iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** V **Month of publication:** May 2018

DOI: http://doi.org/10.22214/ijraset.2018.5153

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

Analysis of Honeycomb Structure

Ms. Poonam Joshilkar[1], Prof. R. D. Deshpande[2], Prof. R. B. Kulkarni[3]
[1]*PG Student,* [2, 3]*Assistant Professor, Dept. of Civil Engineering Gogte Institute of Technology , Belagavi., Visvesvaraya Technological University Belagavi*

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 behaviour 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 core material used in the analysis is copper, steel, aluminium and titanium and the steel material is used for faceplates. The analysis is carried out by varying the thickness of faceplates and core wall of honey comb. The FEA results were obtained and graphs were plotted.

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

I. INTRODUCTION

Honeycomb structure is natural or man-made structure. The geometry of a honeycomb minimizes the amount of material used to minimal weight and minimal cost. The geometry of honeycomb structure can vary. The cells are often columnar and hexagonal in shape. 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. Metal or non-metal materials can be used as the upper and lower surface face sheet material of honeycomb sandwich panels.

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. 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 is 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 as Complex and large hexagonal honeycomb core shapes 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.

II. LITERATURE REVIEW

Xiaojun Yang, et al. carried out the buckling analysis of honeycomb sandwich panel by finite element method with composite skins under dynamic axial compression via ANSYS/LS-DYNA. The conventional methods like honeycomb plate theory; sandwich laminboard theory and equivalent theory were studied. A simplified finite element model on hexagonal structure of a unit cell for sandwich panels was developed utilizing the 3D finite element method. The analysis was carried out and the results were compared with the experimental results. It was found that the yield stress by the finite element analysis in ANSYS/LS-DYNA on the structure is more than that of the experimental test done on the structure by 6.7% . The structural nonlinear was not considered due to which error occurred[1]. Surya Satish Adapa, et al. the deflection variation in honeycomb sandwich structure is carried out. In this paper an analytical analysis, a numerical model and an experimental investigation of 3 point bending test is done. The honeycomb core is of copper with stainless steel face plates. The results obtained from the experiment compared with the analytical results. The deflection for various loads and various core heights of honeycomb structure were compared. The honeycomb core and the face plates are 2mm thick. The faceplates are connected to the honeycomb with spot wielding. It is observed that as the core height increases the deflection in the structure decreases and increase in deflection was found in the lower honeycomb core height[2]. K. Kantha Rao, et al.this paper deals with the design and analysis of aerospace lifting surface with honeycomb core. Honeycomb panel suits for the

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

aerospace lifting surface as the lifting surface bending stresses will be maximum at top and bottom surface, low at the middle surface. The different materials are used as honeycomb sandwich panel. The three point bending test was carried out to understand the bending behavior of honeycomb sandwich panel. The critical load and the deflection of the honeycomb sandwich panel were calculated theoretically. From three point bending test it was observed that titanium alloy has more strength to weight ratio. The crushing test was also carried out by varying the cell thickness and height of the honeycomb core. From crushing test it was observed that wall thickness of the honeycomb core cell is a critical variable affecting the crushing strength of sandwich panel under lateral crushing loads and the height of honeycomb core is not an influential parameter on crushing behavior[3]. Komal A. Jangavali, et al. carried three point bending and impact test is performed on the honeycomb panel. Aluminium material is used as core with FRP face sheet in honeycomb panel. Also ANSYS is used for Finite element analysis of honeycomb panel. The critical load was found theoretically and compared with experimental results and FEA results. From the impact test the deformation was found and compared with FEA results. For the critical load, it was found 5-6% variation in the experimental and FEA results. The deformation found from experimental result was slightly greater than the deformation found in FEA results[4]. Mohiyuddin. C.S, et al. in this paper Experimental and analytical studies on honeycomb sandwich panels were carried out. The behavior of honeycomb sandwich panels under impact loads is investigated. For the analytical study, the modeling was done in HYPER MESH and analysis was done with the help of LS-DYNA. It is observed that with increase in height of fall, the Impact Residual Strength (IRS) will increases and as the thickness of specimen increases energy absorption will be more. As the height of fall increased; the depth of indentation increases[5].

III.OBJECTIVE AND METHODOLOGY

Preparing the model of the honeycomb sandwich panel to understand the behaviour 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 thickness of faceplates and wall of the honeycomb core cells.

