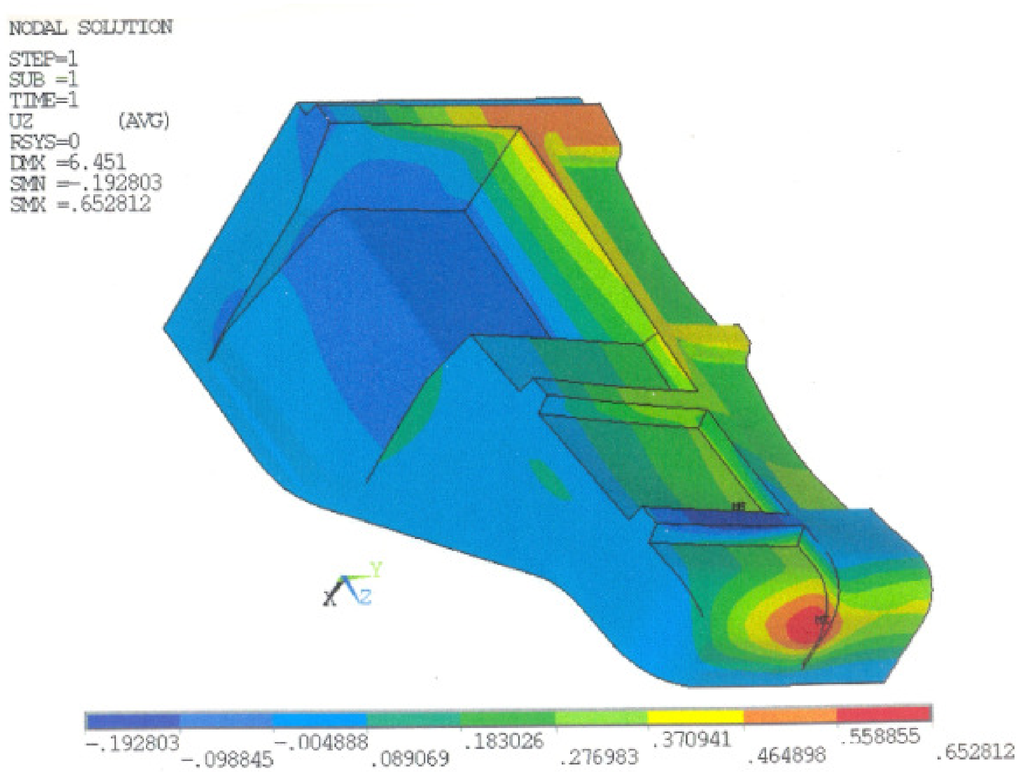

Figure 4: Displacement in Z Direction

The minimum deflection and maximum deflection values are tabulated and presented in the table 1

Table 1: Minimum Deflectionand MaximumDeflection

Thickness of layered element (mm)	Minimum Deflection (mm)			Maximum Deflection (mm)		
	Ux	Uy	Uz	Ux	Uy	Uz
3.75	-0.1724	-1.578	-0.3437	7.539	0.687	0.7497
3.9	-0.1895	-1.672	-0.3903	6.977	0.6383	0.6966
4	-0.1478	-1.304	-0.1928	6.625	0.6087	0.6668
4.2	-0.1452	-1.263	-0.271	6.003	0.5554	0.6122

The contour plots for stresses in X,Y and Z directions are shown in figure 5 , figure 6 and figure 7

