

Structural health monitoring (SHM) on high stress loading of wing bracket

Open
Access

Imran Mohd Salleh ^{1,*}, Shamsul Sarip ², Mohd Nabil Muhtazaruddin ², Mohamad Zaki Hassan ²,
Mohd Yusof Md Daud ², Sa'ardin Abdul Aziz ²

¹ Universiti Kuala Lumpur Malaysian Institute of Aviation Technology, Lot 2891 Jalan Jenderam Hulu, Jenderam Hulu, 43800 Dengkil, Selangor, Malaysia

² UTM Razak School of Engineering and Advanced Technology, Univesiti Teknologi Malaysia, Jalan Sultan Yahya Petra, 54100 Kuala Lumpur, Malaysia

ARTICLE INFO

Article history:

Received 8 October 2016

Received in revised form 15 November 2016

Accepted 18 November 2016

Available online 6 December 2016

ABSTRACT

This paper presents a Structural Health Monitoring (SHM) approach by addressing the modeling based on Cessna 310 wing bracket to collect its baseline data. SHM is a novel technology application in most engineering field including aircraft and aerospace field for defect detection including crack and other damage area. The main purpose of this project is to assess their structural integrity due to designated loading and boundary conditions using ABAQUS modeling technique application. These parts were selected as its structures are specifically complex and considered as ageing aircraft parts. The work can be also regarded as platform for CAD model database for ageing aircraft parts investigation upon their structural analyses and design preference. The works begins with a precise measurement using basic gauges and transfer to a 3-D solid modeling using CATIA V6. The optimization technique introduces in this work will be of a novel approach since the CATIA will be combined with the Matlab programming software and Finite Element Analysis (FEA) model in order to predict the damage location with respect to the prescribing boundaries and loadings conditions. The present works shows the application of the FEA of Abaqus / CAE software programming to build its graphical user interface (GUI) and Matlab software used to control its algorithms for its structural analysis in relation to the components.

Keywords:

Structural health monitoring, Finite element analysis, Wing bracket, Aircraft, Structural analysis, Optimization

Copyright © 2016 PENERBIT AKADEMIA BARU - All rights reserved

1. Introduction

Modeling approach is nowadays can be considered crucial for designing engineering structures, especially for complex structural parts and ageing such as aerospace related components. This technique also being implemented on ageing aircraft for SHM structural design phase and it is very

* Corresponding author.

E-mail address: imransalleh@unikl.edu.my (Imran Mohd Salleh)

much concerned with the context of loading and boundary conditions, critical fail areas and the damage monitoring aspects. This modeling result give the initial collection of data in which it serves as a basis for comparison with subsequently acquired data. It is obvious that, many engineering structures loading conditions have significantly changed throughout operational life. It is even worse when these older components (25 years old or more) do not have detail structural design records that can be utilized by engineers today in order to assess the current structural status using advanced modeling technique. Prior to this phenomenon, the actual critical parts that susceptible to failures cannot be made and predicted using FEA technique. Thus, the need to dismantling the overall structural systems is highly recommended in ensuring the safety and reliability of these older structures. This laborious process require manpower related and demand sophisticated equipment will result in high costing, time consuming and delaying aircraft flying schedule.

The numbers of ageing aircraft are steadily increased in the world today as documented in the National Transportation Safety Board (NTSB) report. It is statically reported that there was an approximately 1.3% increment of civil aircraft ages more than 15 years in the two years' time from 1997 to 1999, and nearly 2130 units ranged between 20-25 years old [1]. Additionally, it is well known that, the adoption of new materials, parts and technologies into aerospace structures is very conservative and heavily dependent upon past design technologies. Therefore, by taking advantages of these stringent regulations parts of the investigated structures involved in this research work are most likely being utilized by the existing modern aircrafts. This work can also be regarded a platform for a CAD model database for older aircraft parts that can be utilized by other interested design based parties and structural analysis system.

