Innovative Journal for Opportunities and Development in Science & Technology,
Volume 1,Issue 10, June 2021
Paper Id: – IJ2021V1I10 - 16 A-B,
ISSN 2582-6026

WEIGHT OPTIMIZATION OF LIFT BY USING COMPOSITE SHAPE STRUCTURES WITH HELP OF ANSYS WORKBENCH

[1]ROHAN P. MAHORKAR, [2]V. C. MALI

[1]PG Scholar, Mechanical Engg. Dept. Tssm's PVPIT Institute of Technology, Bavdhan, Pune Maharashtra, India
[2]Assistant Prof., Mechanical Engg. Dept. Tssm's PVPIT Institute of Technology, Bavdhan, Pune Maharashtra, India
[1]rohanmohorkar55@gmail.com

ABSTRACT :- In this Paper, design and analyses of Sandwich structures are investigated for industrial lift platform. Primary goal is to develop a best cross section of structure material model that is a good substitute for the actual Section core. By replacing the actual Section base structure with the other three cross section model in CATIA, during the FEA in ANSYS, we get a advantages can be obtained with 3D modeling and model modification, solution time and hardware assets. To discover the best same or better model among the surmised logical models that can be found in the writing, an examination is made.

Three models are investigated under a similar stacking and the limit conditions. In limited component examinations, ANSYS finite element program is used. Also we manufacture all models as per sample and test it on UTM. The results are compared to find out the best cross section model. After selection of best model we apply this to lift platform and compared this to original lift platform.

Key ward: CATIA, ANSYS, Optimization, Sandwich Panel**.**

I. INTRODUCTION

A structural sandwich consists of two thin face sheets made from stiff and strong relatively dense material such as metal or fiber composite bonded to a thick light weight material called core. This construction has often used in lightweight applications such as aircrafts, marine applications and wind turbine blades. In this paper the structural analysis of corrugated sand which panel with stainless steel faces sheets and mild steel as core is done using ANSYS work bench and compressive strength is compared with experimental value. The model of the curved corrugated core is done in CATIA and the effect of wave length on strength to weight ratio is analyzed. The sandwich panel model in CATIA is efficiently imported into ANSYS work bench structural analysis is done and max stress is observed at top face. A structural sandwich typically consists of two thin face sheets made from stiff and strong relatively dense material such as metal bonded to a thick lightweight material called core. This concept mimics an I beam, but in two dimensions, where the face sheets support bending loads and the core transfers shear force between the faces in a sandwich board under load. Face sheets utilized in structure are chiefly in three structures level, softly profiled constantly. The face sheets of sandwich boards give underlying solidness and ensure the center against harm and enduring. During stacking the face sheets take compressive and ductile loads and center changes shear loads between the appearances and give high twisting firmness. Sandwich structures are utilized in applications requiring high firmness to weight proportions on the grounds that for a given weight, the sandwich structures has a lot higher snapshot of idleness contrasted with strong or I-pillar structures. Sandwich boards with top and base plates just as the center comprised of steel are called steel sandwich boards. The center constructions are of various sorts as indicated by center designs the steel sandwich structures are divided some of them are I-core, O- core with rectangular beams, Vf/V- core with hat or corrugated sheets as a core, web core, round O-core and X-core with two hats as a core steel sandwich structure with curved corrugated core made of mild steel and stainless steel face sheets are considered. For given length and height of the structure increasing the number of curved waves (3 waves to 4 waves) the strength increases effectively. For increase of 4%weight, the strength is increase to 66%

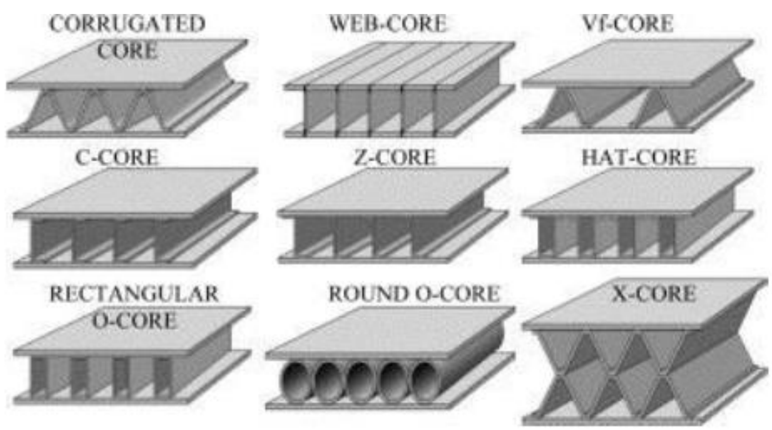

Figure 1: Different sandwich structure with various cores.

1.1 PROBLEM STATEMENT

In the original lift structure having a solid base plate of mild steel structure. Plate having size of 1000mm X 1000mm X 25mm in rectangular shape. This plate is very heavy in weight. Due to self weight of plate we required a higher capacity of motor to lift a structure of industrial lift. Due to this required more electrical energy for this. So we plan to reduce a weight of lift base. There are two methods for weight optimization, first is shape optimization and second is material change or by using composite material. But company required a solution in minimum cost, also required a solution which is easy to manufacturing and raw material is easy to available. So main challenge is not change a material of lift. So we go for shape optimization of base plate.

1.2 OBJECTIVE

Following are the major objectives of Project.
1. The major objective of the proposed research work is to reduce a base plate weight by using a composite structure shapes.
2. To finalization of composite sandwich panel from standard panel, which are easy to manufacturing also.
3. Create CAD models of final sandwich panel structure.
4. Analyze Effect of weight on sandwich panel structure in ANSYS workbench static structural.
5. Manufacture all sandwich panels as per required dimension by using Mild steel only.
6. Take a compression test on UTM Machine for all structures.
7. Compare the experimental result with FEA analysis result.
8. Final a best sandwich panel from all tested panels.
9. Apply this sandwich panel to the base plate of lift structure.
10. Compared a New base with Old base of Lift platform.

II. FINITE ELEMENT ANALYSIS

a. Boundary Condition 4000N for rectangular cross section

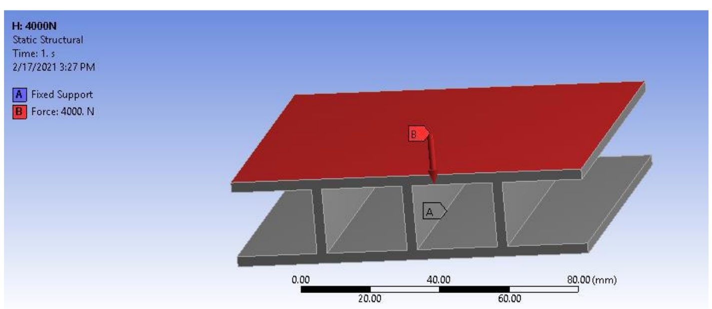

Fig.2. Boundary Condition (4000N) applied on rectangular cross section in ANSYS

Innovative Journal for Opportunities and Development in Science & Technology,
Volume 1,Issue 10, June 2021
Paper Id: – IJ2021V1I10 - 16 A-B,
ISSN 2582-6026

Stress

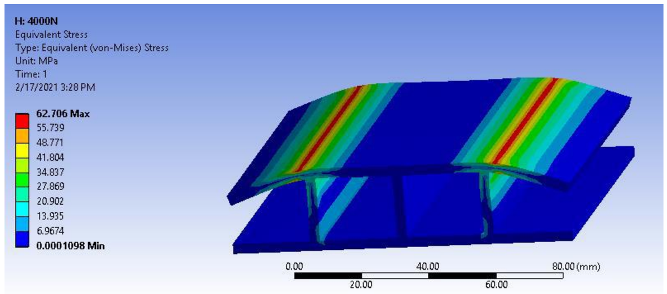

Fig.3. Stress due to applied load of 4000N on rectangular cross section in ANSYS

Deformation

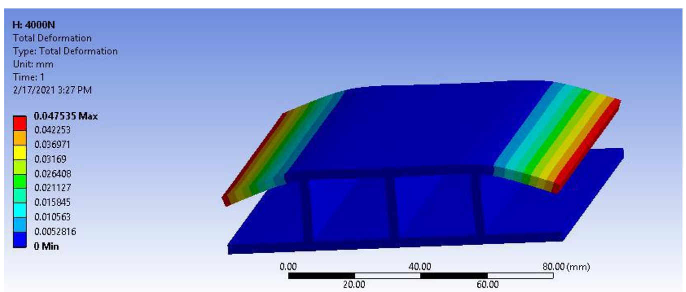

Fig.4. Deformation due to applied load of 4000N on rectangular cross section in ANSYS

b. Boundary Condition 4000N for triangular cross section

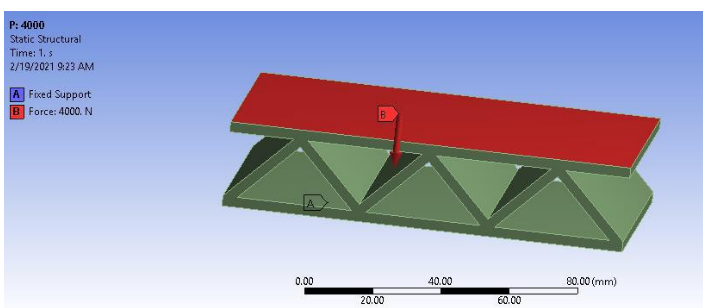

Fig.5. Boundary Condition (4000N) applied on Triangular cross section in ANSYS

Innovative Journal for Opportunities and Development in Science & Technology,
Volume 1,Issue 10, June 2021
Paper Id: – IJ2021V1I10 - 16 A-B,
ISSN 2582-6026

Stress

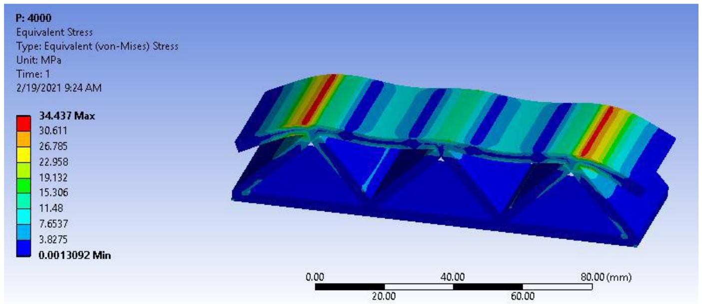

Fig.6. Stress due to applied load of 4000N on Triangular cross section in ANSYS

Deformation

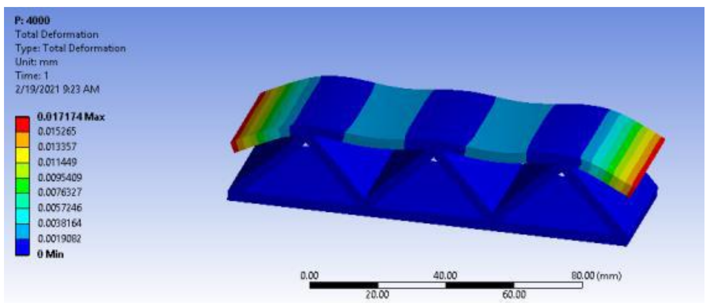

Fig.7. Deformation due to applied load of 4000N on Triangular cross section in ANSYS

C. Boundary Condition 4000N for Circular cross section

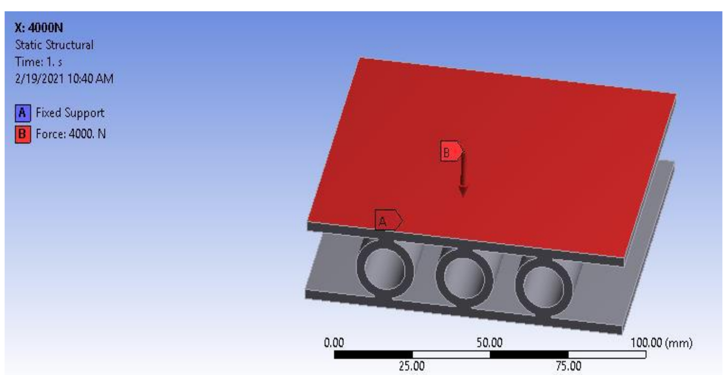

Fig.8. Boundary Condition (4000N) applied on Circular cross section in ANSYS

Innovative Journal for Opportunities and Development in Science & Technology,
Volume 1,Issue 10, June 2021
Paper Id: – IJ2021V1I10 - 16 A-B,
ISSN 2582-6026

Stress

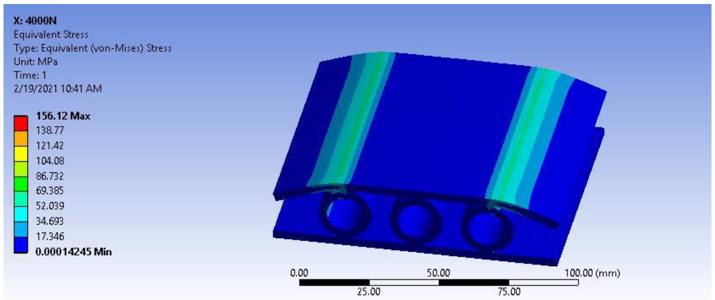

Fig.9. Stress due to applied load of 4000N on Circular cross section in ANSYS

Deformation

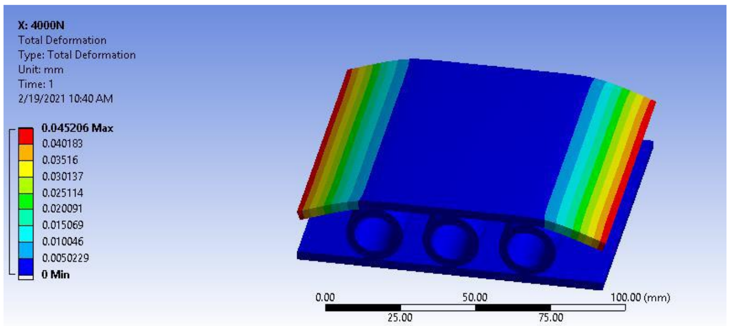

Fig.10. Deformation due to applied load of 4000N on Circular cross section in ANSYS

III. RESULTS & CONCLUSIONS

Table.1. Results of all composite shape by applied force of 4000N

Sr. No.	Shape	Stress (MPa)	Deformation (mm)
1	Rectangular	62.706	0.0475
2	Triangular	34.43	0.0171
3	Circular	156.12	0.0452

The composite structure models in CATIA are efficiently imported into ANSYS workbench structural analysis is done and max stress and total deflection is observed.
By comparing Three-sided composite construction with Rectangular and round composite design it is seen that Triangular composite structure have minimum stresses and also have minimum deflection. So we can final triangular for our application.

REFERENCES

[1] Prof. V. B.Ghagare, Mr. Jayesh Patil, Mr. Rohan Patil, Mr. Rupesh Patil , Mr. Radhay Patil, "Design and Analysis of Corrugated Aluminum Sandwich Structures Using Ansys Workbench," *Volume 5 Issue VI, June 2017 IC Value: 45.98 ISSN: 2321-9653 International Journal for Research in Applied Science & Engineering Technology (IJRASET).*

[2] A.Gopichand, Dr.G.Krishnaiah, B.Mahesh Krishna, Dr.Diwakar Reddy. V.N.L.Sharma, "Design And Analysis Of Corrugated Steel Sandwich Structures Using Ansys Workbench." *International Journal of Engineering Research & Technology (IJERT) Vol. 1 Issue 8, October – 2012.*

[3] Kamlesh G. Ambule,Dr. Kishor P. Kolhe, "FEM and Experimental Analysis of Stainless Steel Sandwich Panels for Weight Reduction"," *IJIRST –International Journal for Innovative Research in Science & Technology| Volume 3 | Issue 02 | July 2016 ISSN (online): 2349-6010.*

[4] A.Gopichand, Dr.G.Krishnaiah, D.Krishnaveni, Dr.Diwakar Reddy.V, "Numerical simulation of steel sandwich plate system (sps) floor," *International Journal of Innovative Research in Science, Engineering and Technology (An ISO 3297: 2007 Certified Organization) Vol. 2, Issue11, November 2013*.

[5] Jukka Säynäjäkangas and Tero Taulavuori, Outokumpu Stainless Oy, Finland, "A review in design and manufacturing of stainless steel sandwich panels," A*utomotive, Stainless Steel World O K T O B E R 2 0 0 4*