VOL. 12, NO. 15, AUGUST 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

# THEORETICAL AND FINITE ELEMENT METHOD OF STATIC STRUCTURAL ANALYSIS AT WING SEGMENT

Matzaini Katon[1, 2], Norhanifah Abdul Rahman[1] and Norhashidah Manap[1]
[1]Fakulti Kejuruteraan Mekanikal, Universiti Teknologi MARA Cawangan Terengganu Kampus Bukit Besi, Dungun, Terengganu, Malaysia
[2]Fakulti Kejuruteraan Mekanikal, Universiti Teknologi MARA, Shah Alam, Selangor, Malaysia
E-Mail: matzainikaton@tganu.uitm.edu.my

**ABSTRACT**

In this paper, the comparison between two methods: theoretical and finite element static structural analyses were studied. These two methods were used in finding the stress value related to static analysis. These comparisons were made on a wing segment of a Blended Wing Body (BWB). MSC PATRAN and MSC NASTRAN were used as for the Finite Element Analysis (FEA) platform. Finite element models for the wing segments were developed in MSC PATRAN. CQUAD4 and CTRIA3 elements were used to represent the individual components of the wing segment such as skin and web.

**Keywords:** finite element analysis (FEA), theoretical structural stress analysis.

## INTRODUCTION

One of the most important processes in designing an aircraft is the structural analysis of the airframe. This process is done in order to obtain the structural behaviour of the aircraft structural configuration. It can be conducted in a various method such as by using the theoretical static structural analysis. However, this method is limited to non-complex shape structural configuration.

Besides that, one of the methods which have the capability in analysing the structural analysis is the Finite Element Method (FEM) [1-6]. The capability of FEM is not only limited in structural analysis, but also has been used in a vast field of analysis [7-9]. FEM is defined as a tool for analysing a prediction of engineering systems response [10]. FEM has the capabilities to find the structural characteristic and these advantages, enables the design process of an aircraft to become more cost and time effective. However, the accuracy and the quality of the FEM analysis is depending on the assumption made, the user skills and experiences.

Finite Element Analysis (FEA) process starts with splitting the structural into a series of smaller elements. These smaller elements later are joined together through nodes. The physical property for the airframe is represented by identification and selection of materials and elements properties. Then, for the specification of environment, the loading and boundary conditions are defined and determined. Generally, the outcomes of the result from the structural analysis are in the form of stress distribution, structural deformation or pressure distribution. The outcomes can be represented in numbers and illustrated in graphical method. Usually, there are three activities related to FEM computer software, which are: pre-processor, solver and post-processor.

The FEM has played an important role in the aircraft industry. Amongst the usage of FEM in airframe structural analysis is static analysis of PW-141 SAMONIT UAV [3]. This airframe model has been established from three-dimensional computer aided design into two and one-dimensional FEM model by using PATRAN software. Static analysis on the lifting surfaces performed by using MD NASTRAN. The model is constraint at the centre of aircraft mass node. The outcome of the result is based on the node displacement from the static equation.

Besides FEM, the theoretical structural analysis also has been widely used to analyse the airframe static analysis. One of the studies related to the evaluation of the accuracy simple classical methods by comparing them to the experimental static test and FEA of CN-235 wing structure has been done [2]. It was found that, the method based on equation (i) and (ii) produce results with a difference around 20% and 10% compared between FEA and test results. The FEA and classical analysis results are quite close to the results given by the static test. It can be concluded that, for a structure with local effect due to the existence at inspection holes, it needs a detailed analysis, in which shear diffusion presented.

$$\sigma_x = \frac{M_y / H}{A_{panel}} \tag{1}$$

$$\sigma_x = \frac{I_y \mathrm{Mz} - I M_y}{I_y I_z - I^{2\mathrm{yz}}} y + \frac{I_z M_y - I_{\mathrm{yz}} M_z}{I_y I_z - I^{2\mathrm{yz}}} z \tag{2}$$

## METHODOLOGY

### Structural configuration layout

Figure-1 shows the structural configuration layout of a wing segment, in which the area of cross section of the wing was based from the cross-sectional area of Blended Wing Body (BWB) model [11] respectively. The wing configuration has a total length of 8000 mm and a total width of 423.84 mm. This structural configurationally layout was used as the simple comparison between static structural analysis and finite element analysis. Aluminium 2024-T4 was used for the wing segment materials.