There is a significant reported by other researcher that are related towards the parameters setting for baseline data. A benchmark study, where some group of people participates by applying some number of monitoring techniques with common action to attain the best result parameters on the same structure thus help forward for a comparison of commonly used techniques [2]. The 15th International Modal Analysis Conference (IMAC XV) known as the earliest attempt conducted and it is successfully identifying the location and severity of damage in all cases of the benchmark problem. [3]. Later on, the International Association for Structural Control (IASC) formed SHM task group under the auspices of the American Society of Civil Engineering (ASCE) [4], aiming at studying various SHM methods. This research work has been continuing under the sponsorship of International Association of Structural Control (IASC) and Dynamics Committee of ASCE. As a result, a series of benchmark for SHM problems starting from the simplest method to realistic approach has been developed by IASC SHM Task Group [5]. The main point here is the modeling techniques also can be as baseline data collections in SHM to minimize the related cost thus maintain the highest reliability and integrity level.

Meanwhile, the very first FEA computer based program was called NASTRAN and it was originally developed for NASA in the late 1960s under United States government funding for the Aerospace industry [6]. The MacNeal-Schwendler Corporation (MSC) was one of the principal and original developers of the public domain NASTRAN code [7]. NASTRAN source code is integrated in a number of different software packages, which are distributed by arrangement of companies [8]. NASA wanted some development proposals for upgrading the program and made an official proposal in 1965. After the technological advancements many new programs emerged started from this evolution and innovation made by other researchers. Abaqus is one of the FEA software suites for finite element analysis and computer-aided engineering, originally released in 1978 [9]. The name and logo of this software are based on the Abaqus calculation tool [10].

2. Research methodology

In the initial stage of this research work a wing aircraft bracket of Cessna 310 was selected and modeled in order to achieve the first phase of the research objectives. The modeling phase will be focusing in identifying the area that is susceptible to damage due to the prescribing boundaries and loadings condition. Few case studies will be conducted here and the optimization technique introduces in this work will be combined with the developed FEA model in order to predict the damage localization occurring on the selected specimens. The findings of this research will have the significant impact not only to SHM research towards extending the lifespan on the structural design studies in minimizing the high stress and strain areas on the investigation.

2.1 Creating and analysing a model using the ABAQUS modules: Part module

A 3-D solid modeling of the investigated specimen was made using CATIA version (Fig. 1). The measurement was made using radius gauge, steel gauge, and vernier caliper to obtain an accurate and precise measurement. ABAQUS/CAE is divided into eight modules where each modules has own tools and function to define the modeling process [11]. In the future the laser scanning and ball probing for 3D modeling purposes in order to get accurate measurement.

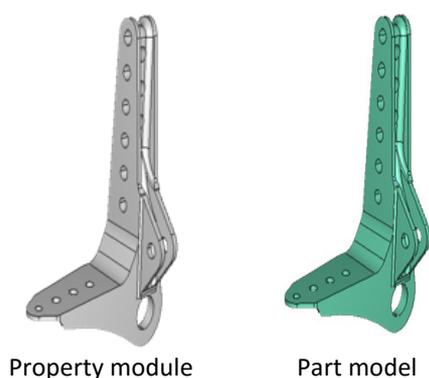


Fig. 1. Property Module and specimen model

2.1.1 Property module

Exotic alloy is used, since most of the metallic structure for this aircraft parts is made for this metal family. The properties are isotropic materials and details are as follow in Table 1.

Table 1
 Properties of specimen material

Properties	Value
Modulus of Elasticity, E	199.94 [GNm ⁻²]
Shear Modulus of Elasticity, G	70 [GNm ⁻²]
Poisson's Ratio, ν	0.29
Weight Density	2800 [kgm ⁻³]

The model is mesh using tetra element C3D4 or three dimensional element region where C is the continuum stress or displacement, 3D is the three dimensional and 4 represent the number of node for linear tetrahedron (Fig. 2). Due to the difficulties to mesh using hex element which is the default

meshing technique, this tetra element is using because of the model can be mesh on the surface, easy to identified parameter and make analysis easier [11 and 12]. Node and element label can be view after the part being mesh. Node and element label is the important detail because stress and critical point can be localized or identified after the job is submitted for analysis. Node and element labels are part of parameter that will be used in optimization technique using MATLAB.

