Finite Element Evaluation of Composite Sandwich Panel Under Static Four Point Bending Load

M.M. Venugopal[1], S K Maharana[2], K S Badarinarayan[3]

[1]***Assistant Professor***, [2]***Professor***, Department of Aeronautical Engineering, [3]***Professor***, Department of Mechanical Engineering, MVJ College of Engineering, Bangalore, India.
Email: venugopal.aero@gmail.com

Abstract—**The sandwich composites are multilayered materials made by bonding stiff, high strength skin facings to low density core material. The main benefits of using the sandwich concept in structural components are the high stiffness and low weight ratios. These structures can carry in-plane and out-of-plane loads and exhibit good stability under compression, keeping excellent strength to weight and stiffness to weight characteristics. In order to use these materials in different applications, the knowledge of their static behavior is required and a better understanding of the various failure mechanisms under static loading condition is necessary and highly desirable.**

The objective of this study is to develop a modeling approach to predict response of composite sandwich panels under static bending conditions. Different models including 2D and 3D with orthotropic material properties were attempted in advanced finite element (FE) software Ansys. Comparison of FE model predictions with experimental data on sandwich panel bending properties helped in establishing appropriate modeling approach. Analytical solutions were also used to verify the some of the mechanical properties such as bending stress and shear stress with the FEM results.

For this study nomax flex core is used as a core material (thickness 15mm) and carbon fiber reinforced polymer composite (thickness 1.2mm each) is used as face sheet material. The experimental load of sandwich panels was taken and applied in steps through FEM and compared the results with experimental one at all steps.

Keywords - **Composite, sandwich panel, Analytical solution, carbon fiber, Nomax flex core.**

I. INTRODUCTION

Composite sandwich panel have been increasingly used in aerospace industry for various applications such as floor panels, comportment partitions, bulkheads, and even the skin and wings. It is important to design light weight structure for aircraft operations, sandwich panel serves this requirement. The sandwich composites are multilayered materials made by bonding stiff, high strength skin facings to low density core material. The main benefits of using the sandwich concept in structural components are the high stiffness and low weight ratios. These structures can carry in-plane and out-of-plane loads and exhibit good stability under compression, keeping excellent stiffness and strength to weight characteristics. In order to use these materials in different applications, the knowledge of their static behavior is required and a better understanding of the various failure mechanisms under static loading condition is necessary and highly desirable.

Belouettar and Abbadi [1] presented experimental investigation of static behavior of composite honeycomb material made up of aramide fibers and aluminium cores are investigated through four point bending test. The local and global parameters considered to evaluating behavior of sandwich composite, but results are not accurate due to the only experimental study is shown there is no comparison made with any other analytical method. Meyer-Piening [2] suggested that local failures in sandwich structures often occurred because of a lack of awareness of designers of important aspects such as the distribution of displacements through the thickness, axial forces in the face sheets, and the difference between the vertical deflections of upper and lower face sheets. Kemmochi and Uemura [3] investigated the stress distribution in sandwich beams made of three kinds of photo elastic materials under four-point bending. Juli F Davalos and Pizhong qiao [4] studied design modeling and experimental characterization of a FRP honeycomb panel with sinusoidal core geometry in the panel and extending vertically between face laminates. The finite element modeling of test sample is conducted. The result correlates with analytical prediction and experimental values excellent matching is achieved between results.

A.Bezazi and A El Mahi [5] studied analysis of stiffness during static test of sandwich panels and their components. Sandwich panel made of glass fiber and epoxy resin, are subjected to three point bending tests, poly vinyl chloride cores of different densities were investigated in this study, the effect of core densities and its thickness on the behavior is highlighted this paper proves that sandwich structure has better mechanical characteristics compared to its components. Engin M, Reis and Sami.H.Rize kalla [6] presented material characteristics of 3D FRP sandwich panel this paper investigated, flexural, shear, tensile and compressive behavior of sandwich panel face sheet made of FRP and GFRP with foam core the top and bottom face sheets connecter with thick fibers, this paper summarized extensive experimental program discussed many parameters to evaluate sandwich panel behaviors.