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

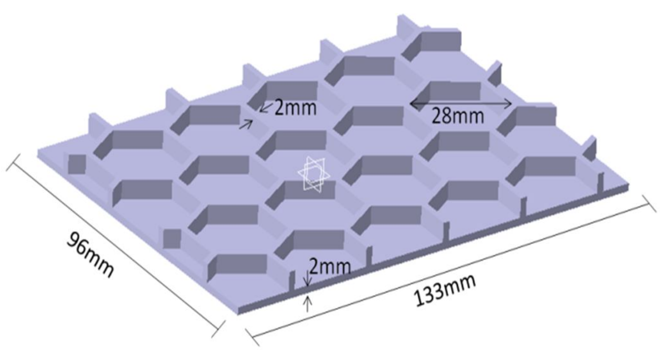


Fig1 3D CAD model of honeycomb structure

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.

A. Meshining

The IGS format of the CATIA modelled 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.

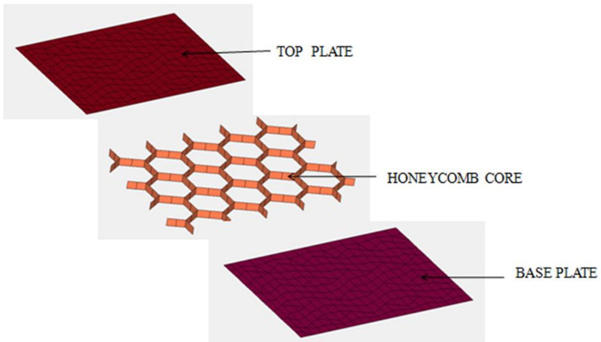


Fig2 Meshed FE model of honeycomb structure

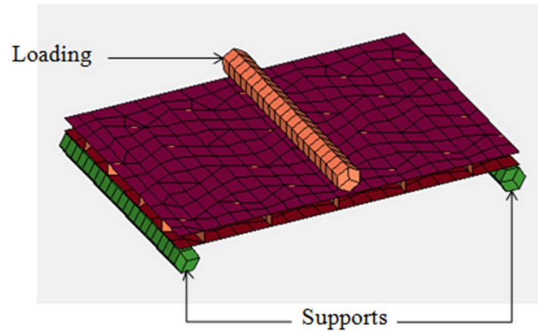


Fig 3 FEM of honeycomb with loading and boundary condition

To study the behaviour of honeycomb structural we have modelled four main models of size 133mmx96mm by changing the core material of honeycomb and keeping the faceplates material same for all four models. The variation in deflection, critical load and stress are compared by changing the wall thickness of core cell as well as thickness of faceplates. Table 2 shows the different types of models with details of the material used. Table 7 shows the material properties for different material used in honeycomb sandwich structure. Table 3 shows the details of model 1 in terms of change in thickness of core wall and faceplates of the honeycomb structure. Table 4 shows the details of model 2 in terms of change in thickness of core wall and faceplates of the honeycomb structure. Table 5 shows the details of model 3 in terms of change in thickness of core wall and faceplates of the honeycomb structure. Table 6 shows the details of model 4 in terms of change in thickness of core wall and faceplates of the honeycomb structure.

Table 1 Model details in terms of materials of core wall and faceplates of honeycomb structure

	Model 1	Model 2	Model 3
Top faceplate material	Steel	Steel	Steel
Core material	Copper	Steel	Aluminium
Base faceplate material	Steel	Steel	Steel

Table 2 Model 1 details in terms of thickness of core wall and faceplates of honeycomb structure

	Model 1		
Model Numbers	1.0	1.1	1.2
thickness of core wall and faceplates	2mm	1.5mm	1mm

Table 3 Model 2 details in terms of thickness of core wall and faceplates of honeycomb structure

	Model 2		
Model Numbers	2.0	2.1	2.2
thickness of core wall and faceplates	2mm	1.5mm	1mm

Table 4 Model 3 details in terms of thickness of core wall and faceplates of honeycomb structure

	Model 3		
Model Numbers	3.0	3.1	3.2
thickness of core wall and faceplates	2mm	1.5mm	1mm

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

Table 5 Properties of Materials

Properties \ Material	Steel	Copper	Aluminium
Young's Modulus(GPa)	210	128	68.3
Poisson's Ratio	0.29	0.36	0.34
Yield Strength (GPa)	0.215	0.100	0.276
Shear Modulus (GPa)	74	45	26
Density(Kg/mm^3)	$7.85X10^{-6}$	$8.96X10^{-6}$	$2.68X10^{-6}$

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 sec.

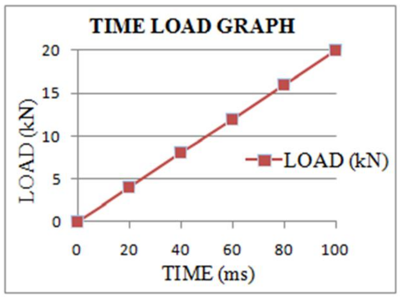


Fig 4 Time Vs Load graph

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 mode 1, 2 and 3.

