PARAMETRIC STUDY ON HONEY COMB STRUCTURE USING FEA

Pooja. R. Shigihalli[1], Prof. R. D. Deshpande[2]

[1]M-Tech student, Dept. of Civil Engineering, KLS Gogte Institute of Technology, Belagavi, Karantaka, India
[2]Professor, Dept. of Civil Engineering, KLS Gogte Institute of Technology, Belagavi, Karantaka, India

---***---

Abstract - The Honey Comb Sandwich construction is one of the most valued structural engineering innovations developed in the composite industry. It finds its applications in industries like aerospace, aero plane, transportation, rails etc. The behavior of honeycombs subjected to three point bending is investigated using Hypermesh and LS-DYNA. The finite element (FE) results like deflection and critical load are verified by theoretical calculation. The honey comb sandwich CAD model is prepared in CATIA software. The material used for the core and for the faceplates is steel. The analysis is carried out by varying core height and the thickness of faceplates is kept constant. The FEA results obtained and theoretical results were compared.

Key Words: Honey comb sandwich, core material, thickness, Hypermesh, LS- DYNA

1. INTRODUCTION

In mechanical structures stiffness, strength and weight efficiency are important factors, in such cases the sandwich construction is commonly used. These sandwich panels are used in satellites, trains, space craft, aircraft, boats, trucks etc. Core material is selected on the basis of performance low density. For core material hexagonal honeycombs are preferred. The sandwich panel is a composition of face plates bonded on upper and lower sides which are strong and stiff with weak core material. The upper and lower surface faces sheet material of honeycomb sandwich panels can be used as metal or non-metal materials. The basic principle of the sandwich panel is that the core carries the shear stresses and the faceplate carries the bending stresses. Honeycomb sandwich structures exhibit high stiffness and strength to weight ratios.

Honeycomb structures are natural or man-made structures. The geometry of a honeycomb minimizes the amount of material used. The geometry of honeycomb structures can vary. The cells are often columnar and hexagonal in shape.
In the aerospace and transportation industry different types of sandwich core structures are used. Such as foam/solid core type are used in ships and aircrafts, honeycomb types of core are used in aircrafts and satellites, truss core type are used in buildings and bridges and web types of core are manufactured by using a variety of base materials.

A complex shaped core material may be replaced by a simple equivalent volume having elastic orthotropic properties, due to the limitations for hexagonal honeycomb core shapes which are complex and large and are difficult to model and also difficult to manufacture which are computationally expensive.

Material used for the honeycomb core should be such that it will offer advantages such as good mechanical properties, low dielectric properties, low thermal conductivity coefficients, fluid control, good acoustic properties, excellent crushing properties, small cross-sectional areas and large exposed area within the cells.

2. OBJEJECTIVES AND METHODOLOGY

2.1 OBJECTIVE:

Preparing the model of the honeycomb sandwich panel to understand the behavior of the simply supported honeycomb sandwich panel structure under concentrated load. Comparing the deflections, critical loads and stresses of honeycomb sandwich structure to study the effect of different materials and varying the Core height of the honey comb structure.

2.2 METHODOLOGY:

- Literature review related to the project work.
- Collection of material properties and constraints.
- Using CATIA V5 tool 3-D model has been prepared.
- Finite element model has been created by using Hypermesh tool.
- Finite element analysis has been carried out by using LS-Dyna tool.
- Finite element analysis results have been viewed by using LS-prepost tool.

2.3 Modelling:

The 3D CAD model is modeled using Catia V5. Modeling is done part by part and then assembled. Base plate and upper plate are modeled separately and the core part is modeled separately. Then all the parts are assembled and then imported into hyperworks.