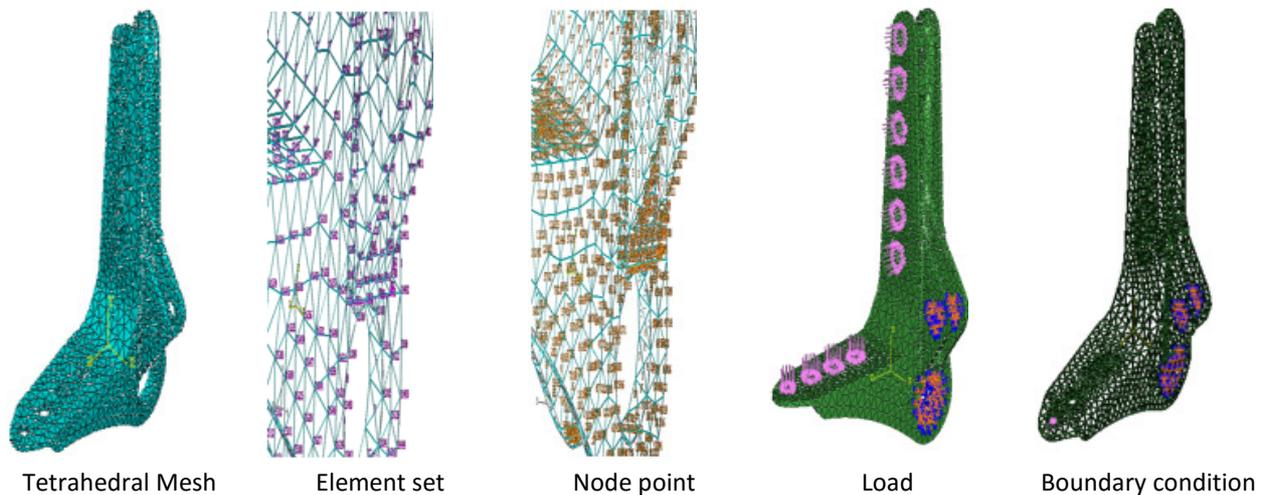


Fig. 2. Tetrahedral Mesh, Element Set, Node Point, Load, Boundary Condition

2.1.2 Job module

Job is created in this module in order to submit for analysis, and then after job is created the input file is writing. The used of input file will be used for identifying fitness value.

2.1.3 Visualization module

The result of the analysis can be view in this module where it shows the higher point stress upon loading and boundary condition given.

2.1.4 Fitness value from wing

The input file is imported from temporary file folder to ABAQUS CAE in order to assign element set and node set to the sample part. ELEMENT SET is used to apply load and NODE SET used to extract the fitness value. Thus, this model has 10 element sets (SET 1 TO 10) and 1 node set (SET 11). Another two study cases created for this sample model where the different load 'sequence' applied on each case according to the element set. The ABAQUS COMMAND used to run the analysis by running the input file. The result can be viewed from. ODB file and the fitness value is extract from this file. Fig. 3 and Fig. 4 show the load module on specimen model and purple arrow shows the load applies while purple and red is the boundary condition.

The red arrow indicates the set and 10 set created for this specimen. The load apply at every set is to simulate the bolt load effect at every hole. This case study shows different result when the sequence of load applies changed. The 1st sequence load apply is from set 1, set 2, set 3, set 4, set 5, set 6, set 7, set 8, set 9, set 10 and the 2nd sequence load apply is from set 10, set 1, set 9, set 2, set 8, set 3, set 7, set 4, set 6, set 5 shown in Table 2.

