JETIR.ORG ISSN: 2349-5162 | ESTD Year : 2014 | Monthly Issue
JOURNAL OF EMERGING TECHNOLOGIES AND INNOVATIVE RESEARCH (JETIR)
An International Scholarly Open Access, Peer-reviewed, Refereed Journal

Design optimization of honeycomb sandwich panel for maximum energy absorption

Jayprakash Sahu, Mukesh Kumar Singh***
** Research scholar, Department of industrial and production engineering, School of Studies of Engineering & Technology G.G.V. Bilaspur.*
Email: jai.jpknk@gmail.com
*** Professor, Department of industrial and production Engineering, School of Studies of Engineering & Technology G.G.V. Bilaspur*
Email:mukeshtggv@gmail.com

Abstract: The use of honeycomb structures in different machine parts aids in reducing mass of that component without any reduction in strength. The use of honeycomb sandwich structures helps to attain good strength to weight ratio and possess good energy absorption capacity. The current research investigates the application of Finite Element Analysis in determining strength of honeycomb structure and conducting design optimization using Taguchi Design of Experiments. The CAD modelling and FE analysis are conducted in ANSYS 20 simulation package. The optimized design of honeycomb sandwich structure has shown better energy absorption characteristics.

Key Words: Honey comb, Design optimization, Impact analysis

1. INTRODUCTION

The amount of material used in any component is significant in determining strength of that part. The use of honeycomb structures in different machine parts aids in reducing mass of that component without any reduction in strength. The use of honeycomb sandwich structures helps to attain good strength to weight ratio and possess good energy absorption capacity. Multiple factors determine the selection of these honeycomb structures such as manufacturability and stress distribution.

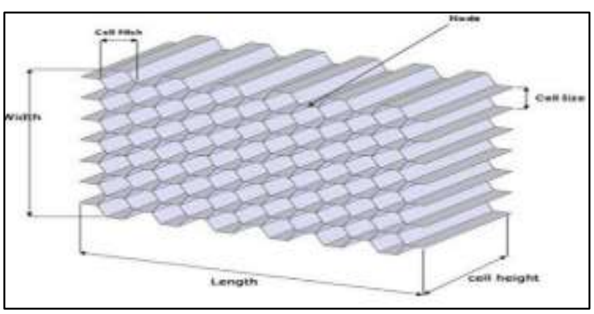

Figure 1: Honeycomb structure geometry

2. LITERATURE REVIEW

Mohmmed et al. [1] has conducted stress analysis of honeycomb structure using experimental methods. The tests conducted are of compression type and bending type. The fractures developed due to external loads are captured in post test images.

Paik et al. [2] has investigated honeycomb sandwich panels made of aluminium material. The tests conducted on sandwich panel are of 3 types i.e. three point bending test, collapse type and crushing type. The structural behaviour of honeycomb sandwich panel under given load is studied.

Tantikom et al. [3] has conducted studies on honeycomb cell structure using intrinsic stress-strain method. This "intrinsic stress–strain response was characterized by the equivalent elastic stiffness and the collapsing deformation behavior" [3].

Chen et al. [4] has investigated the work hardening of sandwich composite material to determine its crushing behaviour [5]. The stress strain response of honeycomb structure is classified as type -I and type -II. These types are based on deformation behaviour.

3. OBJECTIVES

The current research investigates the application of Finite Element Analysis in determining strength of honeycomb structure and conducting design optimization using Taguchi Design of Experiments. The CAD modelling and FE analysis are conducted in ANSYS 20 simulation package.

4. METHODOLOGY

The FE simulation comprises of 3 different stages i.e. pre-processing, solution and posr-processing stage. The methodology flow chart is shown in figure 2 below. The CAD model of honeycomb structure is developed in ANSYS design modeler using sketch and extrude tool. The honeycomb structure dimensions are taken from literature [6].

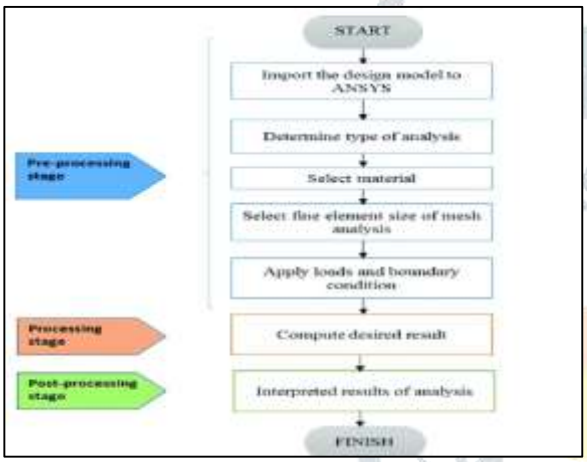

Figure 2: ANSYS FEA flow chart [8]

The single hexagonal structure is patterned to generated multiple hexagonal structure as shown in figure 3 below.