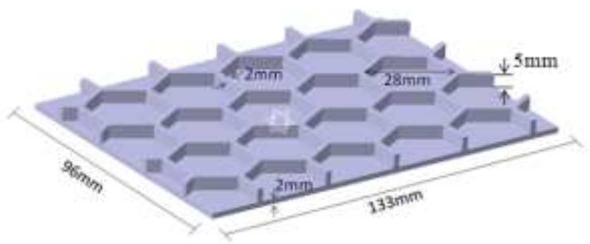

Fig -1: 3D CAD model of honeycomb structure

Figure 1 shows the CAD model of honeycomb structure prepared in the Catia V5 software. The base plate and honeycomb core are showed in the model. The IGS format of the model is imported to hyperworks for meshing.

2.4 Meshing:

The IGS format of the Catia modeled is imported into Hyperworks for meshing. The element size used for meshing varied between 5mm to 7.5mm. Manual method of meshing is used. To save the time model symmetry was utilized and quarter model was meshed and then reflected. The loading and boundary conditions are defined and then the material properties and thickness are assigned to the structure. The analysis is carried out and results are obtained in LS-prepost.

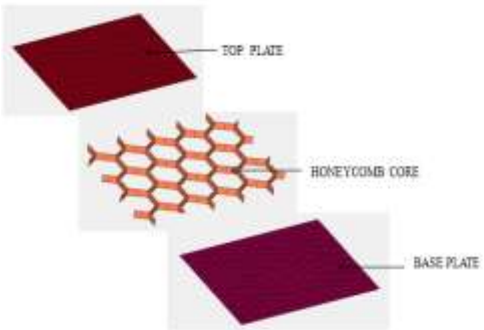

Fig -2: Meshed FE model of honeycomb structure

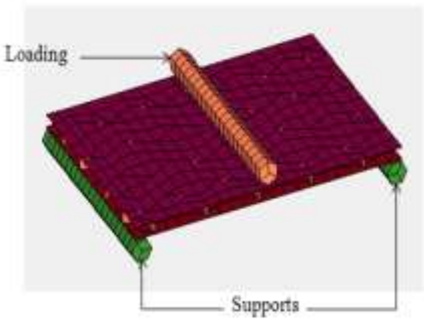

Fig -3: FE model of honeycomb structure with loading and boundary condition

To study the behaviour of honeycomb structure we have modelled five main models of size 133mmx96mm by changing the core height of honeycomb structure and keeping the thickness of faceplates same for all five models. The variation in deflection, critical load and stress are compared by changing the core height and keeping the same thickness of faceplates. The material used for the core and face plates is steel.

Table 1 shows the material properties for material used in honeycomb sandwich structure. Table 2 shows the details of models in terms of change in core height and thickness of core wall and faceplates of the honeycomb structure.

Table -1: Properties of Material

Properties \ Material	**Steel**
Young's Modulus(GPa)	210
Poisson's Ratio	0.29
Yield Strength (GPa)	0.215
Shear Modulus (GPa)	74
Density (Kg/mm^3)	7.85X10^{-6}

Table -2: Model details in terms of thickness of core wall and faceplates of honeycomb structure and core height

Model No	1	2	3	4	5
Core Height	3mm	4mm	5mm	6mm	7mm
Thickness of Top Plates	2mm	2mm	2mm	2mm	2mm
Thickness of Bottom Plates	2mm	2mm	2mm	2mm	2mm

2.5 Time Load Graph:

The maximum load applied on the honeycomb structure during analysis it can be shown by time load graph which represents the load applied on the honeycomb structure every milliseconds (ms).

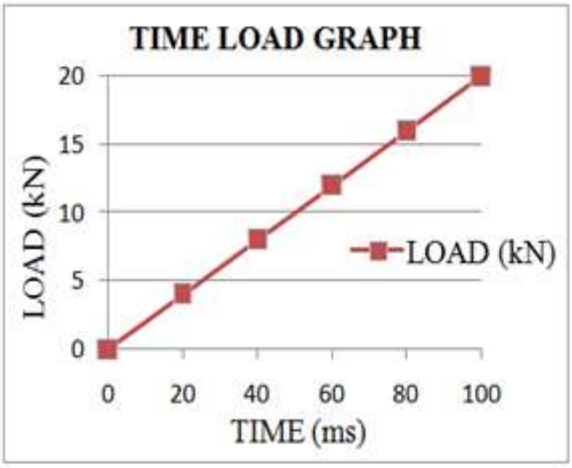

Fig -4: Time Vs Load graph

The above graph shows the maximum load in kN applied on the structure in 100ms. The fig 4 shows the max load 20kN applied on model no. 1, 2, 3, 4 and 5.