IV. RESULTS

Model 1 in which top and base faceplates are of steel material and honeycomb core is having copper material and core height 5mm. The model 1 is subdivided into two models because of change in the thickness of faceplates and wall of honeycomb core cells. In the table 2 it is shown that the thickness is varying as 2mm, 1.5mm, and 1mm. Based on the change in thickness the naming is done. The finite element analysis and theoretical results of three models are shown in the below table 6.

Table 6 Results of model 1

Model no.	Analytical Deflection at maximum load (mm)	Analytical Critical load in the core (kN)	Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/mm^2)
1.0	1.7	5	5	0.102
1.1	2.648	3.6	3.6	0.101
1.2	4.2	1.6	1.6	0.108

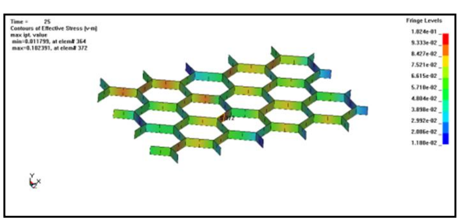


Fig 6 Stress in the core at failure load

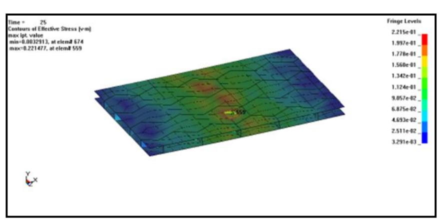


Fig 7 Stress in the plate at failure load

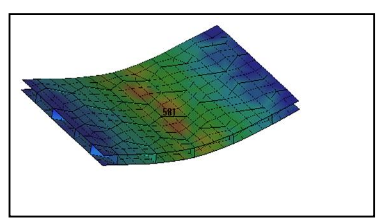


Fig 8 Deflected shape at max load

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

in model no. 1.0 | in model no. 1.0 | in model no. 1.0

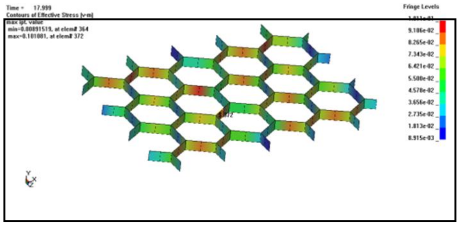


Fig 9 Stress in the core at failure load in model no. 1.1

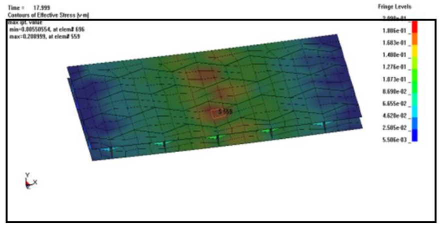


Fig 10 Stress in the plate at failure load in model no. 1.1

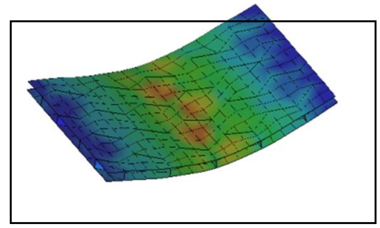


Fig 11 Deflected shape at max load in model no. 1.1

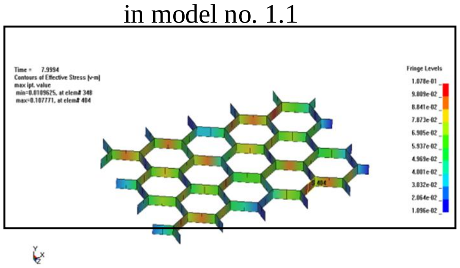


Fig 12 Stress in the core at failure load in model no. 1.2

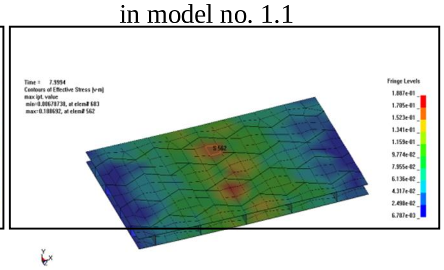


