International Journal of Engineering & Technology, 7 (4.25) (2018) 160-162

International Journal of Engineering & Technology

Website: www.sciencepubco.com/index.php/IJET

Research paper

Multi-Configuration Stiffened Panels under Compressive Load: Part 2 – Finite Element Analysis

Ramzyzan Ramly[1]*, Amir Radzi Abdul Ghani[2], Rizal Effendy Mohd Nasir[1], Zulkifli Mohamed[3], Wahyu Kuntjoro[1]

[1]Flight technology & Test Centre (FTTC), Faculty of Mechanical Engineering, Universiti Teknologi MARA, Shah Alam, Malaysia
[2]Automotive Research & Test Centre (ARTeC), Faculty of Mechanical Engineering, Universiti Technology MARA, Shah Alam, Malaysia
[3] Faculty of Mechanical Engineering, Universiti Technology MARA, Shah Alam, Malaysia
**ramzyzan@salam.uitm.edu.my*

Abstract

Stiffened panels are the structure used in the aircraft wing skin panels. Stiffened panels are often critical in compression load due to its thin structural configuration. This paper is the continuation of the previous work which was focusing on the theoretical aspect. This paper will explore the Finite Element Analysis of the multi-configuration structure of the stiffened panels. Using ANSYS Workbench, the process of drawing the geometry, meshing the model, setting up the boundary conditions, running the program, and obtaining the results were performed. The loads to apply were obtained from the previous research. The results of the FEA shows the critical locations in the stiffened panels such as distortions, deformation, and the stress distributions.

Keywords: *Multi-Configuration Stiffened Panels; Finite Element Analysis; Stress Distribution.*

1. Introduction

The objective of the study is to analyze the stress distribution of the multi configuration stiffened panels under axial loads based on the critical load obtained in the previous study [1] using Finite Element Analysis [2]. Stiffened panels are widely used in structures especially in vehicle such as aircraft, ship [3], high-speed train, steel bridges, and multi-segmented road vehicles. This study however, focused on aircraft structures, where stiffened panels are widely used in the bulkhead of aircraft fuselage and the wing skins and wing ribs [4]. In the similarity study [5-7], the research is more focused on the wing upper skin panels since the upper skin panels experience compression [8] loads while the lower skin panels experience more tensile loads. After the completion of this FEA, the study will be an experimental with embedded sensors [9, 10].

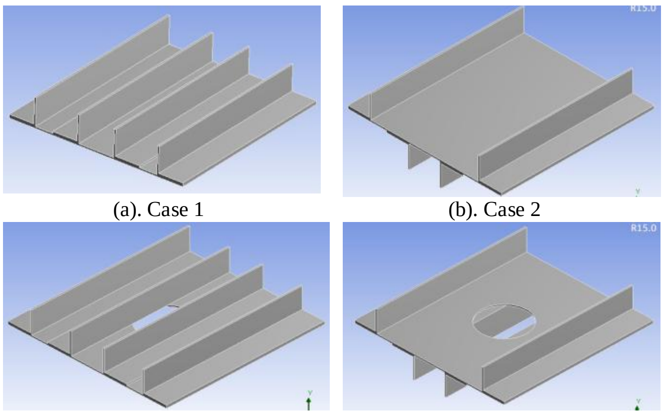
As in the previous study, the material of this study is also aluminum alloy 2024-T3 which has the Young Modulus 70 GPa or 70.0 x 10^9 N/m^2 and Poisson ratio of 0.33.

2. Methodology

In this paper, there are four (4) cases being studied. The size for each case is 100 mm x 100 mm. Each case has four (4) L-shaped stiffener having dimension of 25 mm x 25 mm. This condition will make each case having the same cross-sectional area, but each different configuration will have different moment of inertia, I as can be seen in Fig.1 a, b, c, and d. Even though the moment of inertia for Case 3 equals to Case1, and Case 4 equal to Case 2, the moment of inertia and the cross-sectional area for cases 3 and 4 will be different because there is a hole in the middle of the panel.