2.6 RESULTS:

In all the five models the material used for the top and base faceplates and for the core material is steel. Model 1 in which the core height is 3mm, Model 2 in which the core height is 4mm, Model 3 in which the core height is 5mm, Model 4 in which the core height is 6mm and Model 5 in which the core height is 7mm. The finite element analysis and theoretical results of five models are shown in the below table 3.

Table -3: Results of all the five models

Model no.	Deflection at maximum load		Critical load in the core		Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/m m^2)
	Analytical (mm)	Theoretical (mm)	Analytical (kN)	Theoretical (kN)		
1	1.9	1.88	7.25	6.08	3.3	0.0685
2	1.4	1.33	8.0	7.28	4.1	0.0745
3	1.0	0.98	9.3	8.48	5.2	0.124
4	0.8	0.76	10.4	9.68	6.2	0.235
5	0.7	0.607	11.3	10.86	7.3	0.275

Following Fig 5 to 22 shows stress in the core, stress in the faceplate and deflected shape of the model no. 1, 2, 3, 4, and 5. Fig 23 to 26 shows the max deflection, critical load, failure load and stress variation graphs of model no. 1, 2, 3, 4, and 5.

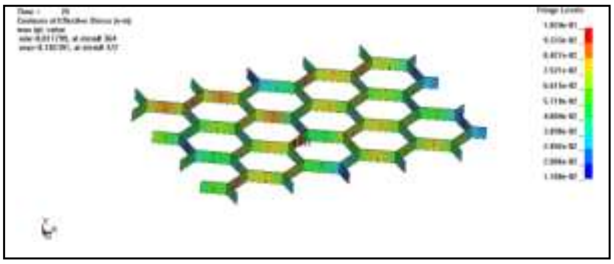

Fig -5: Stress in the core at failure load in model no. 1

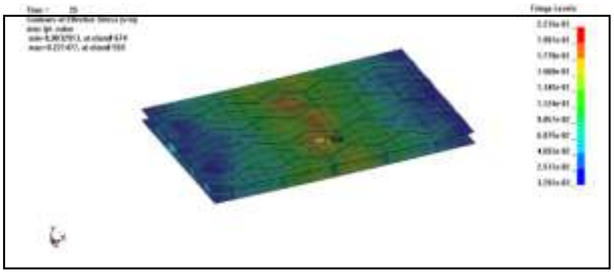

Fig -6: Stress in the plate at failure load in model no. 1

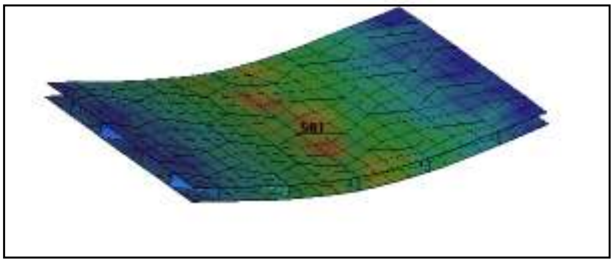

Fig -7: Deflected shape at max load in model no. 1

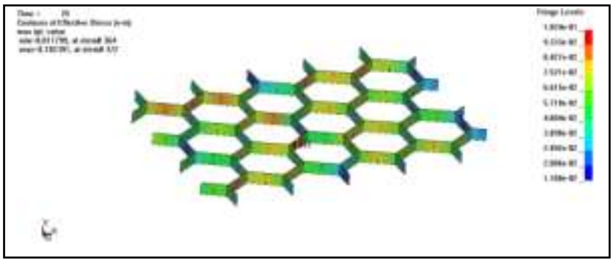

Fig -8: Stress in the core at failure load in model no. 2

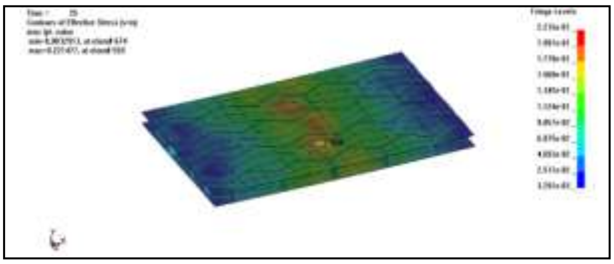

Fig -9: Stress in the plate at failure load in model no. 2

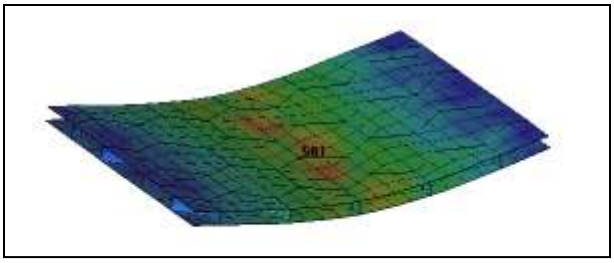

Fig -10: Deflected shape at max load in model no. 2