Fig 13 Stress in the plate at failure load in model no. 1.2

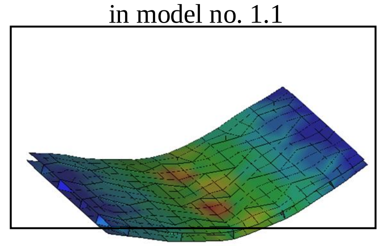


Fig 14 Deflected shape at max load in model no. 1.2

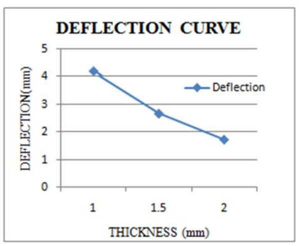


Fig 15 for model 1

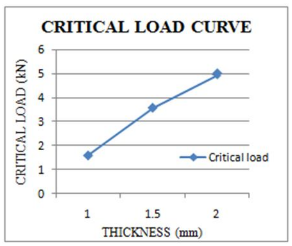


Fig 16 for model 1

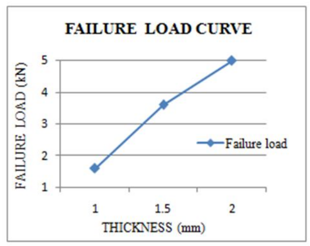


Fig 17 for model 1

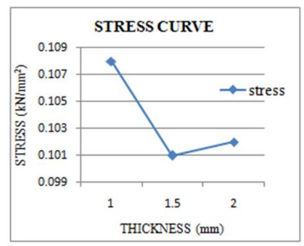


Fig 18 for model 1

Model 2 in which top and base faceplates are of steel material and honeycomb core is also having steel material and core height 5mm. The model 2 is subdivided into two models because of change in the thickness of faceplates and wall of honeycomb core cells. In the table 3 it is shown that the thickness is varying as 2mm, 1.5mm, and 1mm. Based on the change in thickness the naming is done. The finite element analysis and theoretical results of three models are shown in the below table 7.

Table 7 Results of model 2

Model no.	Analytical Deflection at maximum load (mm)	Analytical Critical load in the core (kN)	Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/mm^2)
2.0	1.6	9.2	5.2	0.124
2.1	2.5	6.8	4.0	0.128
2.2	4.03	4.2	2.4	0.145

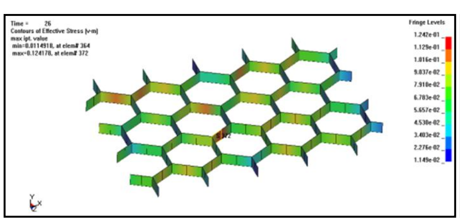


Fig 19 Stress in the core at failure load in model no. 2.0

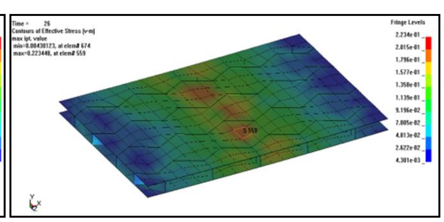


Fig 20 Stress in the plate at failure load in model no. 2.0

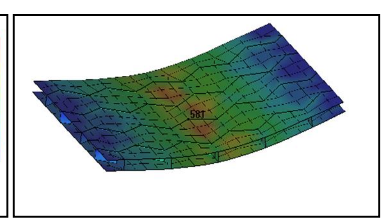


Fig 21 Deflected shape at max load in model no. 2.0

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue V, May 2018- Available at www.ijraset.com

