Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

Finite Element Model Updating of Dissimilar Plate with Rivet Joint

To cite this article: H. A. Rahman *et al* 2019 *J. Phys.: Conf. Ser.* **1262** 012035

View the article online for updates and enhancements.

IOP ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

This content was downloaded from IP address 103.53.34.15 on 15/10/2019 at 05:08

IOP Conf. Series: Journal of Physics: Conf. Series **1262** (2019) 012035 doi:10.1088/1742-6596/1262/1/012035

Finite Element Model Updating of Dissimilar Plate with Rivet Joint

H. A. Rahman[1,3], N. D. A. Khairi[1] and M. S. M. Sani[1,2]
[1]Advanced Structural Integrity and Vibration (ASIVR), Faculty of Mechanical Engineering, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.
[2]Automotive Engineering Centre, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.
[3]Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM) Terengganu

[*]Corresponding author: mshahrir@ump.edu.my

Abstract. This study focusses on experimental modal analysis (EMA), finite element analysis (FEA) and model updating. The result of FEA and EMA will be validated and analysed. Then, the study extended to the early stage of model updating by assuming that the riveted dissimilar plates is a rigid body instead of two dissimilar plates. These modal parameters are crucial in providing various periods at which it will naturally resonate to avoid the structure continue to resonate and experience structural damage. Modal parameters of the dissimilar plate with rivet joint which include natural frequency and mode shapes were determined through EMA and FEA. Comparison of experimental data and the simulation data reveals a good correlation between the values of natural frequency. Minor discrepancies of the percentage of error between EMA and FEA are acceptable with the percentage of error is within the range of 0.4% to 13%. Furthermore, the conducted model updating able to reduce to an error range of 0.03% to 0.1%. Through this study, the best way of conducting finite element analysis and model updating for dissimilar plate material with rivet joint is also determined.

1. Introduction

Over the past years, the use of joining few dissimilar materials has increased particularly in the transportation industry, and other high-performance engineering application. Effective 2008, New European anti-pollution and energy saving laws imposed that requirement concerning the reduction of fuel consumption. One approach to partly tackle this issue is to reduce the total weight of the vehicle by using dissimilar materials as part of the structure. A vehicle's main composition, such as steel made chassis module can be joined with secondary structural elements of aluminum alloy materials [1]. Apart from reducing weight, it is also lower the center of gravity of the vehicle. Katayama concluded that development in joining aluminum alloy with other metals shown impressive high-performance structures. In the aircraft industry, weight reduction can also boost payload and increase range [2].

There are many joining technologies used in industry and mechanical fastening is one of the most commonly used joining technologies. This method allows dissimilar materials to be joined together, using only interference or interlocking at a microscopic or macroscopic level. Dursun & Soutis highlighted on the new development in welding, bonding, and extrusion emerged as a new trend in manufacturing and construction industry slowly replacing rivet joint [3]. Nevertheless, rivet joint is still considered as the most practical method to join dissimilar materials. With that, a reliable joining must be developed and evaluated. Over the years, many new developments in joining two dissimilar materials have been developed to fulfill the demand in the industry. Many riveted joints have been introduced in

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd

automotive assemblies and regard as the most effective solution. But even many riveted joints have been widely used in joining dissimilar material; there is no update on the performance data reported in the open literature as a guideline to provide better analysis of a structure.

Rivet joint has been used for years mainly in the automotive industry which imparts a better solution to join dissimilar material such as steel, aluminum, and magnesium [4]. Numerous researches have been done regarding the mechanical properties of the riveted joint. There are three types of structural rivet steel which are ASTM A502 grade 1, carbon rivet steel, ASTM A502 grade 2, high-strength structural steel rivets and ASTM A502 grade 3, similar to grade 2 but with enhanced atmospheric corrosion resistance. On the research discussed by Sun, Stephens, & Khaleel [5] on the effect of Self-Pierce Rivet (SPR) and Spot Resistant Welding (SRW) and concluded that SPR possessed 100% higher fatigue life compared to SRW which this conform to another previous research done by [6] regarding the comparison of fatigue life between SPR and SRW. Further research has been done in comparing two methods of the joint which is rivet joint and welding joint which proved that it rivets joint is more favorable for fixing a steel door impact beam which proved that rivet joint offers a strong joint compared to spot welding joint and more preferably in joining two dissimilar materials especially aluminum and steel as it does not produce brittle material [7].