II. DESCRIPTION OF PROBLEM

A study of composite sandwich panel of size 800 mm×300 mm×17.4 mm, under uniform static four point bend loading with undamaged models are considered for study. The sandwich panel consists of 8 layers of CFC plate at top and bottom face sheet, each layer has 0.15mm thickness and core is present between top and bottom face sheets which has 15mm thickness. The face plate laid stacking sequence is $[45/-45/0/90]_s$. A sandwich panel that consists of CFC face sheets and Nomax Flex core has been considered for the analysis.

III. FINITE ELEMENT MODELING

The finite element program ANSYS.10 was used to model the sandwich panel, in this analysis both 2D and 3D models are created by using following element type based on convergence test and it is used throughout the study 2D-non linear layered shell element called shell 91 is used for modeling of thick sandwich structures this has ability to take up to 100 layers. The element has six degrees of freedom at each node: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. The element is defined by eight nodes, layer thicknesses, layer material direction angles, and orthotropic material properties. The geometry, nodal locations and coordinate system of the elements are shown on the Fig.1.

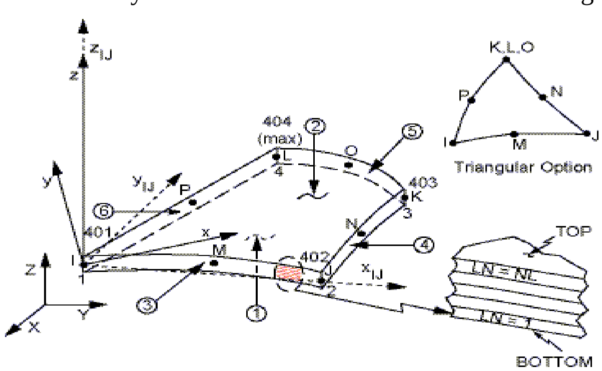

Fig.1 Elemental Geometry of Shell 91
[Source: From ANSYS manual]

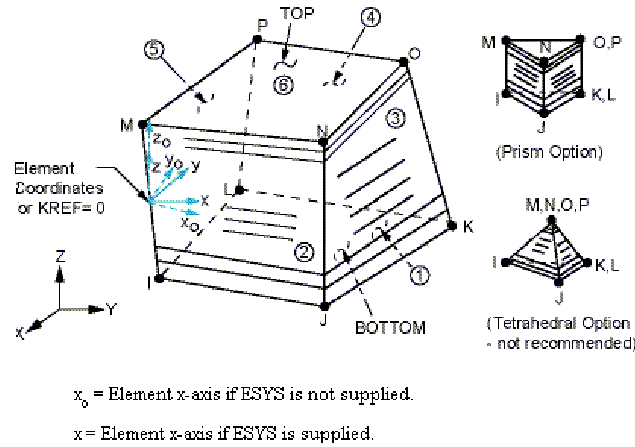

Fig.2 Element geometry of 3D layered solid element
[Source: From ANSYS manual]

And 3D 8 nodded multi layered solid element called as solid46, is used to model for 3D sandwich panel this element takes orthotropic material properties, layered material direction angles and layered thicknesses. The geometry, nodal locations and coordinate system of the elements are shown on the Fig.2, and also the core uses 3D anisotropic structural solid element called solid64. This element has eight nodes having three degree of freedom at each node; translation in x, y, and z directions. The element has stress stiffening and large deflection capabilities and the element has various applications, such as for crystals and composites.

The 2D modeling of sandwich panel based on two dimensions in nature uses shell element to model the all layers of top and bottom face sheets and core. Geometry of the model constructed face as XY direction. The meshed 2D model is shown in the Fig.3

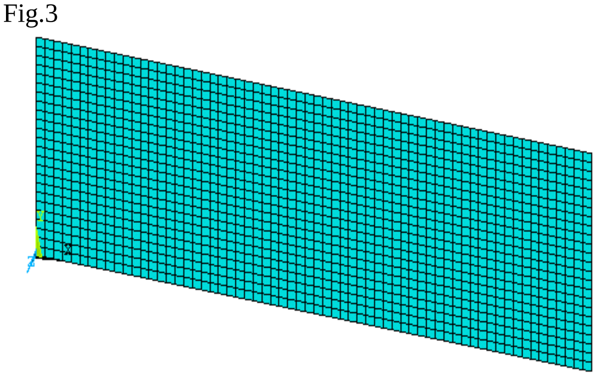

Fig.3 Meshed 2D model
[Generated using ANSYS Software]