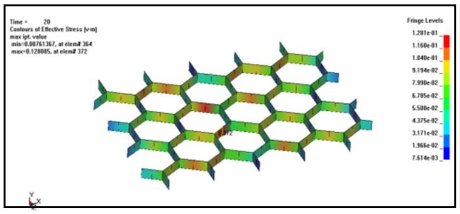


Fig 22 Stress in the core at failure load in model no. 2.1

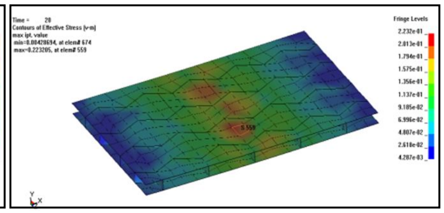


Fig 23 Stress in the plate at failure load in model no. 2.1

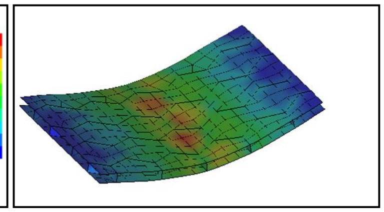


Fig 24 Deflected shape at max load in model no. 2.1

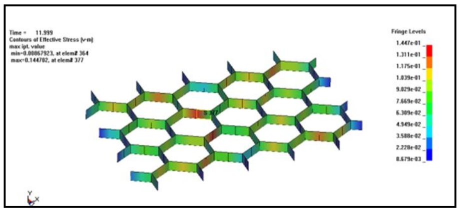


Fig 25 Stress in the core at failure load in model no. 2.2

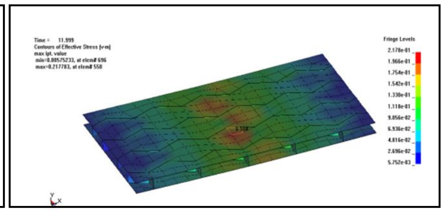


Fig 26 Stress in the plate at failure load in model no. 2.2

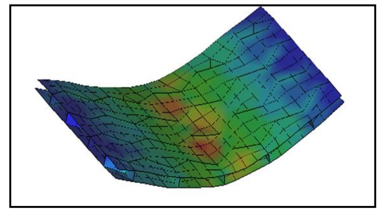


Fig 27 Deflected shape at max load in model no. 2.2

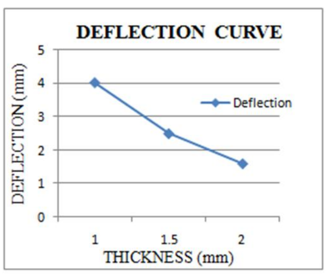


Fig 28 for model 2

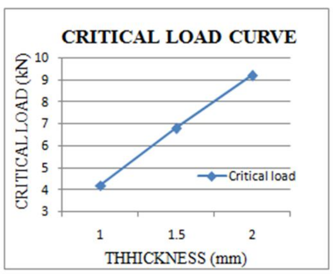


Fig 29 for model 2

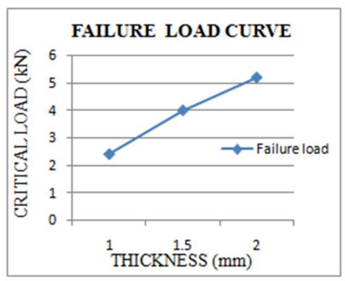


Fig 30 for model 2

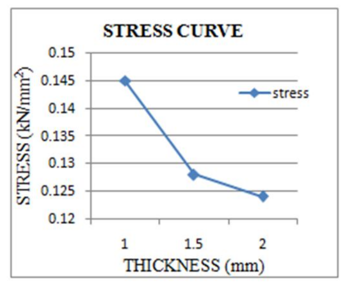


Fig 31 for model 2

Model 3 in which top and base faceplates are of steel material and honeycomb core is having aluminium material and core height 5mm. The model 3 is subdivided into two models because of change in the thickness of faceplates and wall of honeycomb core cells. In the table 4 it is shown that the thickness is varying as 2mm, 1.5mm, and 1mm. Based on the change in thickness the naming is done. The finite element analysis and theoretical results of three models are shown in the below table 8.

Table 8 Results of model 3

Model no.	Analytical Deflection at maximum load (mm)	Analytical Critical load in the core (kN)	Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/mm^2)
3.0	1.88	16.4	4.8	0.0829
3.1	2.88	11.6	3.8	0.0897
3.2	4.725	6.8	2.0	0.109