Finite Element Analysis (FEA) is a method of a numerical technique for finding solutions to structural or performance issues. Basically, it is a branch of solid mechanics that studies the behaviour of solid materials. This method is very handy and convenient for complicated structures with unusual geometric shapes such as trusses, frames, and machine parts. Types of analysis include linear statics which is linear analysis with applied loads and constraints that are static and normal modes which are natural frequencies of vibration. Many types of research have been conducted by using FEA in terms of normal mode analysis[8-13].

A research on experimental investigation and numerical modeling of steel adhesive joint and reinforced by rivets [14] used finite element software to analyze the three types of joint behavior. When the maximum shear strength of the rivet is reached at any point of the rivet, the shearing process starts and finite elements are removed from the model. The numerical results were compared with the experimental data and the result suggested that the strengthening of double lap adhesive joints by rivets introduced by riveting improve both the static strength and the stiffness of joint.

Sani et al. [15] performed an experimental analysis of car chassis to determine the dynamics modal parameter for car chassis. Wira car chassis is drawn in MEscope software then simulated using ME' Scope VES software. The frequency of 40 Hz to 136 Hz is selected for impact hammer test and four mode shapes are obtained from the simulation. Mode shapes obtained shows that the bending occurs on the chassis as in mode 1 when dwelling at the first natural frequency. After that, the twisting occurs on the chassis as in mode 2 when dwelling at the second natural frequency. While in mode 3, dwelling in the third natural frequency produced second bending on the chassis. Lastly, dwelling on the fourth natural frequency displayed second twisting on the chassis on mode 4. On another researcher [16-18] which investigate the finite element analysis and modeling of the structure with bolted joints used shaker or impact hammer vibration in modal testing. From lists of natural frequencies obtained, the author concluded that the bolt models proposed in this study can be employed in a dynamic analysis as well as a static analysis.

This project used finite element analysis (FEA) and experimental modal analysis (EMA) by using impact hammer. These two methods define the dynamic properties of the element and to provide a vibrational analysis data for analysis of dissimilar plate materials with rivet joint. The purpose of this study is to compare and validate the result of modal parameters in dissimilar material using FEA and EMA. Furthermore, this study is extended to model updating so that a feasible model of test structure could be establish. However, this paper only establishes the dissimilar material structure with rivet joints as one rigid body due to the established model updating is at the early stage of this research.

2. Methodology

Figure 1 shows the flowchart of this study. Basically, this research can be summarized into four main phase which is test structure preparation, finite element analysis, model testing and finally, model updating.

2.1. Test Structure Preparation

The test structure was crucially designed at the riveted point of the structure to simulate the joining of two dissimilar material before the final design was selected. Figure 2 illustrates the completed design of riveted structure using SolidWorks 2017 software. The test structure was prepared by undergoing a few processes such as bending, drilling and riveting process. The two dissimilar material properties of the test structure can be shown in Table 1.

Table 1. Material Properties of Test Structure

Material	Young Modulus [GPa]	Density [kg/m^3]	Poisson Ratio
Galvanised Iron	210	7900	0.4
Electro Galvanised	74	3000	0.4