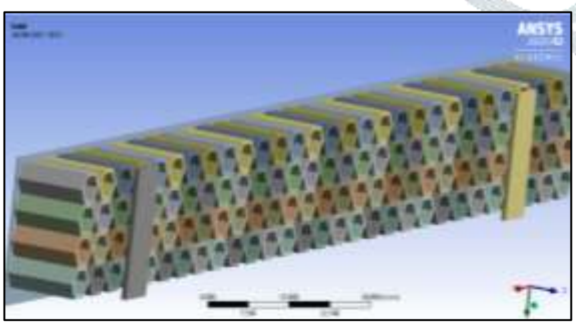

Figure 2: Hexagonal honeycomb structure

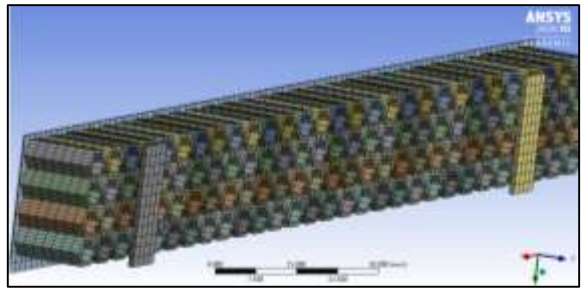

Figure 3: Meshed model honeycomb structure

The CAD model is discretized with hexahedral elements and fine sizing. The number of layers are set to 5 and growth rate set to 1.2. The inflation is set to normal. The number of elements generated is 45487 and number of nodes generated is 68954. The CAD model is then applied with specific loads and boundary conditions as shown in figure 4 below.

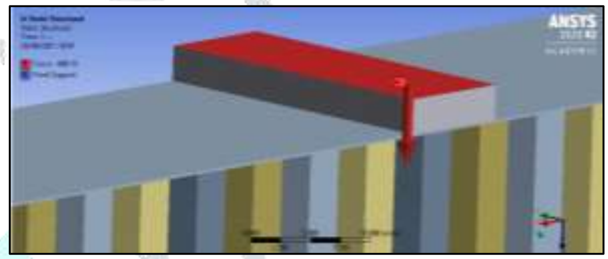

Figure 4: Loads and boundary conditions

The 1st test conducted on honeycomb structure panel is of compressive type. The base of honeycomb panel is applied with fixed support as shown in figure 4 above. The compressive part which is initially placed on top is applied with velocity in downward direction. The applied velocity makes the component to strike the honeycomb structure. The analysis conducted is explicit dynamic type which is implementation of an explicit integration rule together with the use of diagonal or "lumped" element mass matrices. The equations of motion for the body are integrated using the explicit central difference integration rule [7].

$$\dot{\mathbf{u}}^{\left(i+\frac{1}{2}\right)} = \dot{\mathbf{u}}^{\left(i-\frac{1}{2}\right)} + \frac{\Delta t^{(i+1)} + \Delta t^{(i)}}{2}\ddot{\mathbf{u}}^{(i)},$$

$$\mathbf{u}^{(i+1)} = \mathbf{u}^{(i)} + \Delta t^{(i+1)}\dot{\mathbf{u}}^{\left(i+\frac{1}{2}\right)},$$

where $\dot{u}$ is velocity and $\ddot{u}$ is acceleration. The superscript (i) refers to the increment number and $i-\frac{1}{2}$ and $i+\frac{1}{2}$ refers to mid increment values. The matrices are formulated and calculations of each matrix is conducted. The results are generated and interpolated for entire element edge length.

5. RESULTS AND DISCUSSION

The explicit dynamic analysis of honeycomb structure is conducted to determine total deformation and equivalent stresses. The explicit dynamic analysis presents simulation results at different time frames. The total simulation time for the analysis was .004secs.