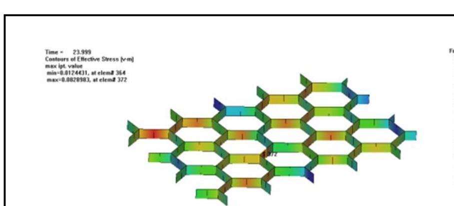


Fig 32 Stress in the core at failure load in model no.3.0

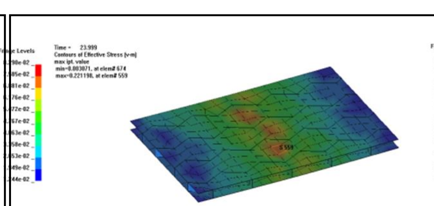


Fig 33 Stress in the plate at failure load in model no.3.0

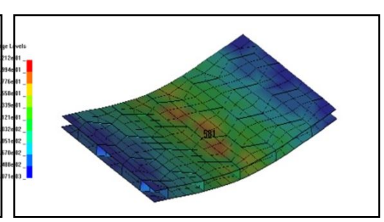


Fig 34 Deflected shape at max load in model no. 3.0

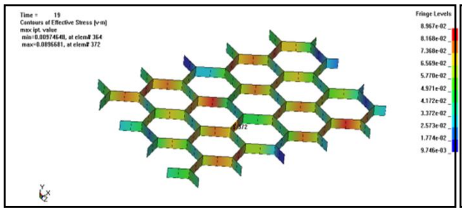


Fig 35 Stress in the core at failure load in model no.3.1

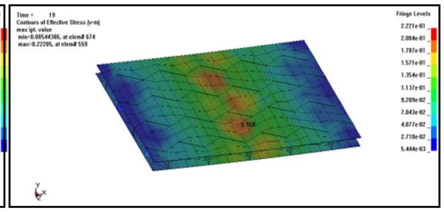


Fig 36 Stress in the plate at failure load in model no.3.1

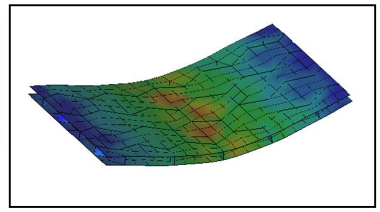


Fig 37 Deflected shape at max load in model no. 3.1

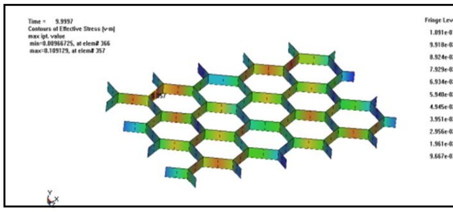


Fig 38 Stress in the core at failure load in model no.3.2

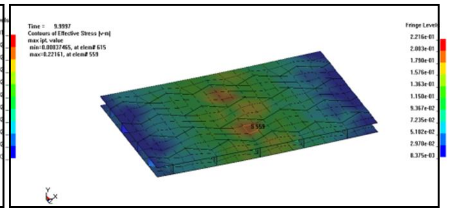


Fig 39 Stress in the plate at failure load in model no.3.2

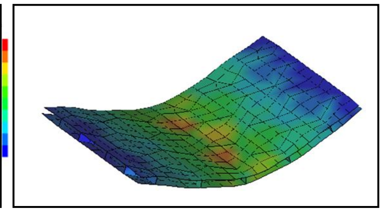


Fig 40 Deflected shape at max load in model no. 3.2

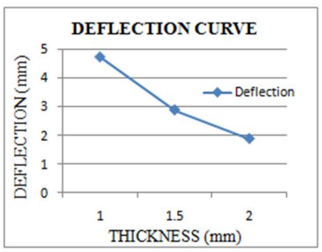


Fig 41 for model 3

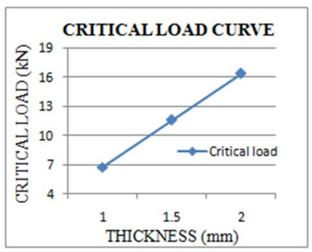


Fig 42 for model 3

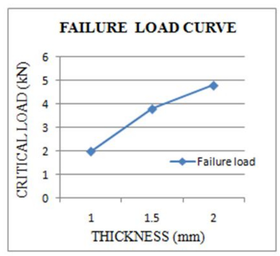


Fig 43 for model 3

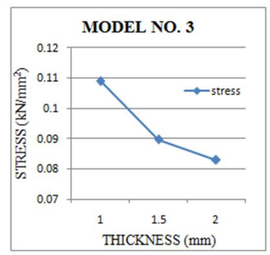


Fig 44 for model 3

To study the effect of different materials used in the honeycomb sandwich structure, comparing the deflection, critical load and stress results of all models and plotting the graph for the models having same thickness of faceplate and honeycomb core wall. Table 10 shows the deflection, critical load and stress results of all models having 2mm thick faceplates and 2mm thick honeycomb core wall.

Results for structure having thickness of faceplate and honeycomb core as 2mm:

Table 10 Results of model no.1.0, 2.0 and 3.0

Model no.	Analytical Deflection at maximum load (mm)	Analytical Critical load in the core (kN)	Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/mm^2)
1.0	1.7	5	5	0.102
2.0	1.6	9.2	5.2	0.124
3.0	1.88	16.4	4.8	0.0829

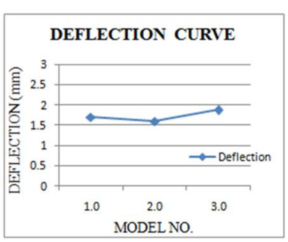


Fig 58

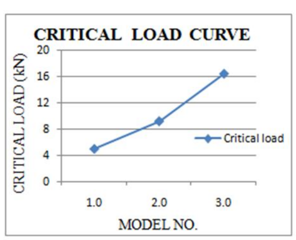


Fig 59

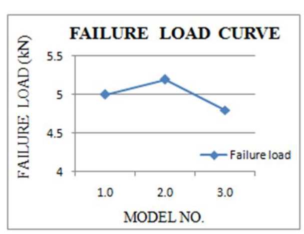


Fig 60

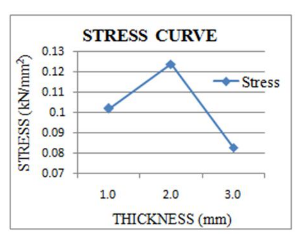


Fig 61

The above fig 58, fig 59, fig 60 and fig 61 shows deflection, critical load, failure load and stress variation for 2mm thick faceplates and core wall model no.1.0, 2.0 and 3.0 Results for structure having thickness of faceplate and honeycomb core as 1.5mm:

Table 11 Results of model no.1.1, 2.1 and 3.1

Model no.	Analytical Deflection at maximum load (mm)	Analytical Critical load in the core (kN)	Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/mm^2)
1.1	2.648	3.6	3.6	0.101
2.1	2.5	6.8	4	0.128
3.1	2.88	11.6	3.8	0.0897

Table 11 shows the deflection, critical load and stress results of all models having 1.5mm thick faceplates and 1.5mm thick honeycomb core wall.

