IOP Conference Series: Materials Science and Engineering

PAPER • OPEN ACCESS

Finite Element Modelling and updating of welded joint for dynamic study of exhaust structure

To cite this article: M. S. M. Fouzi *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **469** 012099

Finite Element Modelling and updating of welded joint for dynamic study of exhaust structure

M. S. M. Fouzi[1,3], M. S. M. Sani[1,2*] and Yusrizal Muchlis[4]

[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]Jabatan Kejuruteraan Mekanikal, Politeknik Ungku Omar, Jalan Raja Musa Mahadi, 31400 Ipoh, Perak, Malaysia.
[4]Fakultas TeknikMesin, Universitas Abulyatama, Banda Aceh, Aceh, Indonesia

*Corresponding author: mshahrir@ump.edu.my

Abstract. An exhaust structure is experienced dynamic loads caused by engine operational and road surface condition that affected its durability and dynamic performance. Hence, the purpose of this study is to perform finite element (FE) modelling of exhaust structure and the used of updating approach to improve its dynamic behaviour. Due to its design, exhaust structure is built-up from several parts connected with welded joints. These welded joints significantly contribute to the dynamic behaviour of the structure. Four types of element connector that are RBE2, CBAR, CBEAM and CELAS have been used to replicate FE model of welded joint on the structure. Modal parameters (natural frequency and mode shape) of the FE model have been obtained from normal mode analysis using finite element analysis (FEA) software, MSC. Nastran/Patran. The precision of numerical predicted result from FEA is compared with its measured counterpart. The measured test data obtained through experimental modal analysis (EMA) using impact hammer and roving accelerometers under free-free boundary conditions. Under correlation process, CBAR element connector was chosen to model the welded joint due to its accurate prediction of natural frequency and contains updating parameters. FE model updating process was performed to improve the correlation between EMA and FEA. Ahead of updating process, sensitivity analysis was done to select the most sensitive updating parameter. As a result, total percentage error of natural frequency for updated CBAR model is reduced significantly from 8.74 % to 3.45 %. Consequently, CBAR element connector was chosen as the most reliable joint element in FE model to represent welded joint on exhaust structure.

1. Introduction

An exhaust system is one of the essential sub-systems in each vehicle power-driven by internal combustion engine. The main function of exhaust system is to filter the hazardous gaseous and lowered the noise produced by combustion process in internal combustion engine before it expelled into surrounding to avoid pollution. The structure is tending to experience the vibration from engine operational itself and high pressure gaseous that propagated along the exhaust pipe. In addition, the uneven road surface contributes the dynamic load which air-borne through the exhaust hanger to the vehicle's chassis. Consequently, both structure-borne and air-borne vibration will affect the comfort or specifically known as noise, vibration and harshness (NVH) for the vehicle. Currently, NVH happen to be one of important criteria among end-user when selection of the vehicle to be made. Reported by previous study that source of the loads came from engine operational and road surface condition [1-5]. In recent years, there has been an increasing interest in dynamic study on exhaust structure as reported in the previous researches [1-4, 6-8]. Despite of dynamic study on exhaust structure, the previous works still lack of information of joining strategy to model and update the welded joints on FE model to replicate the physical structure as real as possible. Thus, the motivation arises for this study is to identify the dynamic behaviour of welded exhaust structure through numerical prediction technique via FEA and verified with its experimental counterpart by EMA.

Usually, exhaust structure is fabricated from few assembled parts such as catalytic converter, extension pipe, resonator, muffler and tailpipe, which been connected each other's by weld joints. Due to its real design, the necessity of joint strategy in FE modelling of exhaust structure in this study applied and briefly expressed in section 2.1. This approach is endeavoured by previous work that reported the behaviour of friction stir welding (FSW) joints plays a significant role in the dynamic characteristics of the structure due to its complexities and uncertainties therefore the representation of an accurate FE model of these joint become a research issue [9]. In others study, FE modelling and updating is carried out for frame structure with bolted joint [10].

However, the FE model frequently has been question on its trustworthiness since it was a numerical prediction solved using computational method on discrete model. Regarding [11], there are three commonly encountered forms of model error which may lead to inaccuracy in the model predictions: (i) model structure errors, (ii) model parameters errors, and model order errors. Hence, FE model updating is required to encounter this hitch. For instance, [12] has perform model updating of a go-kart chassis structure in order to reduce the percentage of error between the EMA and FEA. In addition, there is another motivation for updating the FE models that is an evaluation of health and integrity of the structure and structural health monitoring (SHM) [13].