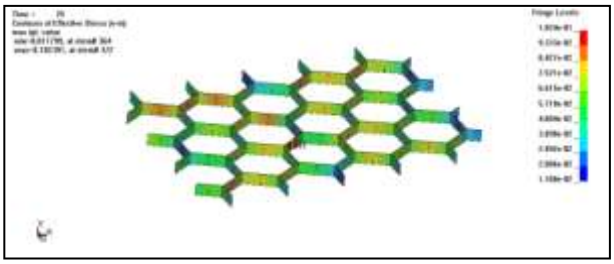

Fig -11: Stress in the core at failure load in model no. 3

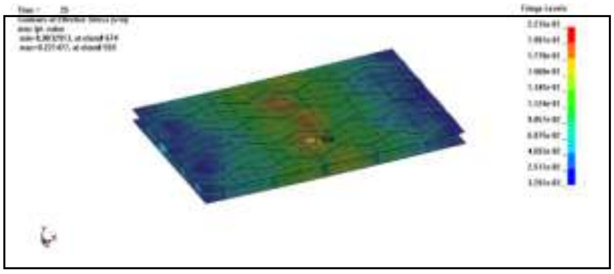

Fig -12: Stress in the plate at failure load in model no. 2

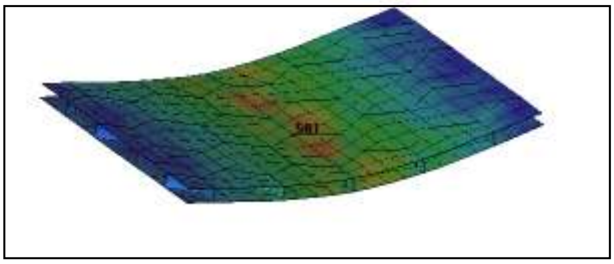

Fig -13: Deflected shape at max load in model no. 2

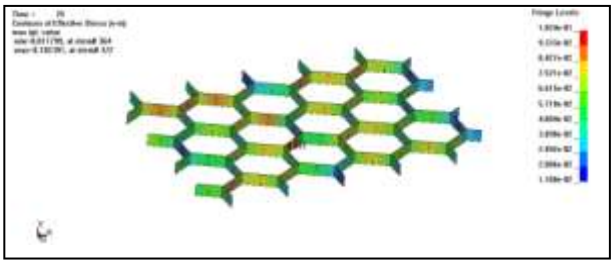

Fig -14: Stress in the core at failure load in model no. 3

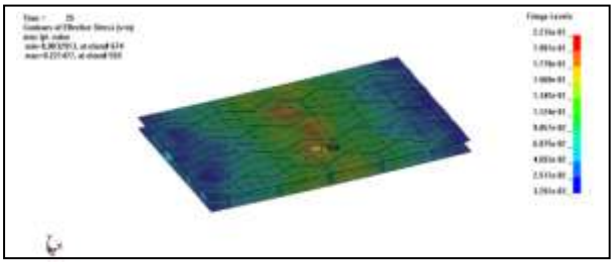

Fig -15: Stress in the plate at failure load in model no. 3

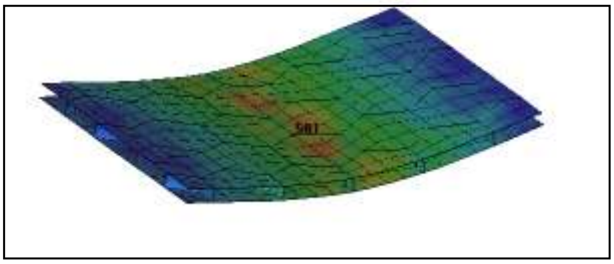

Fig -16: Deflected shape at max load in model no. 3

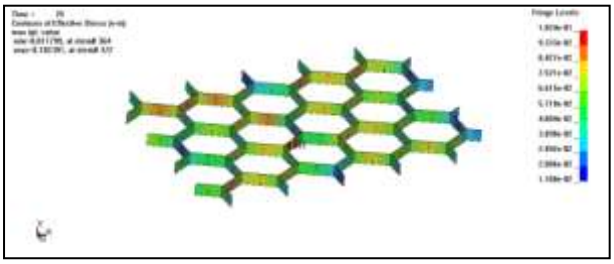

Fig -17: Stress in the core at failure load in model no. 4

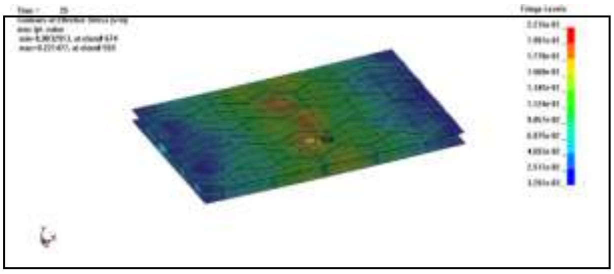

Fig -18: Stress in the plate at failure load in model no. 4

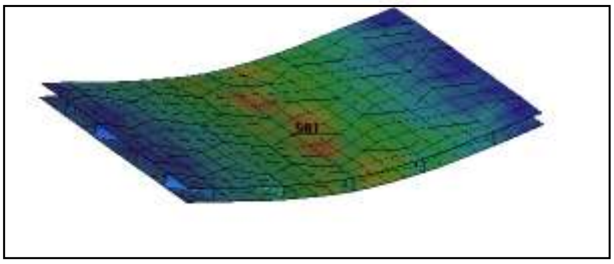