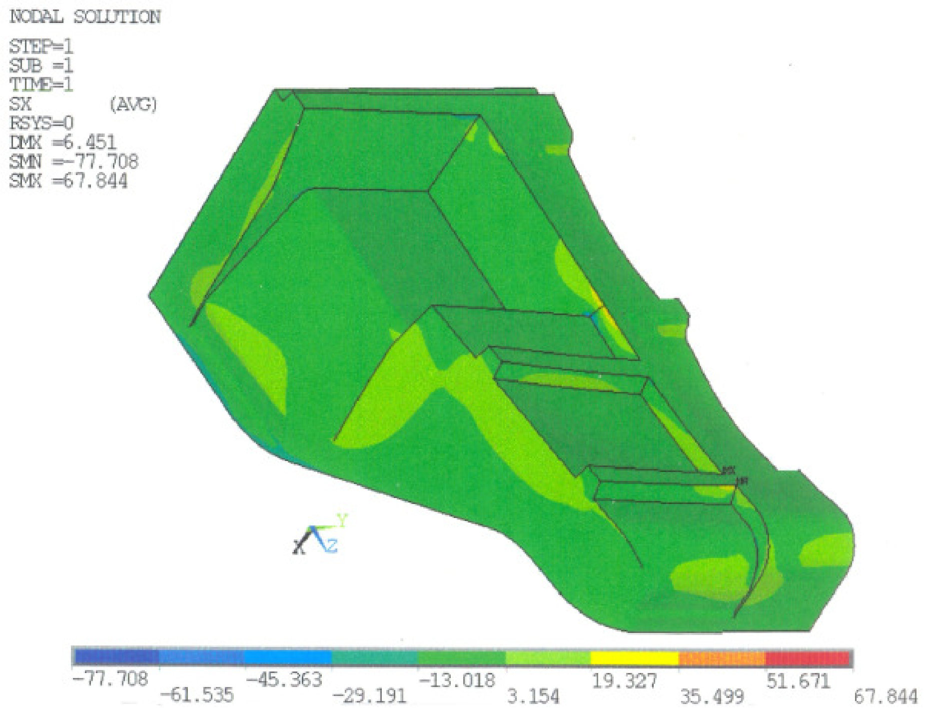

Figure 5: Stress in X direction

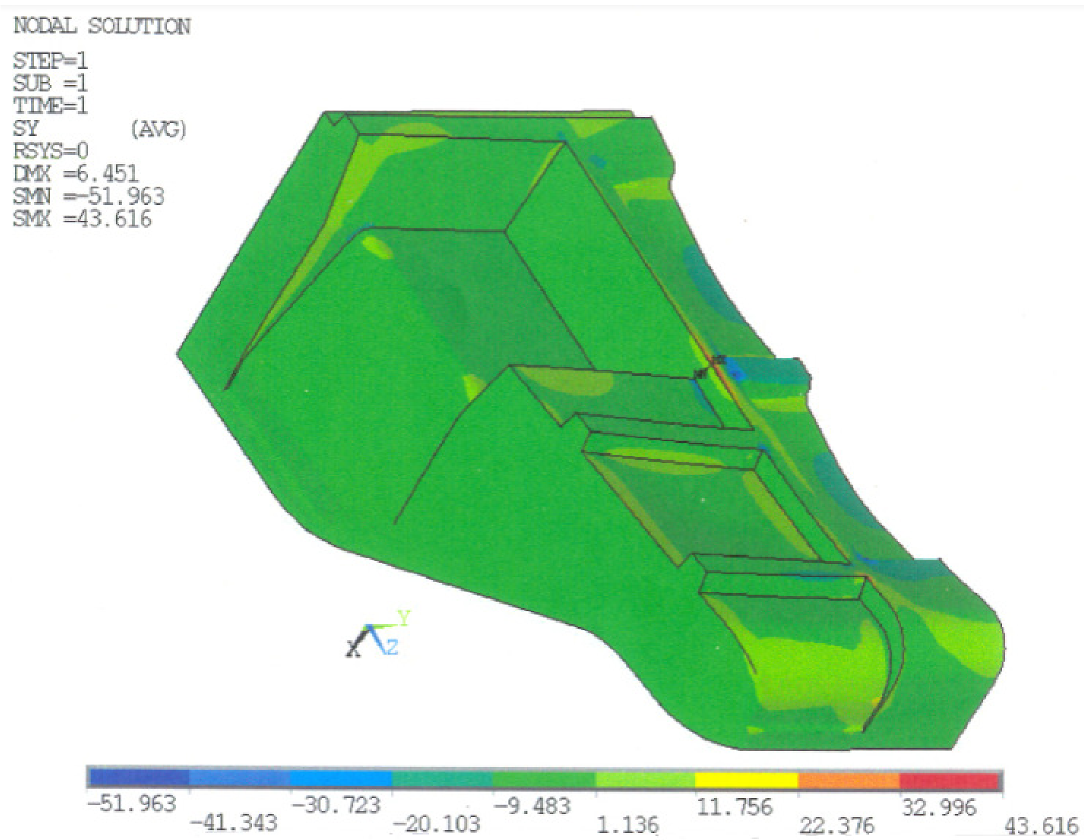

Figure 6: Stress in Y direction

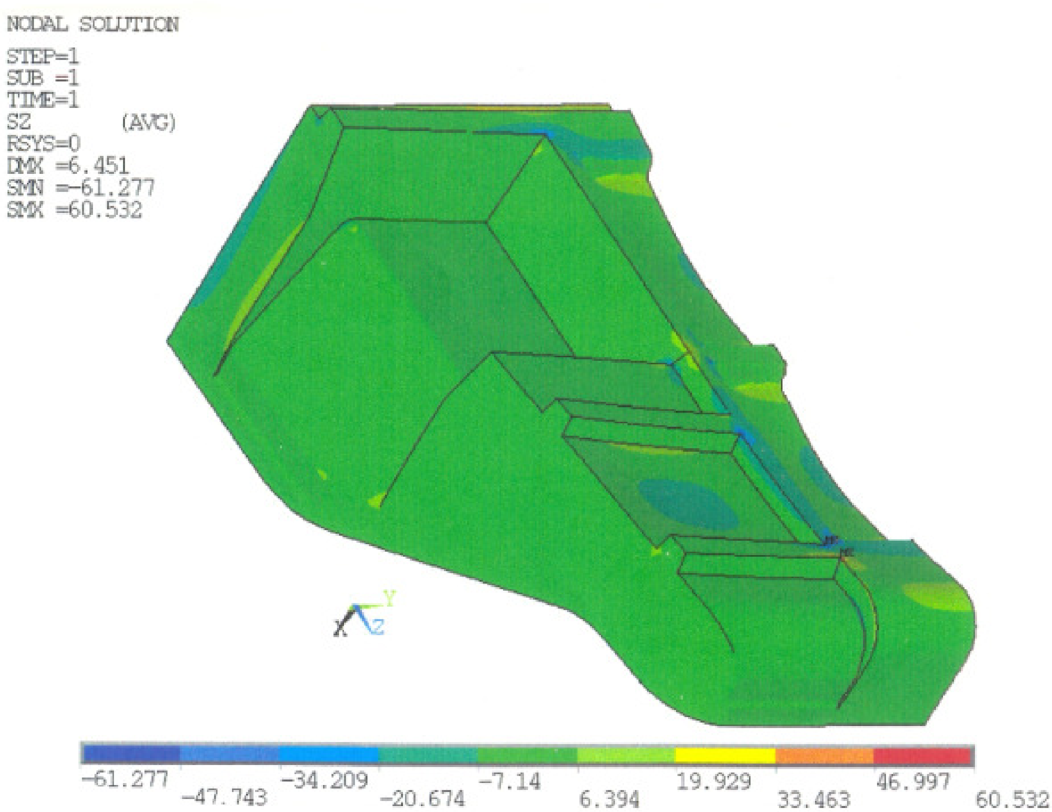

Figure 7: Stress in Z direction

The stress values are tabulated and presented in the table 2

Table 2: Minimum stress and Maximum stress

Thickness of layered element (mm)	Minimum Stress (N/mm^2)			MaximumStress (N/mm^2)		
	Sx	**Sy**	**Sz**	**Sx**	**Sy**	**Sz**
3.75	-79.201	-63.54	-72.687	73.501	55.074	76.738
3.9	-85.142	-67.385	-78.796	69.422	51.468	70.314
4	-77.708	-51.963	-61.277	67.844	43.616	60.532
4.2	-68.997	-56.808	-62.261	62.27	45.06	59.486

We observe that for the thickness of 3.75 mm the stresses in the Sx and Sz directions are highest. The large value of stresses reflect an area of concentrated stress applied at a pointed place. This is highly undesirable. The thickness of 3.9 mm is further checked for stress variation. After number of iterations it is found that the optimal thickness is 4 mm.

At 4 mm thickness as the value of the stresses conform to the values given in the design specification, we can conclude that 4 mm thickness is good for the maximum efficiency of the model.

REFERENCES

1. Rakesh, P. K., Singh, I., Kumar, D. (2010). Failure prediction in glass fiber reinforced plastics laminates with drilled hole under uni-axial loading. Materials & Design, 31(6), 3002–3007.