3. Finite Element Analysis Process

Using Ansys Workbench, the four cases of the solid model of the configurations were produced as can be seen in the Fig. 1.



(a). Case 1 (b). Case 2

(c). Case 3 (d). Case 4

Fig.1: Cases of Multi-configuration stiffened panel.

3.1 Meshing

In order to do the FEA, the model has to be meshed, applying the loads and setting the boundary conditions. By using ANSYS Workbench software v 15.0, the solid models were automatically meshed using solid either SOLID185 or SOLID186 quadrilateral element for Case 1 and Case 2 while triangular solid element for Case 3 and Case 4. Fig.2 shows the meshing figures of the cases.

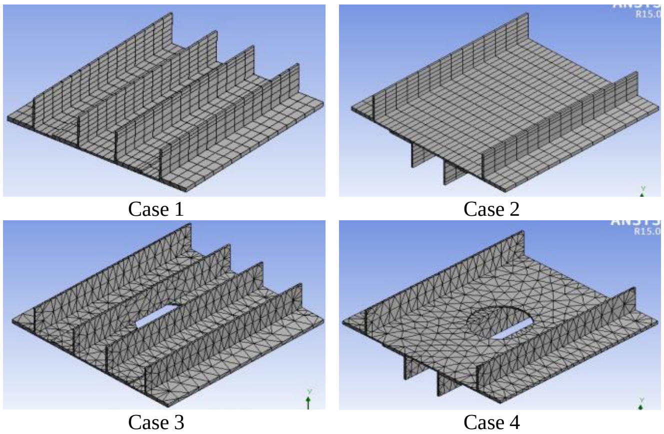


Fig. 2: Meshing of all cases.

3.2 Applying Load and Boundary Condition

The next process in the FEA is loading and boundary condition. The specimen is subjected to compressive axial loads and the direction of the load applied is axially parallel to the direction of the stiffener and can be seen in Fig.3. In the Figure the dashed line is the line (direction of the stiffener).

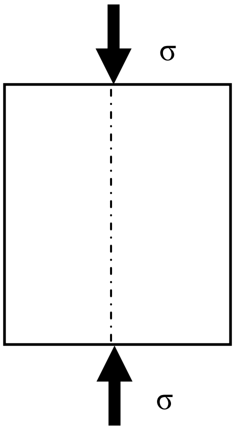


Fig. 3: Axial compression of the stiffened panel.

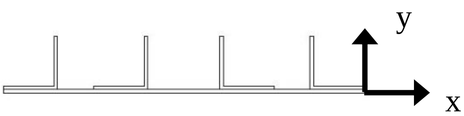


Fig. 4: Axis orientation of the stiffened panel.

The load applied was 1.185 GPa along the top edge (refer Fig 3) and the bottom edge was pin supported in x and y axes. As can be seen from Fig 4, the whole surface is covered by the skin that lies along the x axis, and also the four (4) stiffeners which have flanges in x and y direction.

4. Results and Analysis

The results of FEA from the study was contour plot analysis only and the deformation of each case was observed.

Fig. 5 shows the contour plot of Case 1. The stress being viewed was the normal stress about x-axis. In this figure it could be concluded that this configuration is strong and good configuration. Fig. 6 shows the contour plot of Case 2 configuration. The contour plot being viewed in this figure was deformation about x-axis. The variation of color in this figure where extreme red on one side and extreme blue on the other side are just coordinate system where the maximum deformation was only 0.398 mm each side of the x-axis.