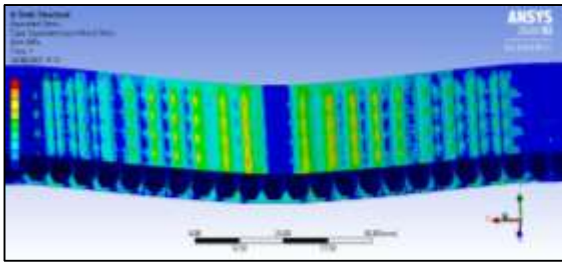

Figure 5: Equivalent stress distribution

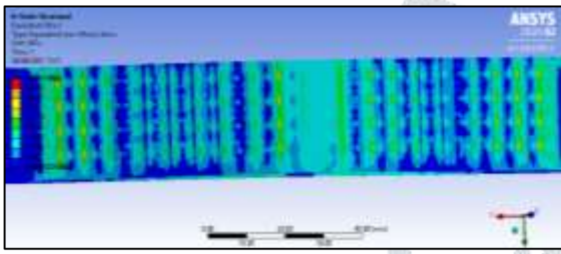

Figure 7: Equivalent stress distribution (underneath)

The equivalent stress plot is obtained for honeycomb sandwich structure as shown in figure 5 and figure 7 above. The plot shows maximum equivalent stress at the zone in immediate contact with load. The stress distribution is higher near the honeycomb edges (as shown by red color) and is of lower magnitude between the panels. The stress distribution underneath (other side) of honeycomb is shown in figure 7. The edges of honeycomb panels shows higher magnitude as represented by yellow and green coloured zones.

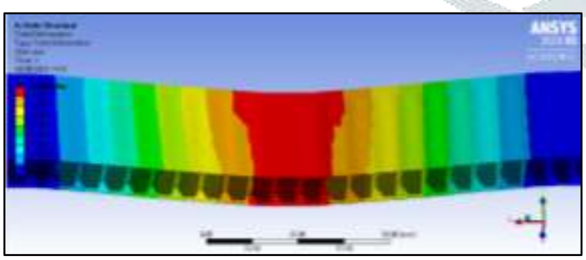

Figure 8: Deformation plot

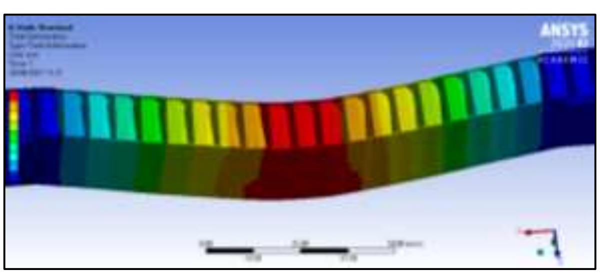

Figure 9: Deformation plot

The deformation plot of sandwich panel is obtained and is shown in figure 8 and figure 9 above. The deformation plot shows maximum deformation at the center region of honeycomb which is in immediate contact with external loading geometry. The maximum deformation region is shown in red color and deformation decreases away from center. The lower deformation regions are represented by green and light blue color.

6. CONCLUSION

The Finite Element simulation is a viable tool to determine energy absorption characteristics of honeycomb structure**.** The critical zones of sandwich structure at which the stress is maximum is identified. The deformation zones are also identified which has maximum chances of crack initiation and propagation. The optimized design of honeycomb sandwich structure has shown better energy absorption characteristics.

REFERENCES

[1]. R. Mohmmed et al., Damage behaviors of glass fabric honeycomb sandwich composite materials under static loading. Int. J. Mech. Prod. Eng. ISSN: 2320-2092

[2]. J.K. Paik et al., The strength characteristics of aluminium honeycomb sandwich panels. ThinWalled Struct. 35(3), 205–231 (1999)

[3]. K. Tantikom et al., In-plane compression response of regularly cell-structured materials. Mater. Trans. 45(2), 509–515 (2004)

[4] D.-H. Chen et al., in Deformation of Honeycomb with Finite Boundary Subjected to Uniaxial Compression. ISSN 2075-4701

[5]. Q. Chen et al., In-plane elastic buckling of hierarchial honeycomb structures. Eur. J. Mech. A/Solid

[6] M. Hussain "Experimental and Computational Studies on Honeycomb sandwich structures under static and fatigue bending loads" Journal of King Saud University – Science 31 (2019) 222–229

[7] ANSYS solver guide

[8] N Fatchurrohman, "Performance of hybrid nano-micro reinforced mg metal matrix composites brake calliper: simulation approach" 4th International Conference on Mechanical Engineering Research (ICMER2017)