Fig -19: Deflected shape at max load in model no. 4

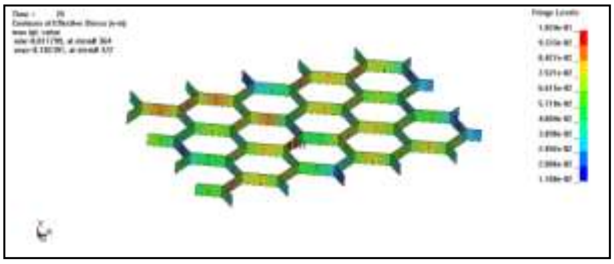

Fig -20: Stress in the core at failure load in model no. 5

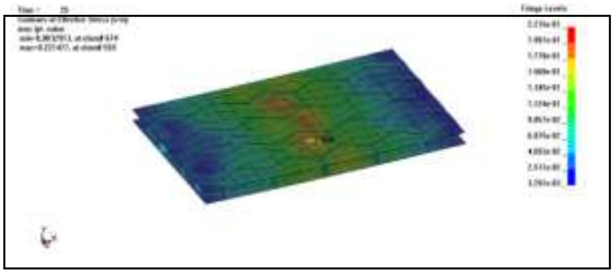

Fig -21: Stress in the plate at failure load in model no. 5

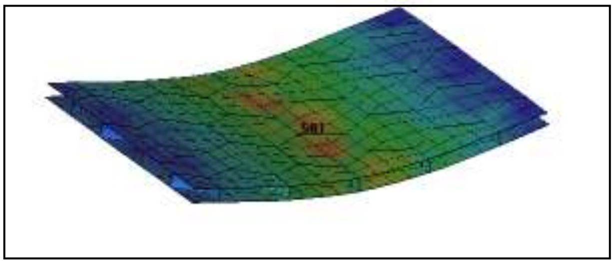

Fig -22: Deflected shape at max load in model no. 1

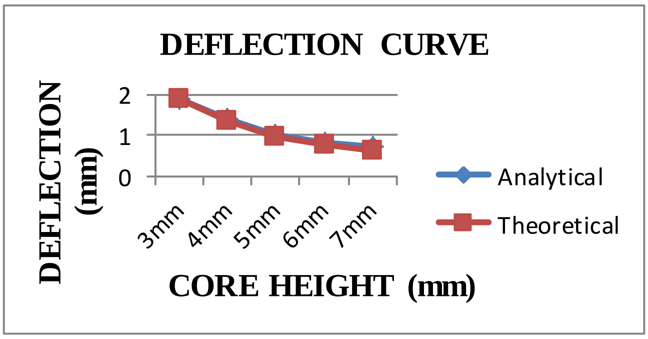

Fig -23: Core height Vs Deflection graph for model no 1, 2, 3, 4, and 5.

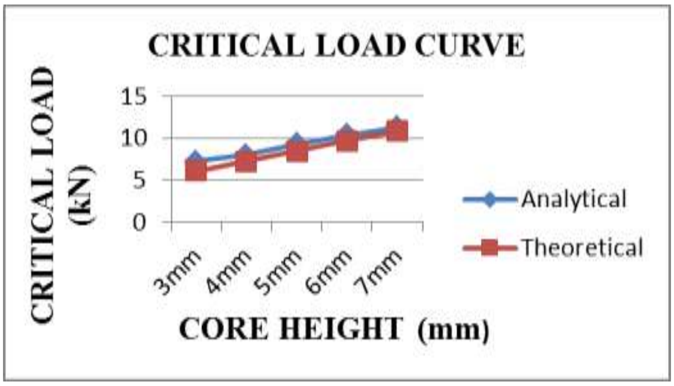

Fig -24: Core height Vs Critical load graph for model no. 1, 2, 3, 4, and 5.

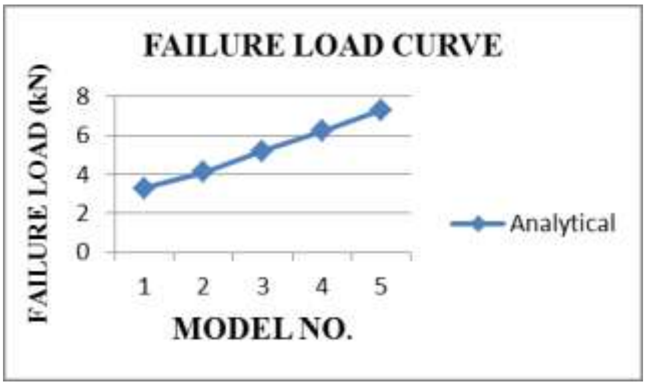

Fig -25: Model no. Vs Failure load graph for model no. 1, 2, 3, 4, and 5.

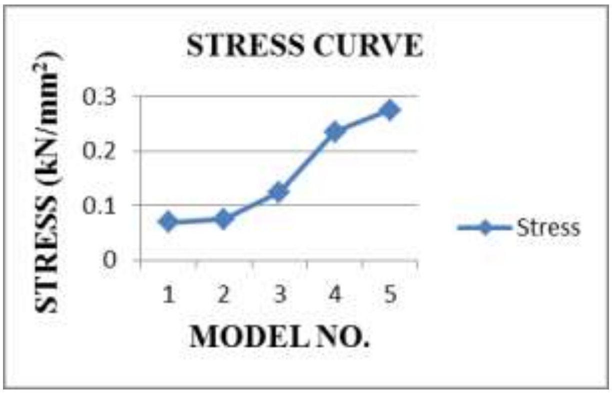

Fig -26: Model no. Vs Stress graph for model no. 1, 2, 3, 4, and 5.