VOL. 12, NO. 15, AUGUST 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

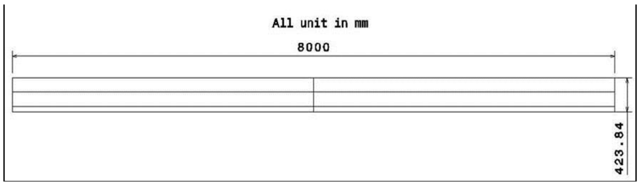


**Figure-1.** Configuration of wing section.

**Theoretical static structural analysis**

The wing segment was assumed to be supported as a cantilevered beam experiment intheoretical static structural analysis. This was due to the resemblance of half BWB with cantilevered beam in nature. Nevertheless, due to the complexity of the wing segment which has the configuration of unsymmetrical cross-sectional area, the general formula of bending stress analysis was utilized. The equation for stress formula was shown in equation (iii);

$$\sigma_{\Sigma} = \left( \frac{M_y I_{xx} - M_x I_{xy}}{I_{xx} I_{yy} - I_{xy}^2} \right) x + \left( \frac{M_x I_{yy} - M_y I_{xy}}{I_{xx} I_{yy} - I_{xy}^2} \right) y \qquad (3)$$

Where, My = Bending Moment of the respective section with respect to y-axis, Mx = Bending Moment of the respective section with respect to x-axis, Ixx = Second moment of area of respective section with respect to x-axis, Iyy = Second moment of area of respective section with respect to y-axis, Ixy = Product of moment of inertia of respective section and x,y = Distance between centroid of respective section to the point of interest.

**Finite element method**

A total of applied loads 1 kN was given at the wing ends (wing tip). The wing segment was fixed to three degree of freedom (DOF). For this reason, the wing segment was assumed to be a cantilevered beam problem. The finite element model was established in MSC PATRAN software. CQUAD4 and CTRIA3 element was used to model skins and webs on the wing segment. The outcome of this study focused on the comparison theoretical and finite element analysis of the resultant stress value at segment 1 (shown in Figure-2).

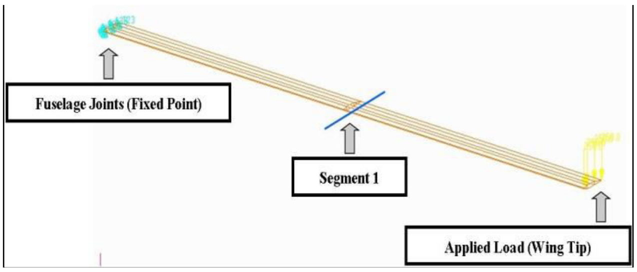


**Figure-2.** Arrangement of segment 1, applied load and fixed point at the wing segment.

**RESULTS AND DISCUSSIONS**

**Theoretical static structural analysis**

Theoretical analysis was done using equation (iii). The wing segment was assumed to be like a cantilevered beam problem. Bending moment of segment 1 with respect to x-axis, $M_{Segement1,\ x}$ of the wing segment was given as follows;

$$M_{\text{Section1},x} = (F_z)(L) \qquad (4)$$

Where, $F_z$ = Total force (loading) acting at the wing segment and L = Distance between force and segment 1.

It was found that, the moment at segment 1 in the x direction is $M_{\text{Segment1},x}$= 4000Nm. The value of moment at location for segment 1 with respect to y-axis, $M_{Segment1,y}$ of the wing was assumed to be zero due to there was no force acting at x-direction. For the value of y and x, these values were determined from the distance of centroid to the selected point. It was found that, the value of y and x were 36.557 mm and -128.616 mm. Hence, the value of stress at segment 1 from equation (iii) is $\sigma_{\text{Segment1}}$= -297.38MPa.

**Finite element analysis**

The finite element analysis was done through the aid of MSC NASTRAN. The resultant stress which occurs at segment 1 was found to be 14 MPa taken from Node ID of 16. Figure-3 shows the finite element analysis of the wing segment.

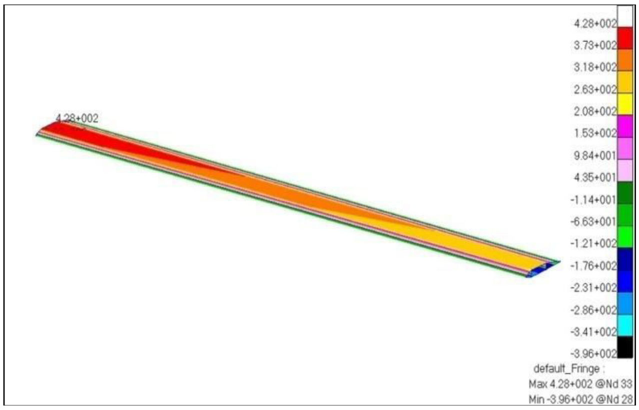


**Figure-3.** Stress distribution in the x-axis.

**Percentage of differences**

The outcome stress value in the x-axis from the theoretical static structural analysis was compared with the finite element static structural analysis. It was found that, there was a slight difference in terms of a value of stress between the two methods. The values of stress for theoretical and finite element static structural analysis were -297.38 MPa and 318 MPa respectively. From the discrepancy of stress value, it was found that the calculated percentage of differences for section 1 is 6.9 %.

## CONCLUSIONS

In this study, a comparison of stress was done by comparing the values between theoretical static structural analysis and finite element analysis. It found to be, the percentage of differences for segment 1 was 6.9%. This difference was due to finite element analysis are more capable in analysing a more complex shape structural configuration model compared to the theoretical method. Furthermore, the formula used in theoretical analysis was more suitable for long slender type of wing layout.

## REFERENCES

[1] W. Kuntjoro, R.E.M. Nasir, W. Wisnoe, A.M.I. Mamat, and M. R. Abdulah, "Computer Aided Design and Engineering of Blended Wing Body UAV Structure," in Proceedings RAeS/CEAS Aircraft Structural Design Conference, Liverpool, UK, 2008: Royal Aeronautical Society, UK.

[2] W. Kuntjoro, S. Kamil, H. K., and M. M. (2002) Investigation of Practical Analysis for the Purpose of Structural Design of Wing Panel. IEM Journal. 7-10.

[3] A. Olejnik, S. Kachel, R. Rogólski, and P. Leszczyński, "Unmanned Aircraft Structural Model and its FEM Analysis for Strength Evaluation," in Czech AEROSPACE Proceedings, 2009, vol. 2, pp. 14-18: Czech Aerospace Manufacturers Association / ALV.

[4] M. Katon and W. Kuntjoro, "BWB Baseline II-E2 UAV Structural Finite Element Model and Analysis," in International Conference on Electrical, Control and Computer Engineering, Pahang, Malaysia, 2011, pp. 187-190.

[5] M. Katon and W. Kuntjoro, "Airframe Design for Modularity of the BWB Baseline II-E2 UAV," in 2011 International Conference on Business, Engineering and Industrial Applications, Kuala Lumpur, Malaysia, 2011, pp. 47-51.

[6] M. Katon, W. Kuntjoro, N.A. Rahman, and Z.M. Ali, "Static Structural Analysis of Blended Wing Body II-E2 Unmanned Aerial Vehicle," Journal of Applied Environmental and Biological Sciences, vol. 7, no. 6, pp. 91-98, 2017.

[7] N. S. Ngarisan, Y. S. Hoe, N. I. Norddin, and N. S. K. Abdullah, "The Application of Finite Element Method to Solve Heat Transfer Problem Involving 2D Irregular Geometry," Journal of Applied Environmental and Biological Sciences, vol. 6, no. 11S, pp. 23-30, 2016.

[8] M. Guettler, C. Jelich, S. Marburg, E. Grasso, and S. D. Rosa, "A vibroacoustic analysis of pre-stressed saw blades to identify instabilities considering gyroscopic effects and centrifugal forces utilizing the finite element method," The Journal of the Acoustical Society of America, vol. 141, no. 5, pp. 3576-3576, 2017.

[9] A. Laadhari and G. Székely, "Eulerian finite element method for the numerical modeling of fluid dynamics of natural and pathological aortic valves," Journal of Computational and Applied Mathematics, vol. 319, pp. 236-261, 01 August 2017.

[10] W. Kuntjoro, An Introduction to the Finite Element Method. McGraw-Hill Education (Asia), 2005.

[11] M. Katon, W. Kuntjoro, R. Ramly, and R.E.M. Nasir, "Progress in the Airframe Design of BWB-Baseline II E2 UAV with Canard," presented at the Proceedings of the 2010 International Conference on Advances in Mechanical Engineering, Shah Alam Convention Centre, Selangor, Malaysia, 2 - 5 December 2010.