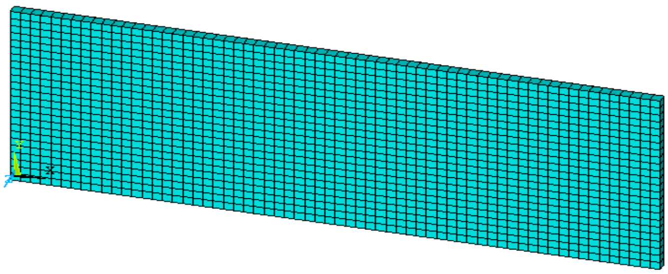

Fig.4 Meshed 3D model
[Generated using ANSYS Software]

The solid model accounts for a three dimensional nature uses solid layered element to model the two face sheets and anisotropic solid element used for model core. This 3D sandwich panel model was able to accommodate both orthotropic and anisotropic material properties. In the course of study, this model was first constructed in the way as shown in figure.3 of 2D model, where the geometry is modeled by creating front face sheet in XY - plane. In addition, the face sheet was extruded in Z - direction with same dimensions of elements. Hence, XY becomes principle plane for material properties.

IV. MATERIAL PROPERTIES OF SANDWICH PANEL

A thick sandwich panel consists of a Nomax flex core and 8 layers of CFC face sheets. The layup sequence for top and bottom face sheets are [+45/-45/0/90/90/0/-45/+45], the thickness of single CFC-UD ply is 0.15mm, Total thickness (t) is 1.2 mm for top and bottom face sheets and core thickness is 15mm and the material properties of composite sandwich panel is given by table.1

Material	CFC-UD	Core
E_{11},MPa	130000	0.2
E_{22},MPa	10000	0.2
E_{33},MPa	10000	255.20
G_{12},MPa	5000	0.2
G_{23},MPa	3500	48.3
G_{13},MPa	5000	55.2
v_{12}	0.32	
v_{23}	0.53	
v_{13}	0.32	

Table.1 Material properties of composite sandwich panel

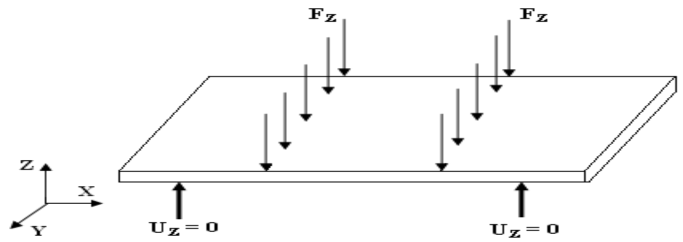

Fig.5 Load applications and boundary condition

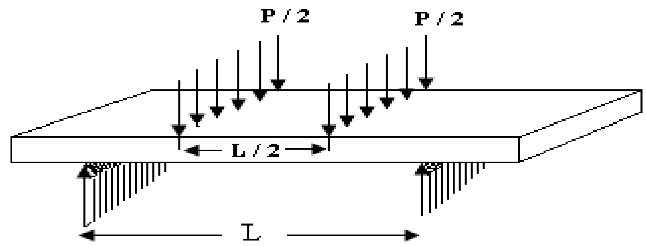

Fig.6 Schematic Representation of 4 point bending test setup

V. BOUNDARY CONDITIONS

The Fig.5 shows the boundary conditions adapted for analyzing sandwich panel. The two supporting points of either end of panel is fixed at translation at Z=0 and static bending load is applied opposite to the supporting point.

The distance between supporting point is 700mm and the distance between support point and loading point is 175mm respectively.

VI. EXPERIMENTAL VALIDATION

The sandwich panels were subjected bending loads, to determine properties. In this study, four point bending scheme was chosen for panel level testing. In a four point bending test, a simply supported bar is loaded with a concentrated load P/2 at two positions of span where P is total applied load. It is also known as quarter point loading. A schematic representation of test, as per ASTM test standard D6272 (2008), and D790 (2008), is shown in Figure.6. This method of testing was used only for panel level testing.

Sandwich panels tested using this method was of the size 800 mm×300 mm×17.4 mm. The panels were manufactured using CFC as a face sheet and Nomax composite as a core material. Panels were simply supported on steel rollers during test. Rollers were supported on beams. Two concentrated loads were applied via a steel beam supported by steel rollers and bearing plates at a certain distance from supports. Load was measured by 500kN load cell attached on a hydraulic ram. Deflections were measured with LVDT at mid span and at loading points.

VII. RESULTS AND DISCUSSION

An application, a full size CFC honeycomb panel of size 800 mm×300 mm×17.4 mm is tested under static four point bending and also analyzed by FE method. The panel bottom surface is simply supported over a span 700mm and subjected to a pitch load. Four point loading condition are applied at mid span to simulate symmetric condition. FE method is conducted at two modeling condition: (1) 2D Sandwich panel model, (2) 3D Sandwich panel model, for each model condition, the deflections were recorded at mid span with corresponding stresses.

In the FE analysis, a sandwich panel with equivalent three layers (top and bottom faces and core) is modeled. For simplicity and verification purposes, the equivalent properties obtained for face laminates and core are used directly in the model. In case of 2D Sandwich panel model the face laminates and core are each modeled in single layer using 8-noded shall element called SHELL91. The displacement contours obtained from ANSYS, FE are shown in Fig.7 (a). In case of 3D Sandwich panel model the face sheet and core are modeled in three different layers using 8-noded solid elements called SOLID 46 for two face sheets and SOLID 64 for Core. The displacement contours obtained from ANSYS, FE are shown in Fig.7 (b) as shown in Table, the 2D and 3D Sandwich panel FE predictions based on equivalent material properties compare favorably with experimental data. For different static loading condition the maximum difference for deflection of 2D Sandwich panel is 18.11% and 3D Sandwich panel is 11.27%.

The bending stresses and shear stresses are predicted from theoretical relations and FEA. The comparison of bending stress and shear stresses from 2D and 3D FEM model with theoretical prediction is shown in the Fig.9 and Fig.10 and by observing this the 2D and 3D sandwich panel models closely following the theoretical trend.

Load, *kN*	**Deflection**	***mm***	
	2D Panel	**3D Panel**	**Exp.**
2.0	3.9	4.3	4.6
2.5	4.9	5.3	5.9
3.0	5.8	6.4	6.9
3.5	6.8	7.4	8.0
4.0	7.8	8.5	9.1
4.5	8.8	9.5	10.3
5.0	9.8	10.6	11.5
5.5	10.7	11.6	12.7
6.0	11.7	12.6	13.8
6.5	12.7	13.6	15.2
7.0	13.6	14.8	16.2
7.5	14.6	15.8	17.3
8.0	15.6	16.9	18.4
8.5	16.5	17.9	19.7

Table.2 The Deflection Value of Predicted 2D and 3D Panel with Experimental Panel result

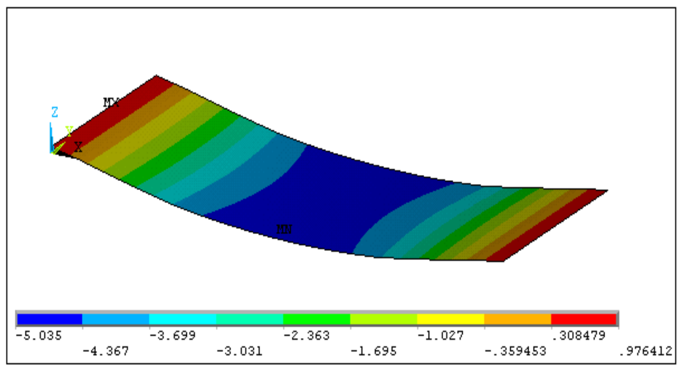

*Fig.7 (a) Deflection Contours of 2D Sandwich Panel

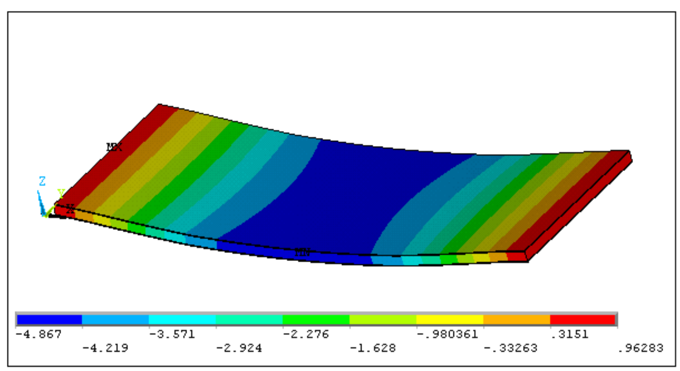

*Fig.7 (b) Deflection Contours of 3D Sandwich Panel
[*Generated from ANSYS Software]

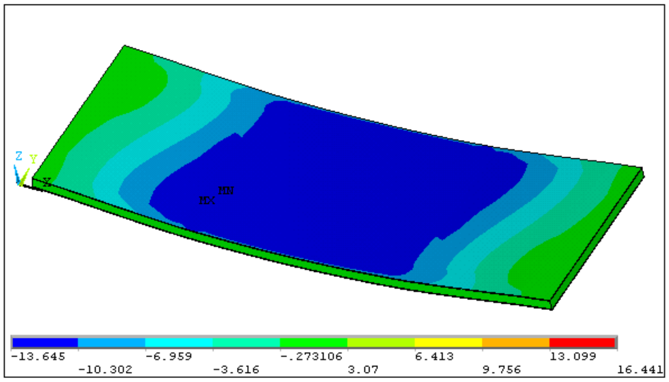

Fig.8 (a) Bending Stress Contour by 3D Sandwich Panel

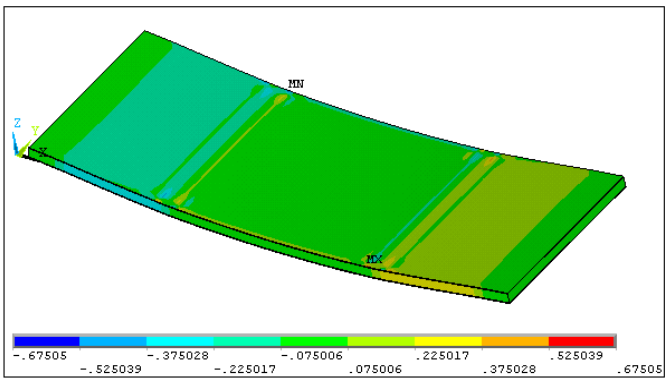

Fig.8 (b) Shear stress contour by 3D sandwich panel

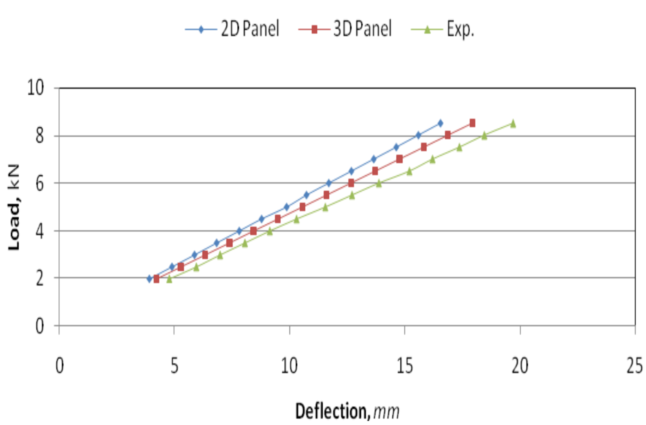

Fig.9 Comparison of 2D, 3D models with Tested Sandwich panel model [*Exp. = Experimental]

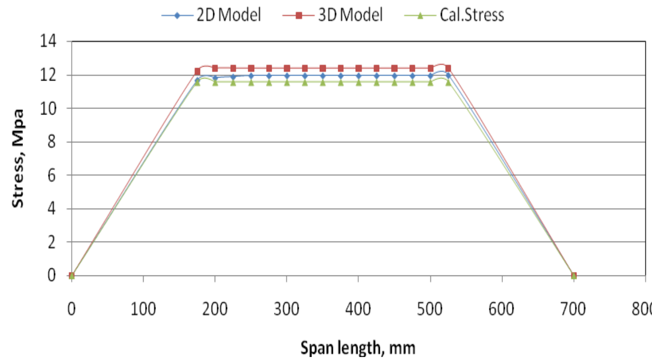

Fig.10 Comparison of Bending Stress Along Span [*Cal.stress = Calculated stresses]

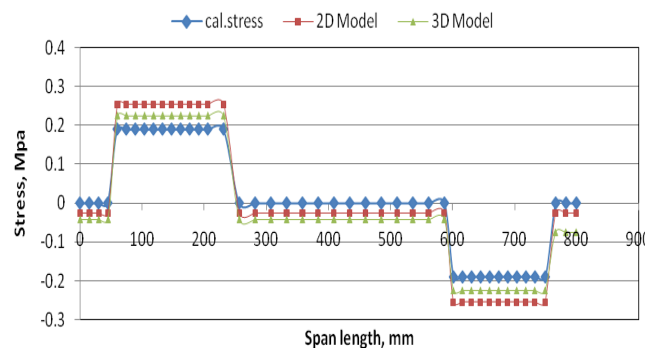

Fig.11 Comparison of shear stress along the span [*Cal.stress = Calculated stresses]

VIII. CONCLUSIONS

This paper presents a combined Finite element modeling and experimental analysis of Carbon fiber composite sandwich panel. The core consists of Nomax honey comb structure presented in between top and bottom face laminate. The emphasis of this study is on evaluation of deflection, bending stress and shear stress response under static four point bending condition. The 2D and 3D FE model predictions correlate with experimental results of Sandwich specimen. The predicted deflection in this study is success fully matching the response of CFC sandwich panels. The 3D FE model under static loading condition is closely matching with experimental deflection. It is found that the maximum percentage error is only 11.27%. Thus, in this study can be used with confidence in design analysis of the CFC sandwich panels.

REFERENCES

[1] Belouettar and Abbadi. "*Experimental investigation of static and fatigue behavior of composites honeycomb materials using four point bending tests*". Composite Science Technology 2003; 70:2556–64

[2] Meyer-Piening H-R. "*Remarks on higher order sandwich stress and deflection analysis*". In: Olsson K-A, Reichard RP, editors. Proceedings of the first international Conf on Sandwich Constructions 1989 P107–27

[3] Kemmochi and Uemura "*The stress distribution in sandwich beams made of three kinds of photo elastic materials under four-point bending*". Journal of Mechanics, 2356-23; Jun 2000.

[4] Juli F Davalos, and Pizhong qiao "*modeling and characterization of fiber reinforced plastic honeycomb sandwich panel for highway bridge applications*". Mechanics of materials 1998; 5642-13 *material in three point bending Part 1. Static tests*" Journal of composites 2001; 4281-14

[5] A Bezazi, and A El Mahi "*Experimental analysis of behavior and damage of sandwich composite*

[6] Engin M, Reis and Sami.H.Rize kalla "*Material characteristics of 3D FRP sandwich panel*". 3rd edition oxford press

[7] Daniel IM, Abot JL. "*Fabrication, testing and analysis of composite sandwich beams*". Composite Science Technology 2000; 60:2455–63

[8] ASTM Standard procedure, D6272 – 02 (Reapproved 2008)

[9] ASTM Standard procedure, D790 – 10 (Reapproved 2005)

[10] Allen, H.G., 1969, "*Analysis and Design of Structural Sandwich Panels*", Pergamon Press, New York

[11] Chung D.D.L., 1994, "*Carbon Fiber Composites*", Butterworth-Heinemann, Massachusetts