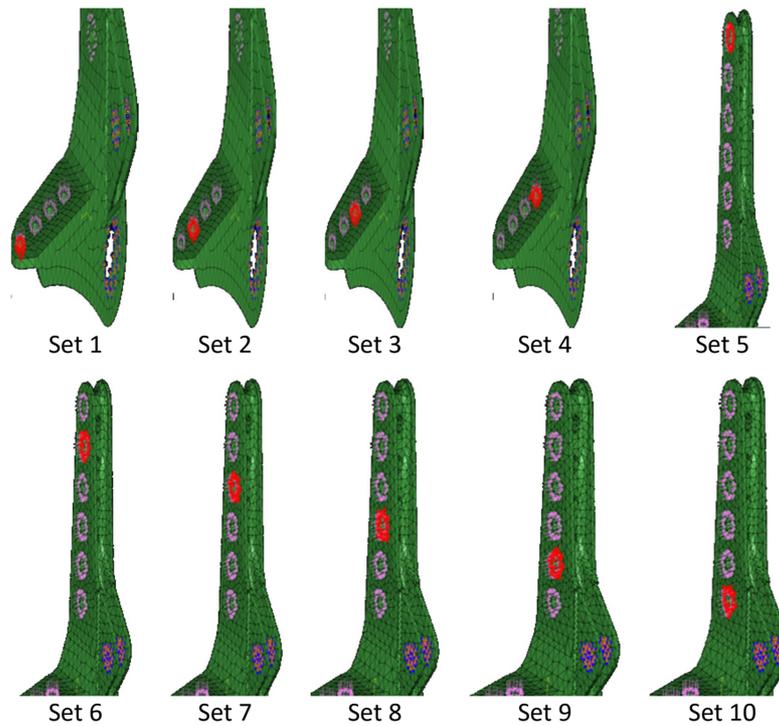


Fig. 3. First sequence

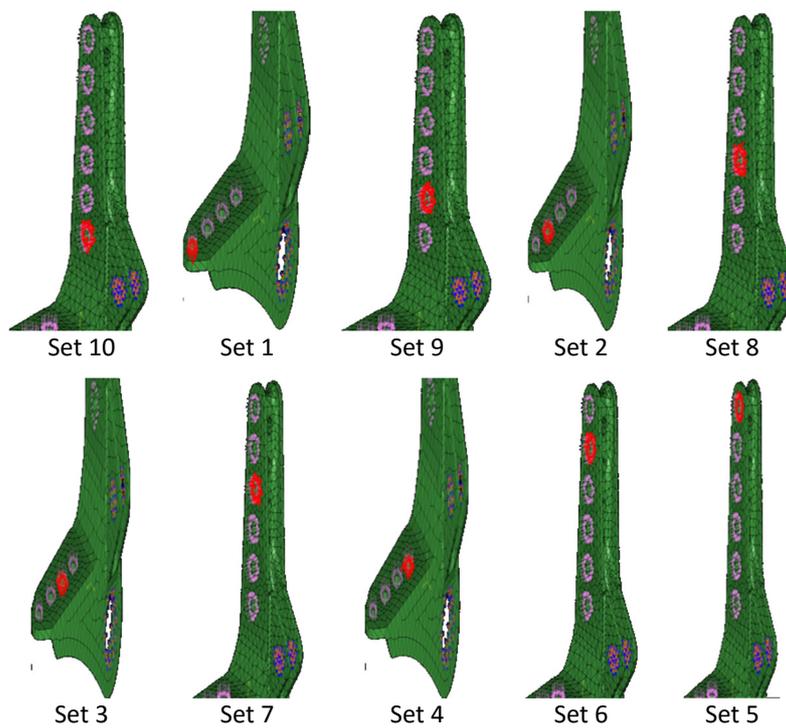


Fig. 4. Second sequence

3. Result analysis

The sequence load apply is done in MATLAB and ABAQUS CAE use to run the analysis and generate *DAT* file, fitness value is view from *DAT* file. The first sequence show set 10 has maximum displacement at node 260. The second sequence show set 5 has maximum displacement at node 260 (Table 3).