Prior to updating process of FE model, modal testing process has been carried out first to extract the modal parameters from real structure. Modal testing or recognized as modal analysis is a process whereby we describe a structure in terms of its natural characteristics which are the frequency, damping and mode shape – it's dynamic properties [14]. The modal testing procedure adopted in this work described in section 3. Once the measured test data prepared, the correlation process is undergoing to see how far the FE model meet its real structure counterpart. Correlation is a process of comparing the data from FEA to the EMA and accessing how far that they are in agreement with each other [12]. This process concisely defined in section 4.

Finally, the updating process used in this work to update the FE model of the exhaust structure. In updating process, it is not just simply update the FE model straightforward by using the measured data. Ahead of this procedure is needs the sensitivity analysis method. The sensitivity method is probably the most successful of the many approaches to the problem of updating finite element models of engineering structures based on vibration test data [15]. Updating process of FE model in this research briefly explained in section 5. As a result, the discrepancy between FEA and EMA successfully reduced and the most reliable joint element of FE model of exhaust structure has been accomplished to be modelled.

2. Illustration of FE modelling for exhaust structure and joint strategy

In FEA, it has become an essential to have FE model ahead before running the analysis. Sometimes, researchers and engineers faced difficulty to sketch directly the FE model in FEA software due to complexity of structure's geometry. For this reason, the CAD software used in this work to model the structure before imported into FEA software. There have been several studies in the literature reporting this kind of approach. In their work, the exhaust system was modelled using CATIA V-5 software and been imported into FEA software, Hypermesh [2, 7, 16, 17].

In this study, the FE model is initially prepared using CAD software, SolidWork as depicted in Figure 1 and been imported and pre-processing in FEA package, MSC Patran as depicted in Figure 2. The CAD file saved as *parasolid* format before been imported into FEA software. Once the FE model imported into FEA package, it has been meshed using solid element (CTETRA4) topology. The FE model of exhaust structure consists of a total of 137043 elements and 46548 nodes. The material properties assigned into the FE model as followed; Modulus's Young (E) = 195 GPa, density, (ρ) = 8000 kg/m^3, and Poisson's Ratio (ν) = 0.29. Once the joint strategy successful applied in the FE model as concisely explained in section 2.1, the normal mode analysis executed using MSC Nastran solver to extract the modal parameters (natural frequency and mode shape). These modal parameters from FEA are summarized in Table 1 for mode shapes and Table 2 for natural frequencies. Both natural frequency and mode shape is compared with measured test data from EMA.