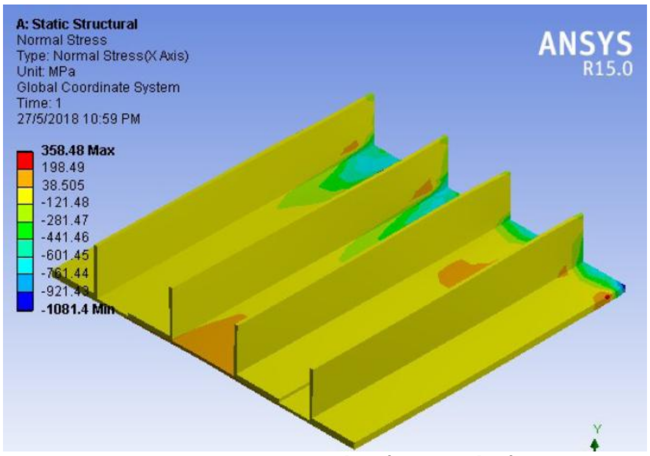


Fig. 5: Isometric view contour plot of FEA result of Case 1.

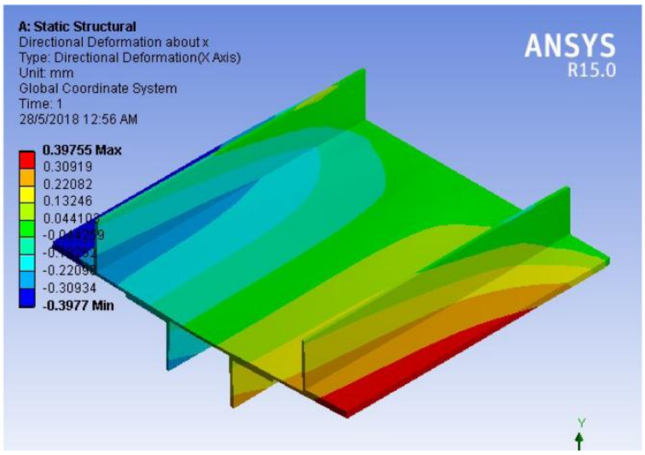


Fig. 6: Isometric view contour plot of FEA result of case 2.

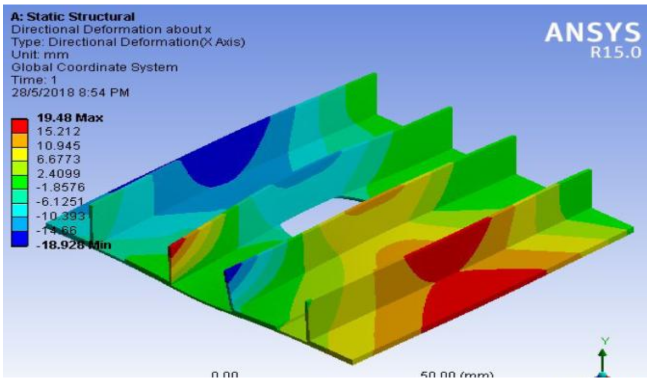


Fig. 7: Case 3: Isometric view.

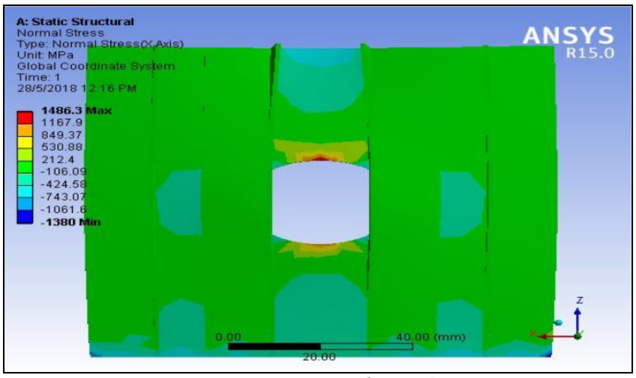


Fig. 8: Case 3: Top view of normal stress.

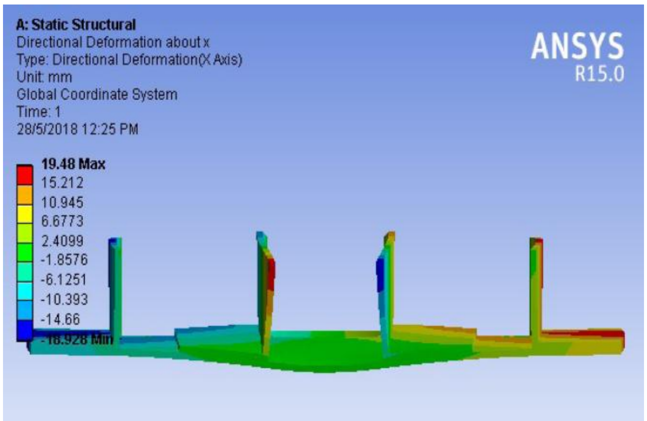


Fig. 9: Case 3: Side view of x direction deformation.

Fig.7 – Fig 9 presented to deformation and the normal stress for Case 3. From the figures, for this stiffened configuration panel, the maximum deformation was 18.48mm and occurred at the area where there was a hole at the middle of the panel. From Fig. 8, the stress can be seen highest at the edge of the hole in y-axis where 1483.6 MPa was recorded. However, when observing Fig. 9, there was only a little physical deformation neither in positive y direction nor negative y direction.

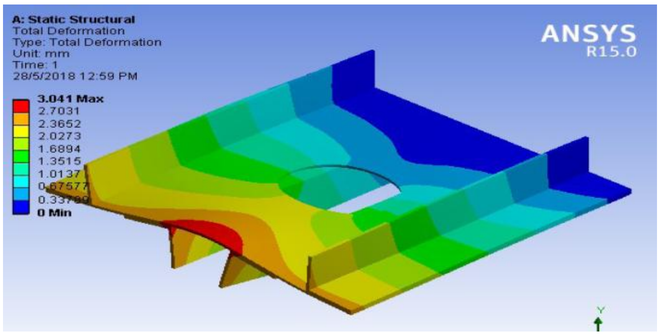


Fig. 10: Case 4: Isometric view.

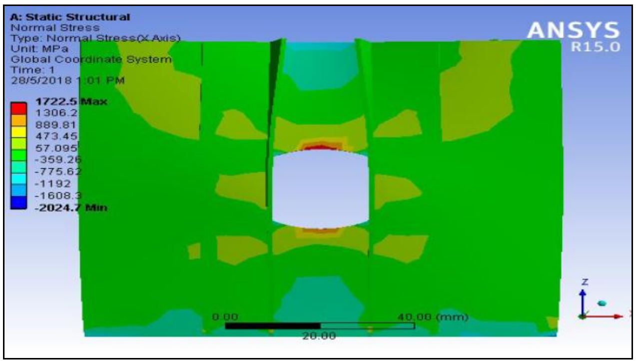


Fig. 11: Case 4: Top view of normal stress.

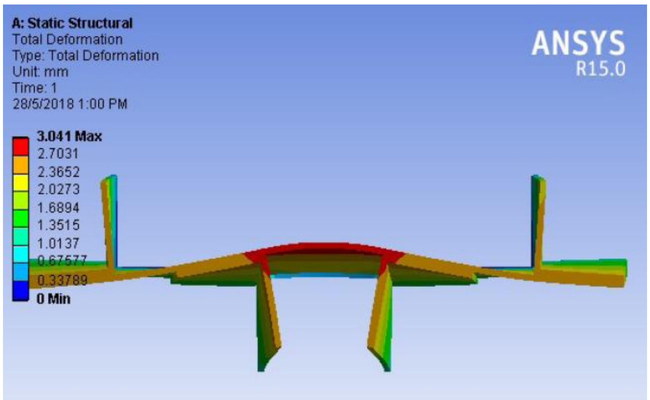


Fig. 12: Case 4: Front view of total deformation.

Fig. 10 – Fig 12 presented the deformation and the normal stress for Case 4. From the figures, for this configuration, the total deformation was only 3.041 mm (Fig.10). However, the maximum normal stress was found to be 1722.5 MPa (Fig. 11), higher than the one found in the Case 3 (Fig. 8). Due to this higher stress, the stiffened panel was distorted more as can be seen in Fig. 12. In the figure, the distortion occurred mostly the region where there was a hole in the flat part of the panel. As can be seen, the two stiffeners in the middle of the panel distorted more than the outer two.

5. Conclusion

This paper has presented the Finite Element Analysis of the multi-configuration stiffened panels normally used in the aircraft's wing section. From the analyses, it could be said that the Case 1 and Case 3 were suitable to be used in the skin of the wing, because the deformation occurred along and towards inside of the panel (with reference that the stiffener's side is the inside). Case 2 and Case 4 were suitable to be used in the rib component of the wing due to the nature of the deformation of the panel being on both sides of the configuration.

Acknowledgement

The authors would like to acknowledge Faculty of Mechanical Engineering, Universiti Teknologi MARA, FKM UiTM for sponsoring the Conference, and also the Institute of Research Management and Innovation, IRMI, Universiti Teknologi MARA, UiTM, Shah Alam, Malaysia in providing the Lestari Fund (Project Number: 600-RMI/DANA 5/3/LESTARI (84/2015)) which was completed on 31 May 2018.

References

[1] R. Ramly, W. Kuntjoro, A. R. A. Ghani, R. E. M. Nasir, and Z. Mohamed, "Multi-configuration Stiffened Panels under Compressive Load: Part 1 - Theoretical Analysis," *International Journal of Engineering & Technology* vol. 7, no. 3.11, p. 5, 2018.

[2] C. Li, Z. Zhu, H. ren, and C. G. Soares, "Finite Element Analysis of the Ultimate Strength of Aluminum-Stiffened Panels With Fixed and Floating Tranverse Frame," *Journal of Offshore Mechanics and Arctic Engineering*, vol. 139, no. 4, p. 10, 2017.

[3] J. K. Paik and A. Duran, "Ultimate Strength of Aluminum Plates and Stiffened Panels for Marine Application," *Marine Technology*, vol. 41, no. 3, p. 14, 2004.

[4] J. S. Przemieniecki, "Finite Element Structural Analysis of Local Instability," *AIAA Journal*, vol. 11, no. 1, p. 6, 1973.

[5] K. L. Tran, C. Douthe, K. Sab, J. Dallot, and L. Davaine, "Buckling of stiffened curved panels under uniform axial compression," *Journal of Construction Steel Resaerch*, vol. 103, pp. 140-147, 2014.

[6] S. Shroff, E. Acar, and C. Kassapoglou, "Design, analysis, fabrication, and testing of composite grid-stiffened panels for aircraft structures," *Thin-Walled Structures*, vol. 119, pp. 235-246, 2017.

[7] J. M. Gordo and C. G. Soares, "Compressive tests on stiffened panels of intermediate slenderness," *Thin-Walled Structures*, vol. 49, no. 6, pp. 782-794, 2011.

[8] C. Lynch, A. Murphy, M. Price, and A. Gibson, "The Computational Post Buckling Analysis of Fuselage Stiffened Panels Loaded in Compression," *Thin-Walled Structures*, vol. 42, no. 10, p. 19, 2004.

[9] Y. Mo, D. Ge, and J. Zhuo, "Experiment and analysis of hat-stringer-stiffened composite curved panels under axial compression," *Composite Structures*, vol. 123, no. May 2015, pp. 150-160, 2015.

[10] F. P. Beer, J. E. Russel Johnston , J. T. DeWolf, and D. F.Mazurek, *Mechanics of Materials Fifth Edition in SI Units*. New York, USA: McGraw Hill, 2009, p. 782.