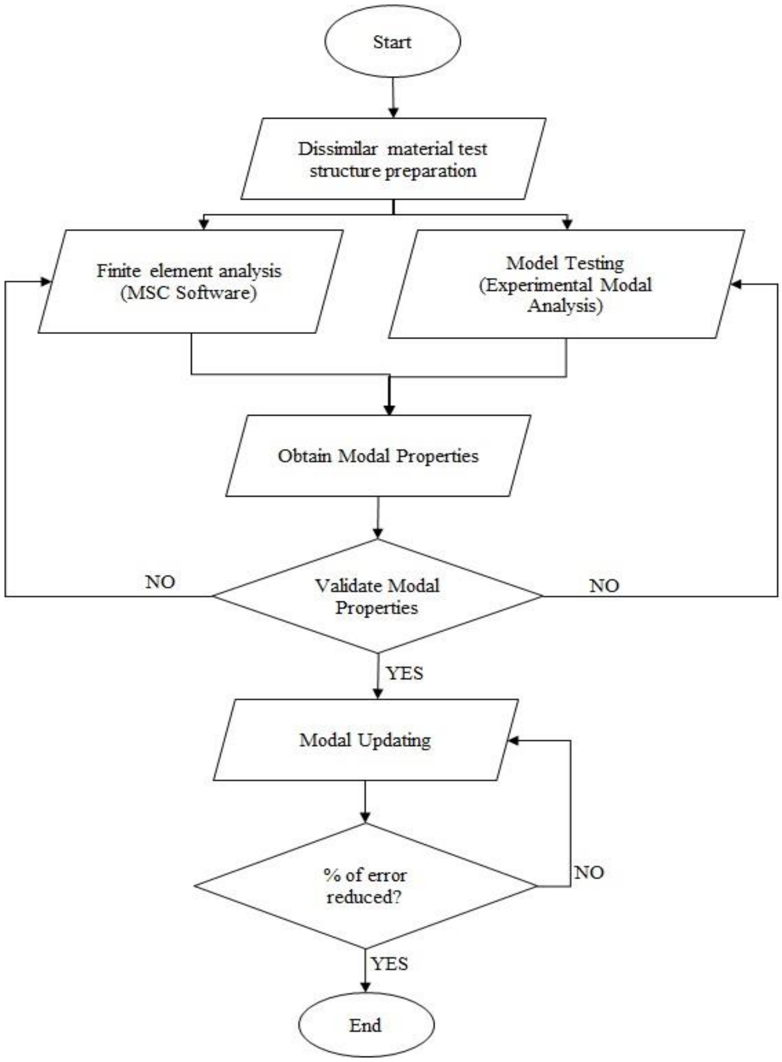


Figure 1. Flowchart of study

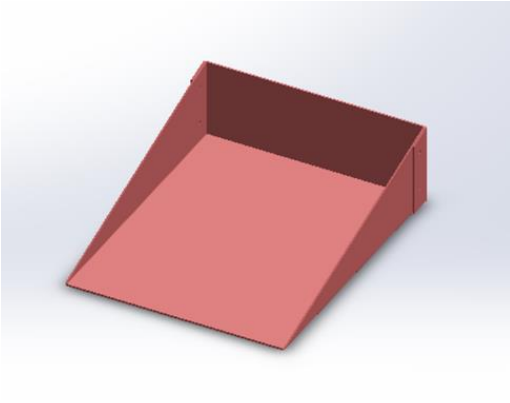


Figure 2. Designed Test Structure

2.2. Finite Element Analysis (FEA) and Experimental Modal Analysis (EMA)

In this project, MSC Nastran Patran software was utilized to perform FEA. The designed test structure was imported to be analyzed and the data or material properties from Table 1 was assigned for the FE model. In this study, a free-free boundary condition was applied and the normal mode analysis SOL103 was performed to obtain modal parameters of the test structure. Initially, FEA was performed earlier than EMA so that the range of natural frequencies and the mode shapes of each mode could be predicted and then compared accordingly to the results from EMA.

In order to obtain the modal parameters experimentally, EMA was performed by using impact hammer and the accelerometer was roved to designated points. The test structure was hanged to simulate free -free boundary condition. Figure 3 shows the schematic diagram of the modal testing and equipment used to perform the measurement.

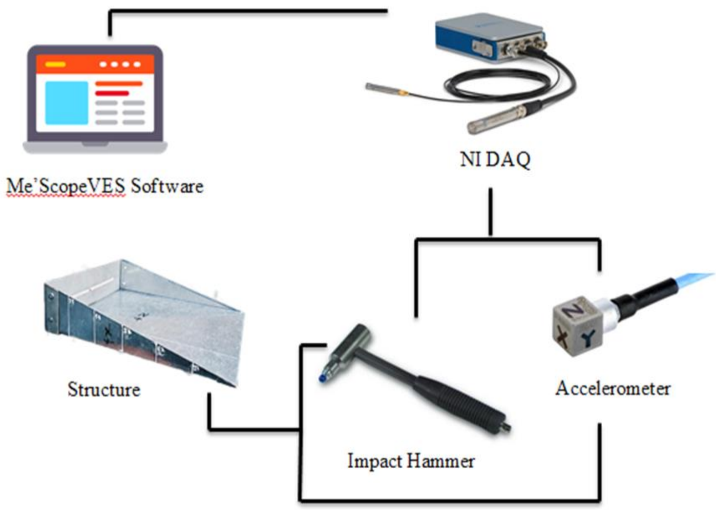


Figure 3. Schematic diagram of modal testing

Initially, the test structure was marked with 55 points and these points was set as measuring points, sketched in a wire-frame model as shown in Figure 4 using modal testing software, ME's Scope VES. The hammer was impacted on one fixed excitation point which is at point 23, while the accelerometer was roved to all 55 measuring points.