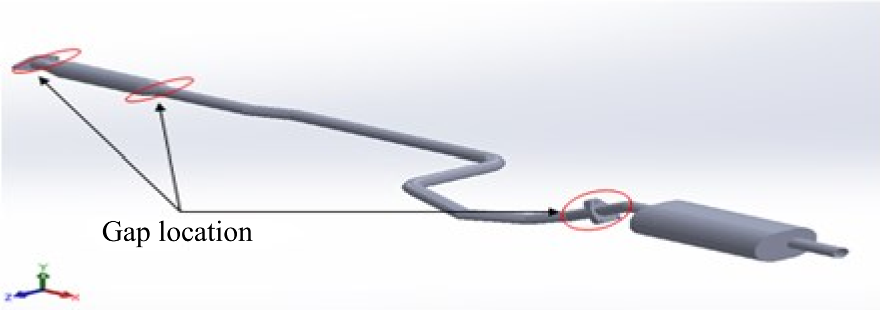


Figure 1. 3D model of exhaust structure designed by CAD software

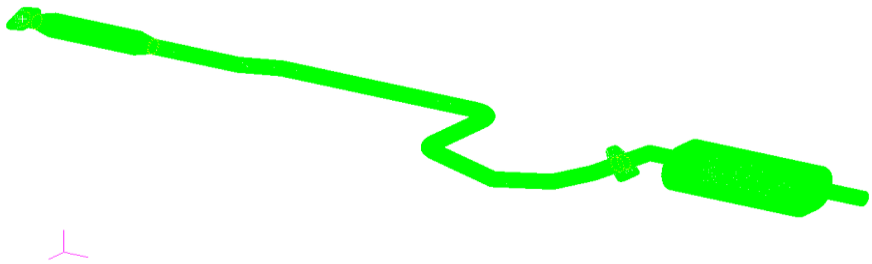


Figure 2. FE model of exhaust structure imported into FEA package

2.1. Welded Joints Modelling

In the purpose of replication, the weld joint model to its real counterpart, the implementation of numerical joint strategy applied in this research. This is because the integrity and dynamic behaviour of the structure is highly dependents on the joint [18]. RBE2 is a rigid element connected to an arbitrary number of grid points. The independent DOFs are the six components of motion at a single grid point. The dependent DOFs at the other grid points all have the same component numbers. In this study, RBE2 was modelled such in Figure 3.

A CBAR element is a straight prismatic element and provides axial, torsional, bending stiffness in two perpendicular planes and shear stiffness in two perpendicular planes, hence providing stiffness in all six DOFs on either grid such as Figure 4. CBEAM element includes the CBAR element capabilities and separate neutral axis and axis of shear centres, effect of cross-sectional warping on torsional stiffness (as is important in the case of open sections), cross-sectional properties specified on both ends and interior point (tapered element), effect of taper on traverse shear stiffness (shear relief), separate axis for the centre of non-structural mass and torsional inertias are included i.e. can be offset from shear centre (for dynamic analysis) and also has nonzero rotational mass moment of inertia about its neutral axis. CBEAM element was modelled like Figure 6.

CELAS element is used to model springs (that provide translational or rotational stiffness) connecting single DOFs at two grid points that should be coincident such illustrated in Figure 5.

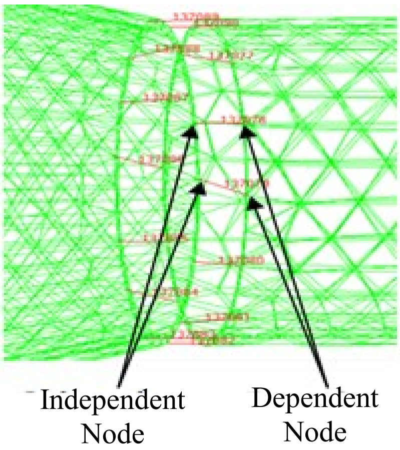


Figure 3. RBE2 element connector

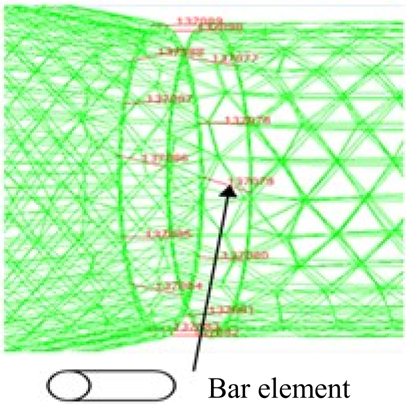


Figure 4. CBAR element connector

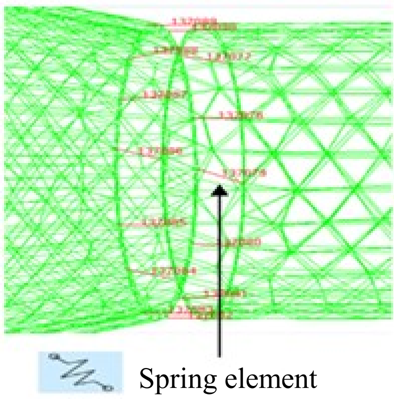


Figure 5. CELAS element connector

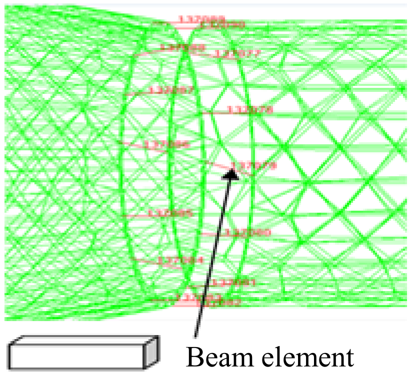


Figure 6. CBEAM element connector

All the element connector RBE2, CBAR, CELAS, and CBEAM modelled at the gap location as showed in Figure 1 to replicate the welded joint. The gap distance on FE model is about 0.003 meter. For every FE analysis, 80 number elements of connector applied on the FE structure. For instance, 80 elements of RBE2, 80 elements of CBAR, 80 elements of CBEAM and 80 elements of CELAS.