3. CONCLUSIONS

In the present study, behavior of the honeycomb sandwich structure with steel as the material for top and base faceplates and as a core material and different core height as 3mm, 4mm, 5mm, 6mm, and 7mm. and the thickness of the top and base faceplates is 2mm were studied. Different parameters such as deflection, critical load and stress variations were studied. From theoretical and finite element analysis results and comparison of both the results it was concluded that,

- The deflection is decreasing as the core height is increasing and thickness of core wall of honeycomb structure is increasing.
- Load carrying capacity of core is increasing as the core height is increasing.
- The stress is increasing as the core height of honeycomb structure is increasing.
- Deflection is minimum in model 5 and maximum in model 1 and deflection curve trends linearly.
- Critical load is minimum in model 1 and maximum in model 5 and critical load curve is almost linear.

4. SCOPE FOR FURTHER STUDY

The finite element analysis and theoretically calculation for deflection and critical load of the honeycomb sandwich structure is carried out for different core materials and for the different thickness of faceplates and core wall. The thickness of wall faceplates can be kept constant and only the thickness of core wall can be changed to study the behavior of honeycomb sandwich structure. The analysis can be carried out for different diameter of core cells and also by varying the height of the core of honeycomb sandwich structure for the same combination of materials.

REFERENCES

[1] Abderrahmane Bentouhami, Boualem Keskes "Experimental Analysis and Modeling of the Buckling of a Loaded Honeycomb Sandwich Composite" Original scientific article, March 2014.
[2] Banoth Ganesh, B VIjay Kumar, D. Muppala "Design and Structural Analysis of Aircraft Floor Panel" International Journal of Advanced Engineering and Global Technology, Vol-03, Issue-12, December 2015.
[3] Ch. Naresh, A. Gopi Chand, K. Sunil Ratna Kumar, P.S.B.Chowdary "Numerical Investigation into Effect of Cell Shape on the Behavior of Honeycomb Sandwich Panel" International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 12, December 2013.
[4] K.Kantha Rao, K. Jayathirtha Rao A.G.Sarwade, M.Sarath Chandra "Strength Analysis on Honeycomb Sandwich Panels of different Materials" International Journal of Engineering Research and Applications, Vol. 2, Issue 3, May-Jun 2012.
[5] Komal A. Jangavali, D. P. Kamble "Finite Element Analysis and Experimental Evolution of Honeycomb Panel" International Journal of Science and Research, Volume 5 Issue 9, September 2016.
[6] Kranti S. Jadhav, S. R. Sandanshiv "Analysis of different Polygonal Cellular Structures under Impact Loading" International Journal of Science and Research, Volume 5 Issue 7, July 2016.
[7] Mohiyuddin.C.S, Jayalakshmi Raju, Manjunath Hedge "A Study On Behaviour of Sandwich Panels under Impact Loads" International Journal of Civil Engineering, April 2015.
[8] Ms. Poonam Joshilkar, Prof. R. D. Deshpande, Prof. R. B. Kulkarni "Analysis Of Honeycomb Structure" Internation Journal For Research In Applied Science And Engineering Technology, Volume 6 Issue 5, May 2018.
[9] Murat Yavuz Solmaz, Kadir Turan "Experimental and Numerical Analysis of Critical Buckling Load of Honeycomb Sandwich Panels" Journal of Composite Materials, Vol. 44, No. 24/2010.

BIOGRAPHIES

Pooja. R. Shigihalli, P.G student, Dept of Civil Engineering, KLS Gogte Institute Of Technology, Belagavi, Karnataka, India.

Prof R. D. Deshpande, Professor, Dept. of Civil Engineering, KLS Gogte Institute Of Technology, Belagavi, Karantaka, India