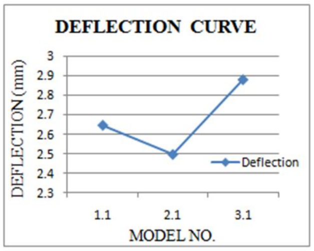


Fig 62

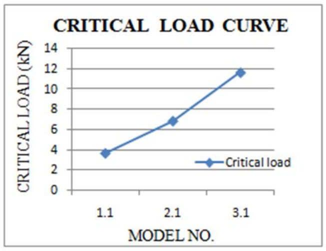


Fig 63

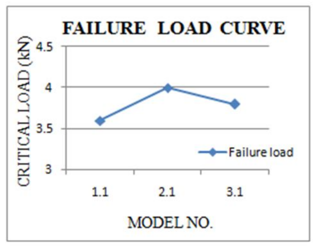


Fig 64

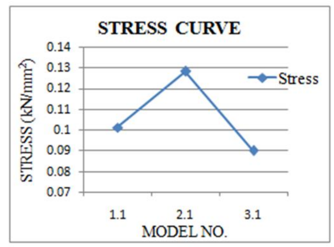


Fig 65

The above fig 62, fig 63, fig 64 and fig 65 shows deflection, critical load, failure load and stress variation for 1.5mm thick faceplates and core wall model no.1.1, 2.1 and 3.1
Results for structure having thickness of faceplate and honeycomb core as 1mm:

Table 12 Results of model no.1.2, 2.2 and 3.2

Model no.	Analytical Deflection at maximum load (mm)	Analytical Critical load in the core (kN)	Analytical Failure load (kN)	Analytical Stress in the core at failure load (kN/mm^2)
1.2	4.2	1.6	1.6	0.108
2.2	4.03	4.2	2.4	0.145
3.2	4.725	6.8	2.0	0.109

Table 12 shows the deflection, critical load and stress results of all models having 1mm thick faceplates and 1mm thick honeycomb core wall.

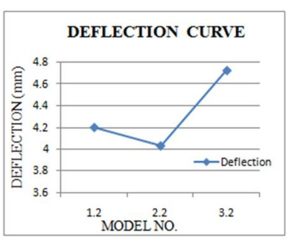


Fig 66

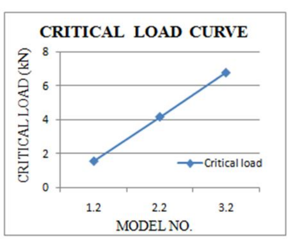


Fig 67

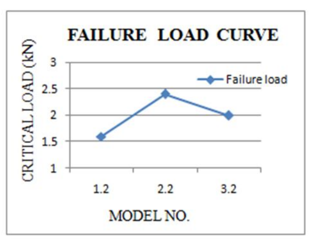


Fig 68

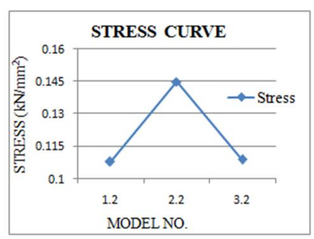


Fig 69

The above fig 66, fig 67, fig 68 and fig 69 shows deflection, critical load, failure load and stress variation for 1.5mm thick faceplates and core wall model no.1.2, 2.2 and 3.2

V. CONCLUSIONS

In the present study, behavior of the honeycomb sandwich structure with steel facing plates and 5mm honeycomb core height under three point bending for different honeycomb core materials such as copper, steel and aluminium; and also for different thicknesses of faceplates and walls of honeycomb core as 2mm, 1.5mm and 1mm were studied. Different parameters such as deflection, critical load and stress were studied. From finite element analysis results it was concluded that,