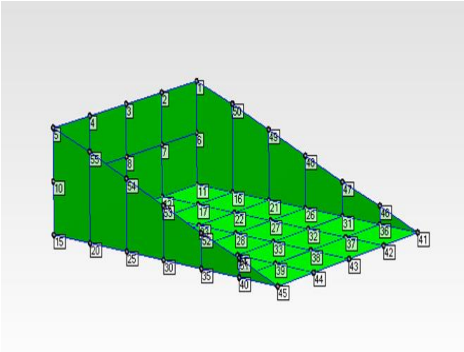


Figure 4. 55 measuring points of test structure

In order to obtained a conclusive experimental data, the signal processing need to be considered in the measuring process. The excitation force from the tip of the impact hammer was set at the magnitude sensitivity of 2.25 mV/g and axis of impact direction was also set accordingly. Piezoelectric tri-axial accelerometer was used to measure the output response of the testing. Then, both excitation force and output response were transferred to the data acquisition system (DAQ). NI 9234 Acoustic and Vibration Data Logger was used as data acquisition system to obtain Frequency Response Function (FRF). FRF was then extracted to obtain the modal parameters which are natural frequency and mode shape using curve fitting method. As a result, 5 modes were successfully extracted from the frequency range of 1 Hz to 1000 Hz. The selected modes were based on predicted results from FEA.

3. Results and Discussion

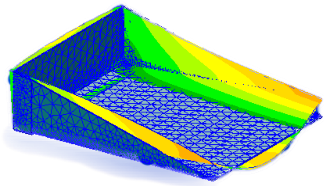
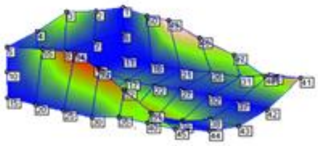
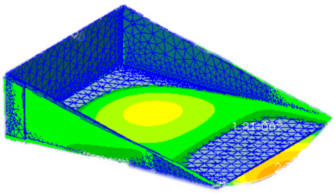
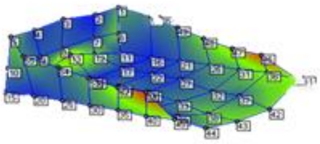
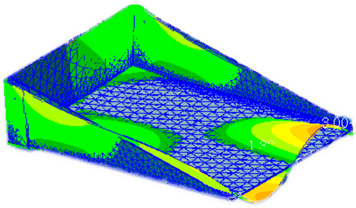
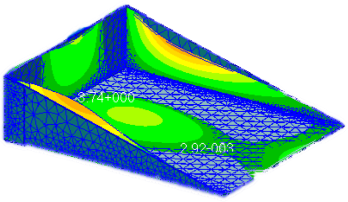
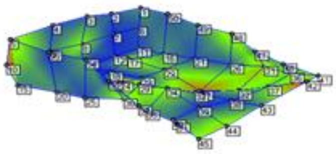
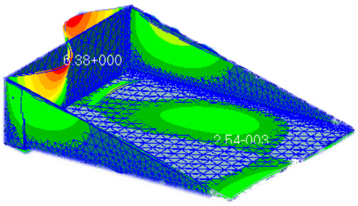
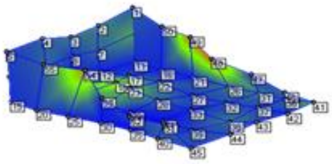
3.1 Correlation Between FEA and EMA

To validate both results from FEA as well as EMA, initially, mode shapes of each selected mode are compared and then the natural frequencies of each mode were also compared. The summarized correlation between FEA and EMA of the test structure can be seen in Table 2 and Table 3. Table 2 shows the correlation of mode shapes between FEA and EMA. Meanwhile, Table 3 shows the correlation of natural frequencies between FEA and EMA.

Table 2. Natural Frequencies Between FEA and EMA

Mode	Natural Frequencies [Hz]		Percentage Error [%]
	EMA	FEA	
1	133	152.74	12.9
2	212	199.26	6.0
3	408	416.49	2.0
4	468	466.28	0.4
5	488	516.75	5.6

Table 3. Mode Shapes Between FEA and EMA

Mode	Mode Shape	
	FEA	EMA
1		
2		
3		
4		
5		

Based on Table 2, percentage of error calculated depicted there is a discrepancy between the result of EMA and FEA. The highest percent of error is Mode 1 which is 12.9 and the lowest is Mode 4 which is 0.4. The other modes show a range of values which less than 10%. The high percent of error can be due to the error which occurred during analysis. While performing the analysis, there are numerous sources of error which can contribute to the discrepancy between FEA and EMA. Source of error in FEA can be categorized into two which are idealization error and discretization error. Idealization error mainly related to the mathematical structure of the model and those that can be improved through model updating [19]. Discretization error can be defined as an error related to element errors and global errors. Apart from the two main source of error, the joining process can also contribute to the error. Rivet joint is seen as the most reliable joining however, there is no established standard practiced in the industry for the riveting process with respect to riveting sequence, the distance between rivets, and the gap between sheets. Joining of the two dissimilar materials might not be perfectly joint due to the misalignment of the hole. The gap between the rivet and the material also contribute to the imperfect joining.

3.2 FE Model Updating

There are discrepancies between the FE or numerical models and modal testing dynamics result from inaccuracy in modelling, approximation of boundary condition, incorrect initial assumptions of geometry and material properties, and limitations of modelling structural connections [19]. Therefore, main objective of FE model updating is to improve a FE model by designing parameters of the model in the light of experimental data to an acceptable of accuracy [20]. Model updating parameter selection is a crucial step in order to avoid ill conditioning problem in optimization. Thus, the sensitivity analysis is required to identify which parameters are sensitive in model updating. The sensitivity analysis performed in this study and represented in Table 4. There are four parameters; Modulus's Young (*E1*) and Density (*ρ1*) for galvanised iron, Modulus's Young (*E2*) and Density (*ρ2*) for electro galvanised iron in sensitivity matrix form.

Table 4. Sensitivity matrix for four parameters

Mode	Young Modulus (E1)	Young Modulus (E2)	Density (ρ1)	Density (ρ2)
1	1.79	69.84	-1.95	-65.18
2	2.03	91.60	-7.41	-80.77
3	2.25	193.10	-4.86	-178.52
4	23.03	196.90	-6.06	-197.82
5	215.59	50.13	-199.10	-38.41

Table 5 shows the updated natural frequency of FE model obtained from output file then compared with the initial value of natural frequency. The discrepancy successfully reduced from initial value 5.38 % of total average error to 5.05 % of total average error.

Table 5. Comparison between initial FEA and updated FEA with measured data from EMA.

Mode	EMA [Hz]	Initial [Hz]	Percentage Error [%]	Updated [Hz]	Percentage Error [%]
1	133	152.74	12.90	147.89	11.20
2	212	199.26	6.00	193.41	8.77
3	408	416.49	2.00	404.08	0.96
4	468	466.28	0.40	449.12	4.03
5	488	516.75	5.60	489.35	0.28
		Average Error	5.38	Average Error	5.05

4. Conclusion

The dynamic characteristic correlation of dissimilar plate with rivet joint are investigated through numerical prediction method via finite element analysis (FEA) software. Experimental modal test was performed to determine the structural response in order to verify the predicted result computed in FEA. In order to reduce the discrepancy between predicted and its measured properties, FE model updating method was applied in this study with sensitive updating parameters. The discrepancy between EMA and FEA successfully minimized from 5.38 % of total average error to be 5.05 % of total average error after model updating process. As conclusion, FE model updating method is a crucial and feasible step in order to produce a reliable FE model in numerical analysis for further engineering analysis.

Acknowledgement
The authors of this paper would like to acknowledge a great support and encouragement by focus group of Advanced Structural Integrity of Vibration Research (ASIVR), Universiti Malaysia Pahang (UMP) for providing all the equipment used for this work Fundamental Research Grant Scheme (FRGS/1/2017/TK03/UMP/02-19) – RDU 170123.