Table 2
 Comparison table

Set	Maximum Fitness Value	Node	Maximum Fitness Value	Node
	1 st Sequence		2 nd Sequence	
Set 1	1.3391E-02	454	1.2318E-02	454
Set 2	2.2689E-02	456	2.0186E-02	454
Set 3	2.7633E-02	456	2.3472E-02	456
Set 4	2.9599E-02	456	2.3830E-02	456
Set 5	2.8129E-02	456	5.7427E-02	260
Set 6	3.4679E-02	260	3.8461E-02	1253
Set 7	4.5448E-02	260	2.1864E-02	456
Set 8	5.2617E-02	260	1.8618E-02	454
Set 9	5.6056E-02	260	1.1022E-02	454
Set 10	5.7427E-02	260	4.5557E-03	454

The result is different when the sequence change, this indicates result optimization is needed in order to get accurate result. The maximum fitness value and node point repetition is two data need to be collect in every sequence. This data will determined the specimen behavior when sequence change.

Table 3:
 Node Repeating

Sequence	454	456	260	1245
1 st sequence	1	4	5	0
2 nd sequence	5	3	1	1

4. Conclusion

From this research, it shows that at this stage the optimization of the damage location subject to loading and boundary conditions were done as a baseline data where it runs on several possible sequences using MATLAB programming software. Simulation runs parallel to the developed algorithm to predict the damage locations. Therefore, it can be conclude here this SHM modeling approach may become a first step in developing another series of SHM methods.

References

- [1] National Transportation Safety Board. *Aircraft Accident Report. National Transportation Safety Board*. Public Inquiries Section, RE-51 490 L'Enfant Plaza, S.W. Washington, D.C. 20594, 2007.
- [2] Caicedo, J.M., Dyke, C.J., Johnson, E.A. "Natural excitation technique and eigensystem realization algorithm for phase I of the IASC-ASCE benchmark problem: Simulated data." *Journal of Engineering Mechanics* 130, no. 1 (2004): 49-60.
- [3] Black, C.J., Ventura, C.E. "Blind test on damage detection of a steel frame structure." In *Society for Experimental Mechanics, Inc, 16 th International Modal Analysis Conference.*, vol. 1, pp. 623-629. 1998.
- [4] Yuen, K.-V., Au, S.K., Beck, J.L. "Two-stage structural health monitoring approach for phase I benchmark studies." *Journal of Engineering Mechanics* 130, no. 1 (2004): 16-33.
- [5] Bernal, D., Beck, J. "Preface to the special issue on phase I of the IASC-ASCE structural health monitoring benchmark." *Journal of Engineering Mechanics* 130, no. 1 (2004): 1-2.
- [6] Mal, A., Ricci, F., Banerjee, S., Shih, F. "A conceptual structural health monitoring system based on vibration and wave propagation." *Structural Health Monitoring* 4, no. 3 (2005): 283-293.
- [7] Wong, K.-Y. "Design of a structural health monitoring system for long-span bridges." *Structure and Infrastructure Engineering* 3, no. 2 (2007): 169-185.
- [8] Davidoff, S.M. *Small Deals, Big Issues*. The New York Times, 2009. Retrieved on 2013-02-06.

-
- [9] Hartmann, D., Smarsly, K., Law, K.H. "Coupling sensor-based structural health monitoring with finite element model updating for probabilistic lifetime estimation of wind energy converter structures." In *Proceedings of the 8th international workshop on structural health monitoring*, pp. 13-15. 2011.
- [10] Friswell, M.I., Mottershead, J.E. "Inverse methods in structural health monitoring." In *Key Engineering Materials*, vol. 204, pp. 201-210. Trans Tech Publications, Switzerland, 2001.
- [11] Loh, X.H., Daud, M.A.M., Selamat, M.Z. "Study on Fibre Length and Composition of Kenaf-Polypropylene (K-PP) Composite for Automobile Interior Parts." *Applied Mechanics and Materials* 695 (2014): 36.
- [12] Hind, M.Kh., Özakça, M., Ekmekyapar, T. "A Review on Nonlinear Finite Element Analysis of Reinforced Concrete Beams Retrofitted with Fiber Reinforced Polymers." *Journal of Advanced Research in Applied Mechanics* 22, no 1 (2016): 13-48.