ISSN XXXX XXXX © 2016 IJESC

Research Article Volume 6 Issue No. 12

Design and Numerical Investigation of Hexagonal Honeycomb Sandwich Panels

Manjunatha M[1], S.N.Amith kumar[2]
PG student[1], Assistance Profressor[2]
Department of Mechanical Engineering
Dr.Ambedkar Institute of Technology Bangalore, India

Abstract:
The weight optimization is special concern in present day's technology. Composites are more interested because of higher strength to weight ratio.in composites Sandwich composites are the special type of structures that are replaced with existing structures design. Sandwich panels are made of weak core section bonded by strong face sheet. In this work, the solid deflector used in missile launcher vehicle to protect the vehicle controller besides the pilot cabin, are to be replaced by sandwich panels with same mechanical strength. For this sandwich panel is designed based on the data hand book available in open sources and this sandwich panel is studied under static and dynamic load cases. Analysis carried on sandwich panel with boundary condition prescribed by data hand book. And it validate by ABAQUS static FE code. In another case the transient load due to uncontrolled thrust force created during launching time is assumed to be TNT equivalent load and dynamic simulation is carried out with ABAQUS/EXPLICIT with TNT explosive load defined by incident wave property in abaqus/CAE. The structure is results within permissible design limit in both face sheet and core section. And it is safe under static load condition. In ABAQUS/explicit analysis energy balance methods are used validate numerical results and PEEQ is plastic equivalent strain within material elongation. Sandwich panel is can use as alternative to solid deflector. By this cost of fuel can be reduces, it depend on the weight of vehicle.

Keywords: Composites, sandwich panels, hexagonal honeycomb core, finite element methods, abaqus/explicit

I. INTRODUCTION

Sandwich structure is commonly used to achieve strength, stiffness, and weight efficiency compared to conventional structure. Most commonly, Sandwich Panels are used in Aircraft, Space craft, Satellites, Automobiles, Trains, Trucks, Boats etc. [1]. The strength is mainly depends on the weight consumption mainly in aerospace and military application. From experiment and numerical investigation [2] and [3] will shows the cellular structure which have properties superior than material with higher strength to weight ratio. Hexagonal core section is more energy absorption because of large number of sides. The regular arranged core section is have more resistance to dynamic crushing as compared to irregular section which leads to arrange the core section with regular hexagonal pattern in solid steel plate that is to reduce its weight with same working performance is carried in this work. [3] and [4]. The uncontrolled thrust load arise at that the starting is can be assumed as the TNT eqvalent load [5] and can be considered as the CONWEB surface blast load in the structure.

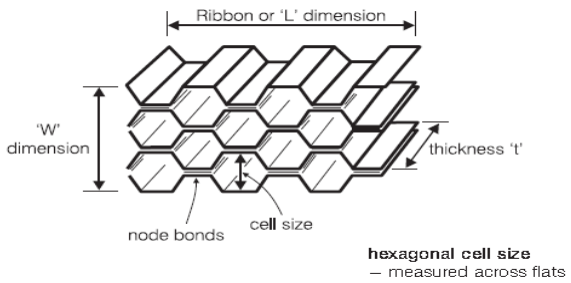

Figure.1. Basic configuration ofhexagonal core structure[1]

The basic concept of a sandwich panel is that the faceplates carry the bending stresses whereas the core carries the shear stresses. The core plays a role which is analogous to that of the I beam web while the sandwich facings perform a function very much like that of the I beam flanges

II.OBJECTIVE OF WORK:

The aim of this work is to design and dynamic analysis of sandwich panels subject to non-linear transients load.

- The preliminary step is to design and model of hexagonal sandwich panel with help of design data handbook and Finite element analysis to estimate strength of the sandwich panel with static load.
- Comparative study of result between static FE results and theoretical calculations and a non –linear finite element analysis for the dynamic blast load on sandwich panel is to be performed.

III.DESIGN OF HONEYCOMB SANDWICH

The design of basic configuration is carried by using the Open source data handbook.it is used to calculate strength of sandwich panel's [6].
Determine Plate Coefficient

$$\frac{b}{a} = \frac{260}{260} = 1 \qquad \text{(equation 1)}$$

R=L/W = $$R = \frac{220L}{112} = 1.98$$

$$V = \frac{\pi^2 E\, tf\, h}{2b^2\, Gw\,(1-\mu^2)} \qquad \text{(Equation 2)}$$

$$V = \frac{\pi^2 * 165 * 4 * 15}{2 * 260^2 * 112\,(1-.25^2)} = 0.98$$

From data book

K_1 = 0.018

K_2 = 0.03

K_3 = 0.042

Deflection $\delta = \frac{2K1q\ b4\ (1-\mu 2)}{Ef\ tf\ h2}$ (equation 3)

$$\frac{2 * .011 * 5 * 260^4 (1-.25^2)}{165 * 4 * 15} = 3.2 \text{ mm}$$

Facing Stress: - $\sigma f = \frac{K2\ q\ b2}{htf}$ (equation 4)

$$\sigma f = \frac{0.03 * 5 * 260^2}{15 * 4} = 180MPa$$

Core Shear: -$\sigma c = \frac{K3\ q\ b}{h}$ (equation 3)

$$\sigma c = \frac{0.042 * 5 * 260}{15} = 3.5MP_a$$

IV.FINITE ELEMENT ANALYSIS

Honeycomb sandwich plate is analysis when is subjected dynamic loading steel plate made of hollow section and its corralte with experimental results and honerycomb sandwich panels subjected high impact loading [7] and [8] which leads use the numerical methods to validate the results Commercially available ABAQUS software is used in our work to perform both static and dynamic analysis. The basic geometric configuration of sandwich is designed by open source data hand book [1]. And it modeled by commercially available modeling software CATIAV5. The Figure-2 show the assembly of missile launcher vehicle with solid deflector is modeled with sandwich plate and vehicle is modeled for visualization purpose. And it is not considered in result post processing.

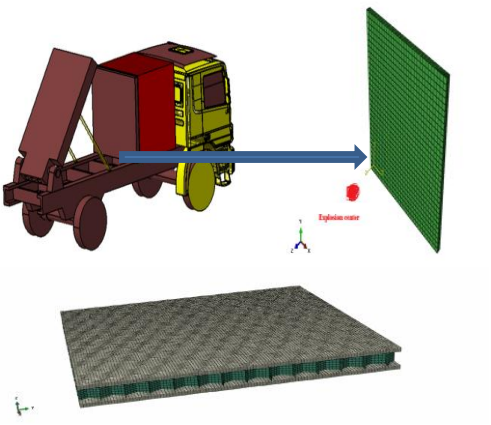

Figure.2. Finite element modeling of sandwich panels with representation of solid plate with vehicle

The sandwich panel is converted in finite element model by using the ABAQUS meshing module. The face sheet is modeled with solid element which has translation degree of freedom only and it's used to represent bending stiffness of sandwich panel.

Table .2. element details of core and face sheet

	Type of element	Number of elements
core	S4R	22472
Face sheet	C3D20R	89880

Similarly core section is model by shell element to resembles the both shear and bending stiffness and complete details on number of elements and nodes is given in table-2 and the material are selected as the core section is modeled with homogenous shell section to represent both shear and bending stiffness and face sheet assign with the solid section and the bonding between face sheet and core is done by using tie constant in abaqus which allows the complete bonding. And material properties are summarizes in Table-3

Table-3 Core and face sheet material properties

	CORE material(aluminium5052)	Face sheet material(steel 1006)
Young modulus	165MPa	205MPa
Poisson ratio	0.25	0.29
Density	40kg/m^3	7800kg/m^3
Compression strength	0.9MPa	285MPa
Shear strength	0.65MPa	150MPa

V.STATIC ANALYSIS

The linear static analyasis is carried out on the sandwich plate to check basic design of structure is safe under static loading condtion. All degree of freedom is constrained at edges of sandwich plate as shown in Figure- 3. The staic pressure of 1Mpa is applied on upper face of sandwich

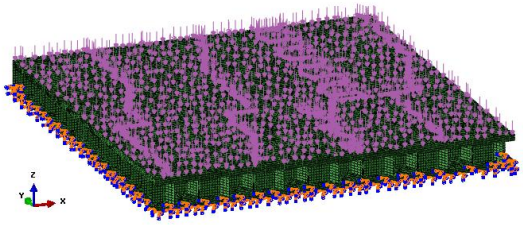

Figure .3. Boundary condition for static analysis

The sandwich panel when it subject to the static loading, analysis is carried with the abaqus/standard. The load of 1Mpa is applied on the face of sandwich to with constant time period of 60 seconds. The displacement of plate when static load of 1Mpa is applied in load module on upper face of sandwich plate.

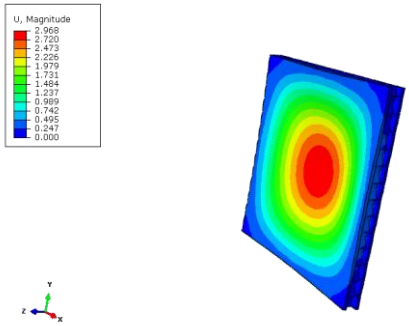

Figure.4. Displacement countour plot in static analysis

Maximum Displacement of sandwich of 2.98mm is observed at center of plate. In countor plot the red colour indicates maximam displacement and blue colour indicate the minimum value of displacement.

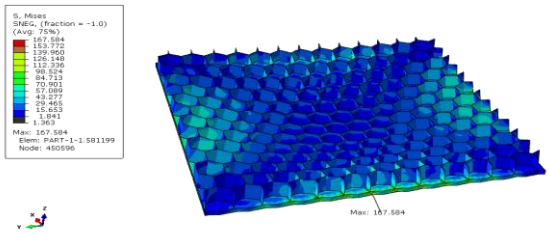

Figure . 5. Von mises stress countour plot in static analysis

Figure 5 shows the maximum von mises stress in sandwich assembly under static load condition. The von mises stress of 167Mpa is results at the weaker region and its shown in red colour in countor plot.

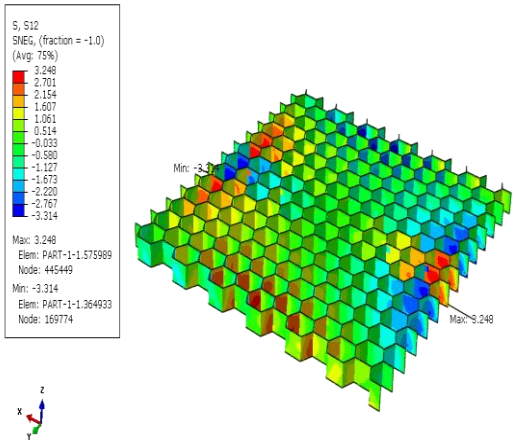

Figure . 6. Shear stress countour plot in static analysis

As we observed in literacture works , the core section is to withstand the shear load and it hexagonal core is modelled by shell element in abaqus .the shear stress in normal direction for appiled load. The maximum 3.2Mpa and and minimum principle stress is -3.3Mpa is is shown in Figure 6

VI.DYNANIC ANALYSIS

Abaqus explicit time intigration method is used to analysis the sandwich plate under transient dynamic load.Thurst load arises when misssile launching time in opposite direction of missile propagated direction. The scaled distance is referred to 200 mm[9].and when for weight of 6gm of charge is used in loading condition and its taken as the INCIDENT WAVE in abaqus/cae .

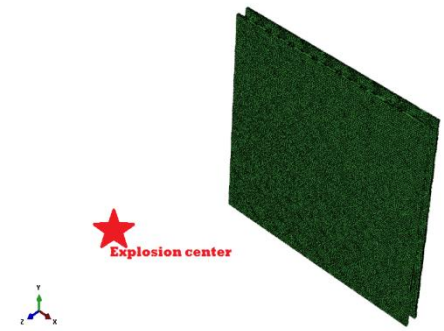

Figure . 6. Loading configuration for blast loading which detonate at scaled distance in dynamic analysis

the charge weight is convertred to TNT eqivalent calculated by following formula [9].

$$W=W_{explosion}$$

$$\frac{heat\ generated\ in\ c4\ explosive}{heat\ generated\ in\ TNT\ explosive} = 6 * \frac{5.86}{4.50} =$$

$$8gm\ of\ TNT$$

the energy relished in explosion of 8gm of ton of TNT is 32.472 joules.

When sandwich plate subjected to transient load of TNT equivalent. The dynamic analysis is carried out defined the energy transfer from center of detonation point to the sandwich is analysis by explicit time integration in abaqus .the results obtained are explained below

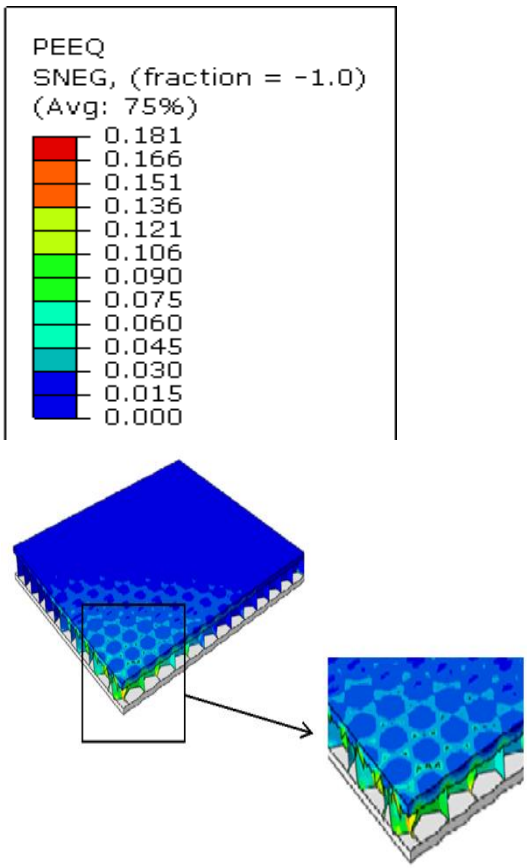

Figure. 7. Plastic equilvant elongation in sandwich plate

The plastic elongation in core section materials of 3% and plastic elongation by numerical methods is of 0.1% and Total energy of system is constant and it the kinetic energy is transformed into structure. Internal energy is increases. Figure- 8 shows the energy distribution.

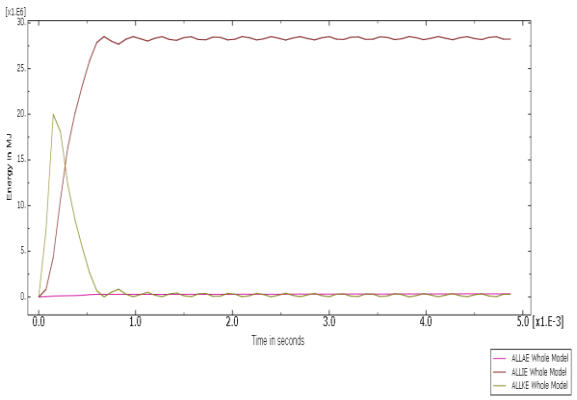

Figure . 8. Energy Distrubution in dynamic analysis

The internal artificial energy is generated due to the mesh distortion during analysis. This energy is to be within range of 6% of total energy then we can say the solution is converges. In this work we get lesser than 2% artificial energy.

VII. CONCLUSION AND FUTURE WORK

- The sandwich plate is designed based on the hex web data book and static analysis is carried out the von misses stress value and shear stress is obtained in static analysis is 180MPa and its within permissible material properties.so structure when its subjected to the static load is safe within its design stress.
- the deformation value of sandwich plate is 2.4 mm and its comparable with the theoretical values

Table.7.1 Resultant Table for static analysis

	Analytical methods	FE methods
Deflection in mm	3.2	3.0
Face sheet stress in MPa	180	167
Core shear stress in MPa	3.5	3.2

- The dynamic analysis is carried out on sandwich. the plastic equivalent output shows that the sandwich plate is more energy as its yield in plastic condition and it shows the honeycomb sandwich plate is suitable for the replacement for solid plate
  The laboratory scale test is to be conducted as the future work to understand the response of sandwich plate to dynamic time–load history. And the experimental work and numerical analysis is to be carried for different material configuration to get accuracy and energy absorption.

VIII.REFERANCES

[1]. A Chawla S, Mukherjee, Dileep Kumar M. Ueno Dept. of Mechanical Engineering Indian Institute of Technology

[2]. Amy.F.domae. "An investigation on cell shape in honeycomb structure vs. structure strength'' 1997

[3]. Gibson and Ashby Xing et al 'an experimental investigation solid core structure under dynamic loads ''2010

[4].Amin Ajdari, Dynamic crushing and energy absorption of cellular structure 2010

[5].S. K. Lahiri and L. Ho, "Simulation of rapid structural failure due to blast loads from conventional weapons (CONWEP)" in Proceedings of the NAFEMS World Congress, Boston, Mass, USA, May 2011

[6]. Hex web honeycomb sandwich design technology design data hand book

[7] .Sebastian Hems, "Dynamic Shear Rupture of Steel Plates," Journal of Mechanics of Materials and Structures, vol. 2–10, pp. 2049–2066,

[8]. K. P. H. N. G. Wadley, Z. Xue, and J. W. Hutchinson, "Mechanical Response of Metallic Honeycomb Sandwich Panel Structures to High-Intensity Dynamic Loading," Journal of Impact Engineering, vol. 35, pp. 1063–1074, 2008

[9].5th European LS_DYNA user's conference used in LS – DYNA REFERNCE MANNUAL