Design and Finite Element Analysis of Sandwich Panels Subjected to Static Loads

Manjunatha M[1], S.N. Amith kumar[2]
[1, 2] Department of Mechanical Engineering
[1, 2] Dr. Ambedkar Institute of Technology Bangalore-560056

Abstract- *The weight optimization is special concern in present day's technology. Composites are more interested because of their higher strength to weight ratio. Sandwich composites are the special type of structures and made of weak core section bonded by strong face sheet. In this work, Sandwich panel is designed based on the data hand book available in open sources under static loading and design is validate by numerical ABAQUS finite element code. From both numerical and analytical methods, the stress developed face sheet are within permissible limit and its shows sandwich panels are safe under designed load and it can be used to alternative for the solid geometry under static loading cases.*

Keywords- Composites, sandwich panels, honeycomb core, finite element methods, static load, and abaqus/std

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 depending on the weight consumption mainly in aerospace and military application. From experiment and numerical investigation [2] and 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. [3]- [4] The regular arranged core section is having more resistance to dynamic crushing as compared to irregular section which leads to arrange the core section with regular hexagonal pattern [5] both tensile and bending strength of fiber laminates is studied in both numerical methods. And a good agreement in both analytical and numerical results.

The theatrical design and analysis of honeycomb when it subjected to pure bending is d carried to investigate modes of failure and its exceeds of its properties. [6]. In this work metallic sandwich plate is studied under static load. the following figure-1 shows the honeycomb sandwich core section.

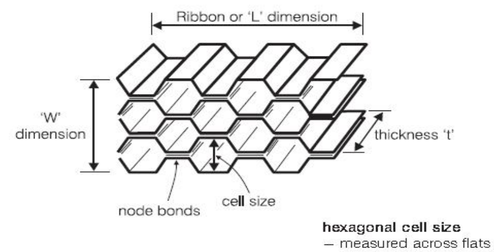

Figure-1 Basic configuration ofhexagonal core structure[1]

II. OBJECTIVE OF WORK

The aim of this work is to design and dynamic analysis of sandwich panels

- The preliminary step is to design and model of hexagonal sandwich panel with help of design data handbook
- Finite element analysis to estimate strength of the sandwich panel with static loading condition.

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

$$\frac{b}{a} = \frac{260}{260} = 1 \quad \text{(Equation 1)}$$

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

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

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

From data book

K_1	=	0.018
K_2	=	0.03
K_3	=	0.042

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

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

IV. FINITE ELEMENT 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

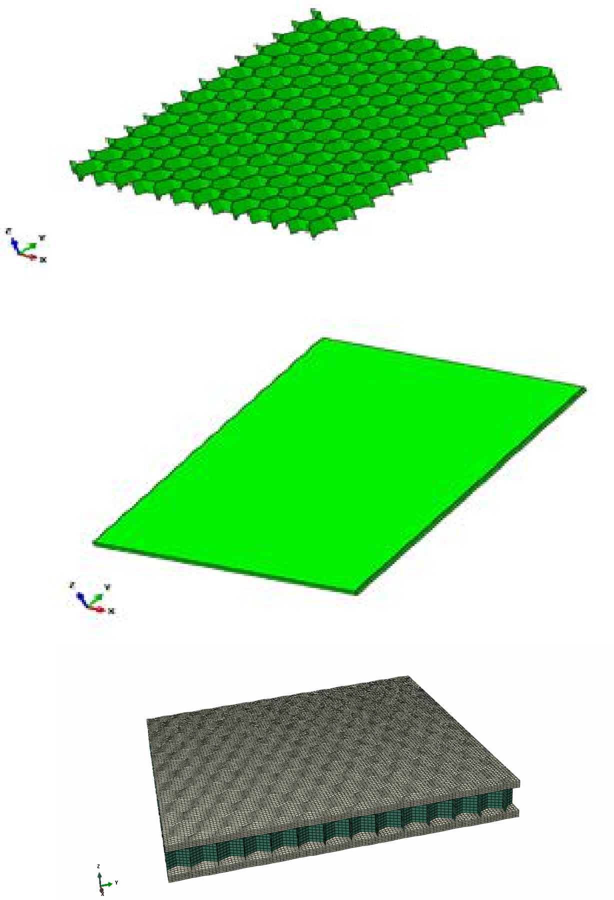

Figure -2 Finite element modeling of sandwich panels

The sandwich panel is converted in finite element model by using the ABAQUS meshing module [7]. 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 abacus which allows the complete bonding. And material properties are summarizing 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 analayisis is carried out on the sandwich plate when it subjected to 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

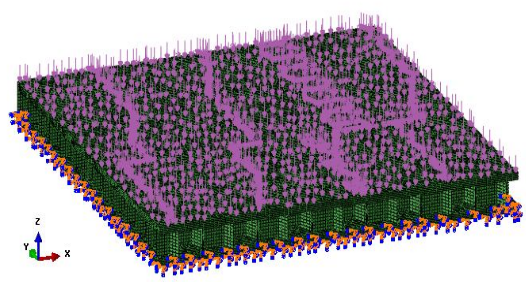

Figure -3 Boundary condition for static analysis

. 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.

VI. RESULTS AND DISCUSSION

The sandwich panels are designed with help of data hand book and its results is validating by the finite element methods.

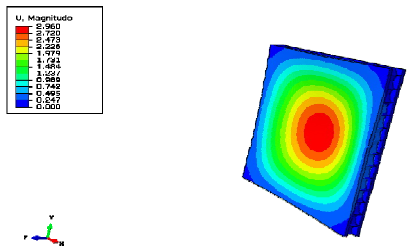

Figure -4 Displacement countour plot in static analysis

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.

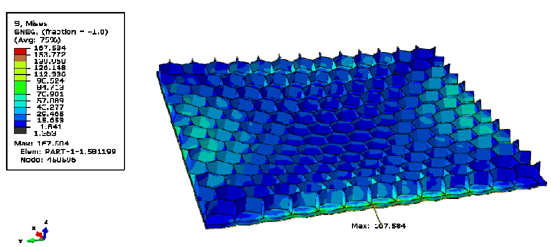

Figure – 5 Von mises stress countour plot in static analysis

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 .

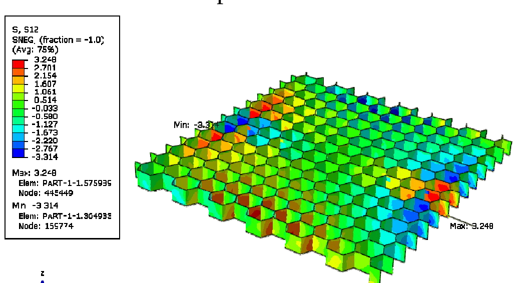

Figure – 6 Shear stress countour plot in static analysis

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 .

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 Map	180	167
Core shear stress in Map	3.5	3.2

Laboratory testing is to be conducted to validated the results the similar design is can be extended to different material condition.

REFERENCES

[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] Iman Dayyani et.all “Experimental Investigations on Mechanical Behavior of Composite Corrugated Core” 2011

[6] ABAQUS user manual version 6.14

[7] Gaetono et.all “theortical design and analysis honeycomb panel sandwich structure in pure bending” Engineering failure analysis 2008