References
[1] Mathieu A, Shabadi R, Deschamps A, Suery M, Matteï S, Grevey D & Cicala E 2007 Dissimilar material joining using laser (aluminum to steel using zinc-based filler wire) *Optics and Laser Technology* 39(3) 652–661.
[2] Katayama S 2004 Laser welding of aluminium alloys and dissimilar metals *Welding International* 18(8) 618–625.
[3] Dursun T & Soutis C 2014 Recent developments in advanced aircraft aluminium alloys *Materials and Design* 56 862–871.
[4] He X, Pearson I & Young K 2008 Self-pierce riveting for sheet materials: State of the art *Journal of Materials Processing Technology* 199(1) 27–36.
[5] Sun X, Stephens E V & Khaleel M A 2007 Fatigue behaviors of self-piercing rivets joining similar and dissimilar sheet metals *International Journal of Fatigue* 29(2) 370–386.
[6] Booth G S, Oliver C A, Westgate S A, Liebracth F and Braunling S B 2000 Self-Piercing Riveted Joints and Resistance Spot Welded Joints in Steel and Aluminium *SAE Technical Paper* 01-2681.
[7] Sakayama T, Murayma G, Naito Y, Saita K, Miyazakki Y, Oikawa H & Nose T 2013 Dissimilar metal joining technologies for steel sheet and aluminum alloy sheet in auto body *NIPPON Steel Technical Report* 393(103) 91–98.
[8] Fouzi M S M, Jelani K M, Nazri N A and Sani M S M 2018 Finite Element Modelling and Updating of Welded Thin-Walled Beam *International Journal of Automotive and Mechanical Engineering (IJAME)* 15(4) 5874-89
[9] Abdullah N A Z, Sani M S M, Hussain N A, Rahman M M and Zaman I 2017 Dynamics properties of a Go-kart chassis structure and its prediction improvement using model updating approach *International Journal of Automotive and Mechanical Engineering (IJAME)* 14 3887-97
[10] Jelani K M, Razip M H M, Nazri N A, Sani M S M and Yasar M 2018 Dynamics investigation on motorcycle chassis based on Finite Element (FE) modelling and updating *IOP Conf. Ser. Mater. Sci. Eng.* 469 012103
[11] Mohd Zin M S, Rani M N A, Yunus M A, Wan Iskandar Mirza W I I, Mat Isa A A and Mohamed Z 2017 Modal and FRF Based Updating Methods for the Investigation of the Dynamic Behaviour of a Plate *Journal of Mechanical Engineering* 4 175–89
[12] Zahari S N, Sani M S M, Husain N A, Ishak M and Zaman I 2016 Dynamic analysis of friction stir welding joints in dissimilar material plate structure *J. Teknol.* 78
[13] Sani M S M, Abdullah N A Z, Zahari S N, Siregar J P and Rahman M M 2016 Finite element model updating of natural fibre reinforced composite structure in structural dynamics *MATEC Web of Conferences* 83 (03007)
[14] Sadowski T, Kneć M & Golewski P 2010 Experimental investigations and numerical modelling of steel adhesive joints reinforced by rivets *International Journal of Adhesion and Adhesives* 30(5) 338–346.
[15] Sani M S M, Rahman M M, Noor M M, Kadirgama K & Izham M H N 2011 Identification of dynamics modal parameter for car chassis *IOP Conference Series: Materials Science and Engineering* 17(1) 012038.
[16] Kim J, Yoon J-C & Kang B-S 2007 Finite element analysis and modeling of structure with bolted joints *Applied Mathematical Modelling* 31(5) 895–911.
[17] Izham M H M, Abdullah N A Z, Zahari S N and Sani M S M 2017 Structural dynamic investigation of frame structure with bolted joints *MATEC Web of Conferences* 90 (01043).

[18] Izham M H M, Sani M S M, Rani M N A and Yunus M A 2019 Damage assessment on frame structure with bolted joints based on experimental modal analysis IOP Conference Series: Materials Science and Engineering 469 (012102)
[19] Mottershead J E, Link M and Friswell M I 2011 The sensitivity method in finite element model updating: A tutorial *Mech. Syst. Signal Process.* 25 2275–96
[20] Fouzi M S M, Sani M S M, and Muchlis Y 2019 "Finite Element Modelling and updating of welded joint for dynamic study of exhaust structure" *IOP Conference Series: Materials Science and Engineering* vol. 469 pp. 1-10