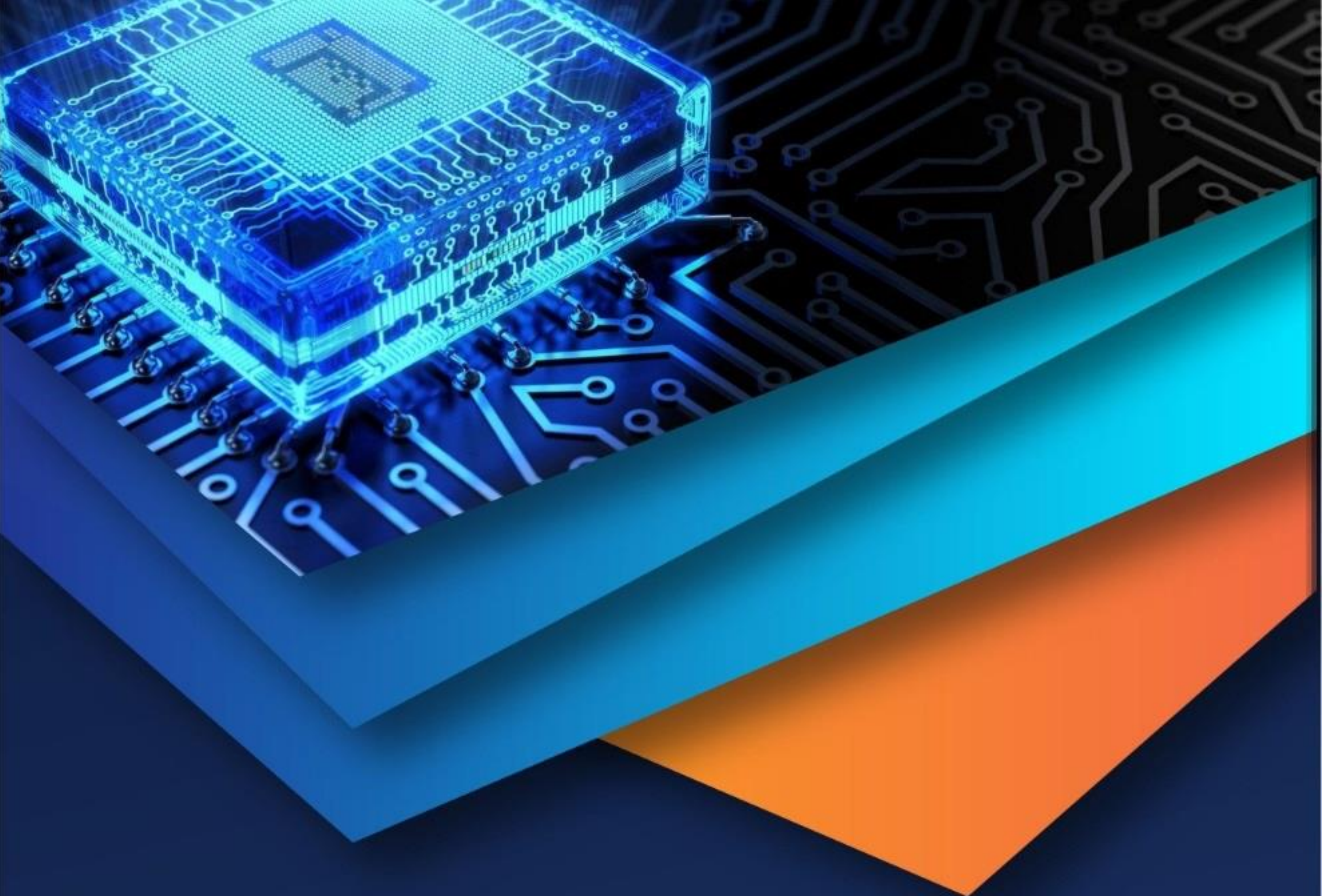
A. The deflection is decreasing as the thickness of the faceplates and thickness of core wall of honeycomb structure is increasing.
B. Load carrying capacity of core is increasing as the thickness of the faceplates and thickness of core wall of honeycomb structure is increasing.
C. The stress is decreasing as the thickness of the faceplates and thickness of core wall of honeycomb structure is increasing.
D. Deflection is minimum in model 1 and maximum in model 3 and deflection curve trends linearly.
E. Critical load is minimum in model 1 and maximum in model 3 and critical load curve is almost linear.
F. Failure load is nearly same for all the models as the faceplates material is steel in all models.
G. The stress variation at failure load among all the models is very close or almost same as the faceplates material is steel in all models.

VI. FUTURE SCOPE

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 wall faceplates can be kept constant and only the thickness of core wall can be changed to study the behaviour 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] 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.
[9] Surya Satish Adapa, Janardhan Jaggavarapu and Vijaykumar Vedangi “Structural analysis of copper honeycomb structures” International journal of advances in engineering & technology, Vol. 8, Issue 6, pp. 950-957, Dec 2015.
[10] S. S. Bhavikatti “Finite Element Analysis” second edition.
[11] Xiaojun Yang, Qingshan Lan and Yuning Zhong “Buckling analysis and experiment of fiber-paper honeycomb sandwich structure composites” Advanced materials research, August 2011.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY