Jurnal
Teknologi

Full Paper

Determination of Delamination Size in Honeycomb Sandwich Panel Using Finite Element Method

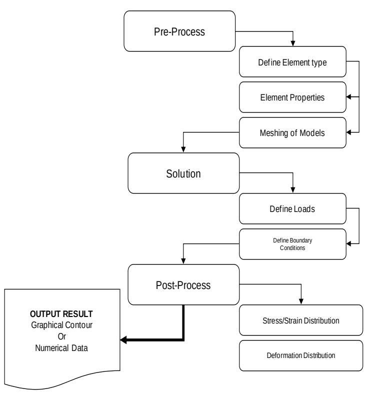
Ramzyzan Ramly*, Wahyu Kuntjoro, Wirachman Wisnoe, Rizal Effendy Mohd Nasir, Aman Mohd Ihsan Mamat, Firdaus Mohamad

Flight Technology and Testing Centre, FTTC, Faculty of Mechanical Engineering, UiTM Shah Alam, Malaysia

Article history
Received
13 February 2015
Received in revised form
30 April 2015
Accepted
09 June 2015

*Corresponding author
ramzyzan@salam.uitm.edu.my

Graphical abstract



Abstract

This paper describes the determination of a relative delamination size of the skin to the honeycomb core of the honeycomb sandwich panel using the Finite Element Method approach. In the analysis, the honeycomb sandwich panel was modelled in the actual dimension using CATIA. The delamination of two different sizes (10 mm diameter and 30 mm diameter) were modelled to simulate the delamination cases. Using Nastran/Patran, the models underwent a three-point-bending test in order to simulate a result. The results were compared between the case of no delamination, 10 mm delamination, and 30 mm delamination. From the simulation, there was a significant difference of displacement of the skin (facing) between the 10 mm diameter delamination and the 30 mm diameter delamination.

Keywords: Determination of delamination, size, honeycomb sandwich panel, finite element method

1.0 INTRODUCTION

Composite materials have long replaced the metallic structure in the rudder skin box of modern civil aviation aircraft [1, 2]. These composite materials are usually made of sandwich panels i.e. honeycomb core and carbon fibre skin [2, 3]. One of the problems in all composite structures is delamination occurring inside the sandwich panel itself where it is very hard to monitor or be seen with the naked eyes [3-5]. In this paper, the delamination sizes would be probed to see if there was an indication it the delamination was relatively small or relatively bigger. Some other researchers have used Finite Element Analysis to analyse the delamination inside the sandwich panel for example Davies et. al. who did analysis in predicting the delamination in aerospace structures using numerical approach [6] and also Chen and Ozaki who conducted the study on the stress concentration due to the defects in the honeycomb sandwich composites [7]. The prediction of delamination using Finite Element Analysis approach was done by Han et. al. [8] while reverse method of Finite Element was employed for displacement and stress monitoring of sandwich structure was conducted by Cerracchio P. et al [9].

2.0 OBJECTIVES

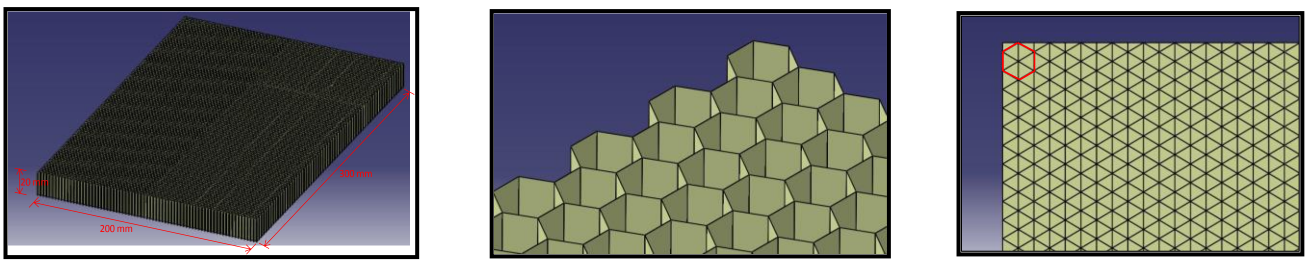
The first objective of this paper is to compare the stress load in the sandwich structure with delamination and without delamination. The second objective of the paper is to determine either a relatively small or relatively large delamination by comparing the displacement between the 10 mm delamination to the 30 mm delamination using Finite Element Analysis.

3.0 METHODOLOGY

A 3D full-sized honeycomb core with carbon fibre skin sandwich panel was modelled using CATIA software. The panel was about 300 mm length and 200 mm in width with 20 mm thick. Three models were modelled to represent three cases namely no delamination, 10 mm diameter delamination and 30 mm diameter delamination. The delamination was made at the centre of the panel on one side below the skin. The delamination was made by having a circular void in the solid model honeycomb core so that there will be an empty area created to represent a delamination. The following Figure 1 shows the 3D solid model of the honeycomb sandwich panel.

About six cells of the skin would cover the top of the honeycomb core cell. The six cells which had six corners, were used to ensure that all six corners of the hexagonal honeycomb core cells would be shared by the corners of the triangular cells in the skin. This factor was very important when finite element analysis was carried out later on.

The following Figure 2 shows the process flow of finite element analysis. The finite element pre-processing software used was MSC Patran, which was available at the Faculty of Mechanical Engineering, UiTM. The solid models developed in the CATIA were saved as ".igs" extension file and were directly imported in the MSC Patran environment.



(a) (b) (c)

Figure 1 The solid model of the honeycomb core (a), detailed structures of the honeycomb (b), the skin model of the sandwich panel

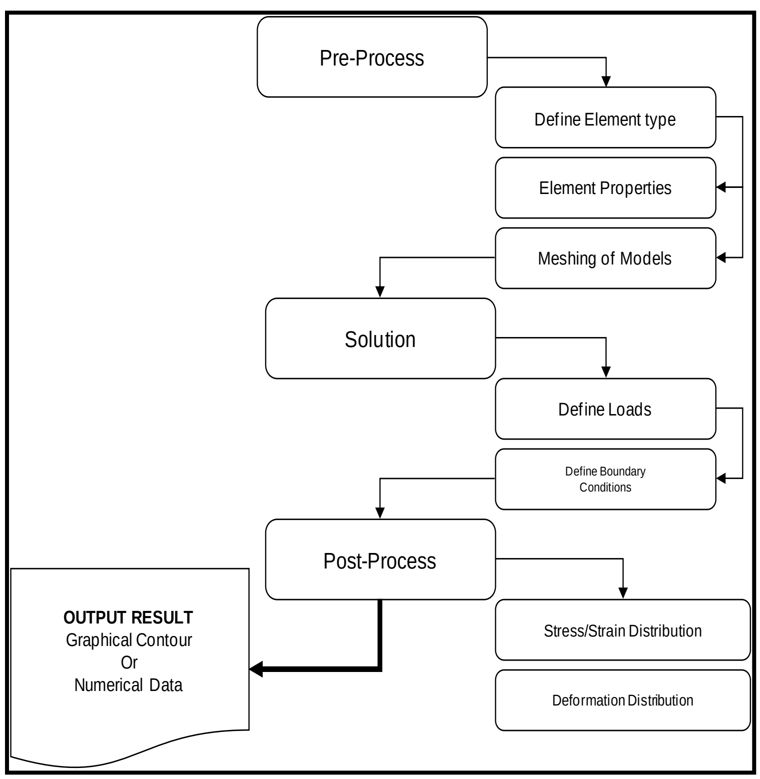


Figure 2 Flow process of finite element analysis

The sandwich panel structures were in the form of thin sheets, the honeycomb cell wall was a thin material, and therefore, it was treated as a 2D isotropic shell. The following Fig. 3 shows the full model of the finite element model.

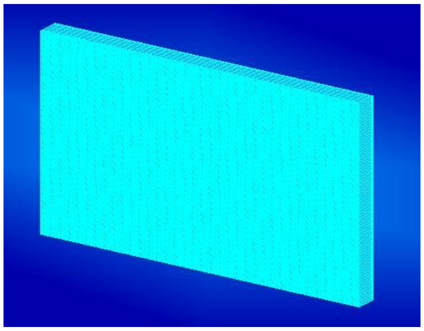
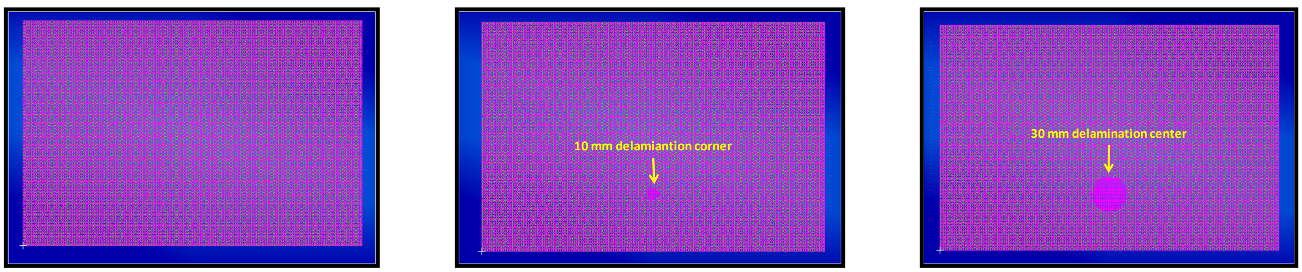


Figure 3 Full finite element model of the sandwich panel

The following Fig. 4 (a), (b), and (c) shows the finite element model of the sandwich panel for the cases of no delamination, 10 mm delamination at the centre, and 30 mm delamination at the centre. The delamination was made at the bottom half of the model because the model was symmetrical horizontally. The material properties for the honeycomb core and the carbon fibre skin were as the following Table 1 [10-12].

For the load and boundary conditions, the following Fig. 5 shows the arrangement. In Fig.5, the model underwent a 3-point bending test where the supports were rigid displacement on both of the top ends of the models. The load was applied from the bottom centre while the delamination was at the top of the model. In the FEA, the parameter that will be analysed was the displacement of the approximate location of Point A and Point B in the model. The load applied was 0.0 kN to 2.5 kN with an increment of 0.2 kN.



(a) (b) (c)

Figure 4 Full FE models of sandwich panel. (a) without delamination, (b) 10 mm delamination centre, (c) 30 mm delamination centre.

Table 1 The material properties of the sandwich panel used in the Nastran/Patran.

Material Properties	Carbon Fibre	Honeycomb Core
Elastic Modulus 11 (N/mm^2)	70,000	137.9
Elastic Modulus 22 (N/mm^2)	70,000	-
Poison's Ratio 12	0.10	0.49
Shear Modulus 12	31,818	46.27
Shear Modulus 23	-	22.8
Shear Modulus 13	-	44.1
Density (kg/mm^3)	1.6 e-3	4.8 e-5
Thickness (mm)	0.1	0.05

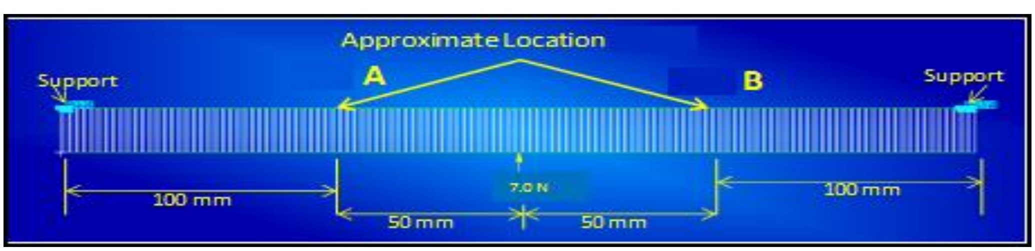
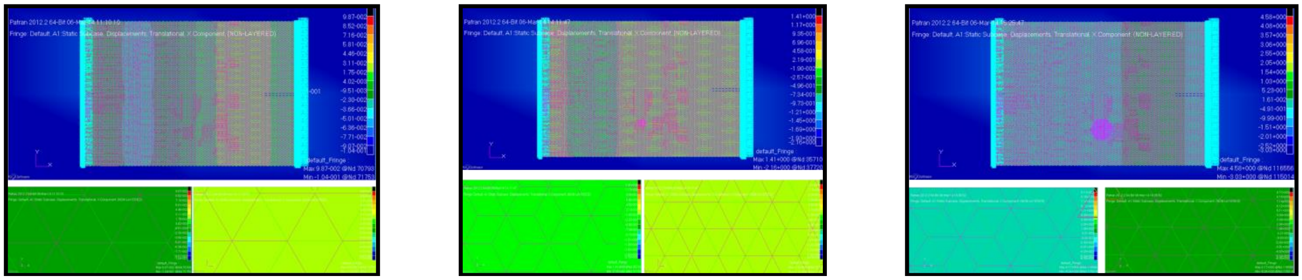


Figure 5 FE model with the set-up of load and boundary conditions.

4.0 RESULTS AND DISCUSSION

The following Fig. 6 (a), (b), and (c) show the displacement contour of the FEA for no delamination, 10 mm delamination, and 30 mm delamination respectively. From the displacement contour, the result was tabulated in the Table 2 and plotted in a graph as in the following Fig. 7 (a).

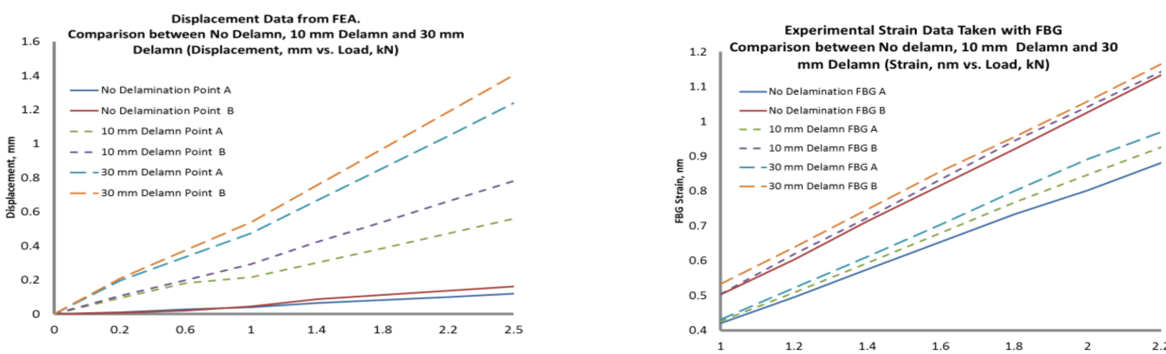


(a) No delamination (b) 10 mm delamination (c) 30 mm delamination

Figure 6 Displacement contour of the FEM of the sandwich panel under transverse loading at the load of 1 kN.