2. Meghdad Gholami , HamedAfrasiab, Ali Mohammad Baghestani, Alireza Fathi "Hygrothermal degradation of elastic properties of fiber reinforced composites: A micro-scale finite element analysis" , Composite Structures, Volume 266, 15 June 2021,113819.

3. Sami Alshurafa, Hanan Alhayek, Dimos Polyzois " Finite element method for the staticand dynamic analysis of FRP guyed tower", Journal of Computational Design andEngineering, Volume 6 , Issue 3, July 2019, Pages- 436-446.

4. Jyothirmayi Narne, Varimadugu Sandhya , Nagini Yerramsetty and V. Jaipal Reddy, "Experimental and Numerical Investigations on Crack Propagation in Titanium Alloys", International Journal of Current Engineering and Technology, Vol.8, No.4, Pages- 922-

5. Fatima Ghassan Alabtah, E Mahdi, " The effect of sizing optimization on the interface between high strength steel and fiber reinforced composite", Composite Structures, Volume 266, 15 June 2021, 113740

6. Jeyaprakash, P., Prabhakaran, V., & Devaraju, A. (2018). Experimental and Numerical

Analysis of Carbon Epoxy Fibre Composite under Buckling Load. Materials Today: Proceedings, 5(6), 14526–14530.

7. Michael Franz, Benjamin Schleich, Sandro Wartzack, "Influence of layer thickness variations on the structural behavior of optimized fibre reinforced plastic parts", Procedia CIRP, Volume 85, 2019, Pages 26-31

8. Keisuke Tanaka, Kazuya Oharada, Daiki Yamada, Kenichi Shimizu, " Effect of test temperature on fatigue crack propagation in injection molded plate of short fiber reinforced plastics", Procedia Structural Integrity, Volume 2, 2016, Pages 58-65