Table 2 Percentage displacement of top skin due to load at Point A and Point B of FE.

No Delam A	No Delam B	10 mm Delam A	10 mm Delam B	30 mnm Delam A	30 mm Delam B
12%	16%	56%	76%	124%	120%



(a) (b)

Figure 7 (a) The FE result of the displacement vs. load, (b) Experimental result of the optical strain vs. load

From Fig. 7 (a) above, it could be seen that for the sandwich panel without delamination, the displacement is very low (12% & 16%), while for the 10 mm delamination; the displacement is greater (56% & 76%) whereas for the larger 30 mm delamination, the displacement was the largest (124% & 140%). The FE was validated by experiment using FBG to record the optical strain displacement. The result showed similar trend of results where the size of the delamination could be determined by comparing the displacement of the skin.

4.0 CONCLUSION

In conclusion, the paper has presented the determination of delamination size of honeycomb sandwich panel using Finite Element Method. The comparison was made between the sandwich panel without delamination with the 10 mm centre delamination, and 30 mm centre delamination respectively. In the simulation it was found that the FEA was able to determine the relative size of the delamination of the skin of the sandwich panel. The results of the finite element simulation had been verified through experimental procedure and the trend was found to be similar.

Acknowledgement

The authors would like to thank Flight Technology and Testing Centre, FTTC, Faculty of Mechanical Engineering, Universiti Teknologi MARA, UiTM, who made possible the publication of this paper.

References

[1] Herrmann, A. S., Zahlen, P. C. and Zuardy, I. 2005. Sandwich structures technology in commercial aviation: Present applications and future trends. *Sandwich Structures 7: Advancing with Sandwich Structures and Materials. O. T. Thomas and e. al., Eds., ed Netherlands: Airbus Deutschland GmbH*: 13-26.

[2] Stickler, P. 2002. Composite materials for commercial transport-issues and future research directions. *Americal*

Society for Composites 17th Technical Conference, West Lafayatte, Indiana, USA.

[3] Niu, M. C. Y. 2008. *Composite Airframe Structures: Practical Design Information and Data*. Hong Kong: Hong Kong Conmilit Press Ltd.

[4] Van-Tooren, M. J. L. 1998. *Sandwich Fuselage Design*. PhD, Faculty of Aerospace Engineering, Delft University of Technology, Delft, the Netherlands.

[5] Takeda, N., Minakuchi, S. and Okabe, Y. 2007. Smart composite sandwich structures for future aerospace application - Damage detection and suppression-: A review. *Journal of Solid Mechanics and Materials Engineering.* 1: 3-17.

[6] Davies, G. A. O., Hitchings, D. and Ankersen, J. 2006. Predicting delamination and debonding in modern aerospace composite structures. *Composite Science and Technology.* 66: 846-854.

[7] Chen D. H. and Ozaki, S. 2009. Stress concentration due to defects in a honeycomb structure. *Composite Structures.* 89: 52-59.

[8] Han, T.-S., Ural, A., Chen, C.-S., Zehnder, A. T., Ingraffea, A. R. and Billington, S. L. 2002. Delamination buckling and propagation analysis of honeycomb panels using a cohesive element approach. *International Journal of Fracture.*115: 101-123,.

[9] Cerracchio, P., Gherlone, M., Sciuvia, M. D. and Tessler, A. 2015. A novel approach for displacement and stress monitoring of sandwich structures based on the inverse Finite Element Method. *Composite Structures.* 127: 69-76.

[10] I. Core Composites. *Nomex Honeycomb: Lightweight Non-Metallic Composite Honeycomb*. [Online] From: www.corecomposites.com/composites/core/honeycomb/nomex/.[2012, 20 March 2013].

[11] H. Corporation. 1999. HexWeb H*oneycomb Attributes and Properties: A comprehensive guide to standard Hexcel honeycomb materials, configurations and mechanical properties. ed.* Pleasanton, California: Hexcel Composite.

[12] P. C. Ltd. *Mechanical Properties of Carbon Fibre Composite Materials*. [Online] From: www.performance-composites.com/carbonfibre/mechanicalproperties_2.asp. [2009, 20 March 2013].