3. Description of Modal Testing of Structure

Modal testing or normally known as experimental modal testing (EMA) is used to identify the dynamic behaviour of the structure. Since the very early days of awareness of vibrations, experimental observations have been made for the two major objectives; (a) determining the nature and extent of vibration response levels and (b) verifying theoretical models and predictions [19]. Thus, in this study modal testing data used to verify the numerical prediction result from the FEA. Regarding [20], experimental modal analysis has grown steadily in popularity since the advent of the digital Fast Fourier Transform (FFT) spectrum analyzer in the early 1970's and become well-known technique in modal testing. The testing performed to obtain the modal parameters of the test structure. These modal parameters are estimation from frequency response functions (FRFs) which use excitation input and the corresponding output of the test structure [21].

The testing for this study is running as depicted in Figure 7 by hanging the test structure with bungee cords to replicate the free-free boundary conditions. Measurement process carried out with the assistance of EMA equipments like portrayed in Figure 8. The impacts hammer excitation technique adopted for measurement process in this research since it fast, convenient, and very useful for quick diagnostics as claimed by [21]. Initially, the test structure modelled as wireframe in modal analysis software as portrayed in Figure 9. The test labelled with 66 measurement points. Two tri-axial accelerometers used and roving accelerometer method applied to this measurement. Roving accelerometer method means the output response measured by moving the accelerometer on each measurement points while there only one fixed excitation point in the test structure.

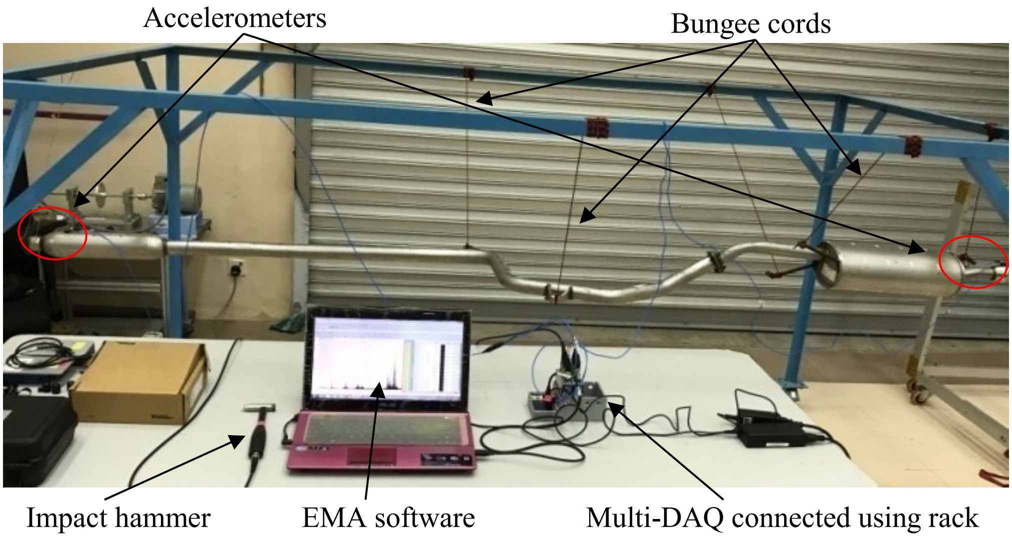
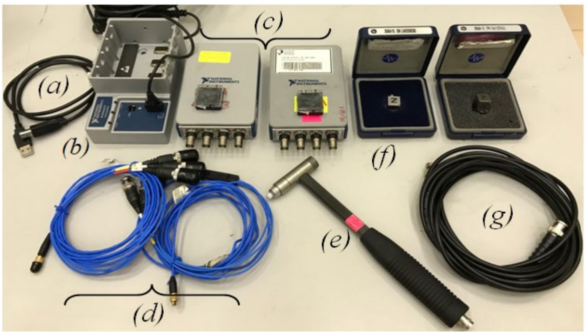


Figure 7. Execution modal testing of free-free boundary condition test structure



(a) USB cable
(b) NI-DAQ rack
(c) NI-DAQ
(d) Accelerometer cable
(e) Impact hammer
(f) Tri-axial accelerometer
(g) BNC cable

Figure 8. Equipments for modal testing

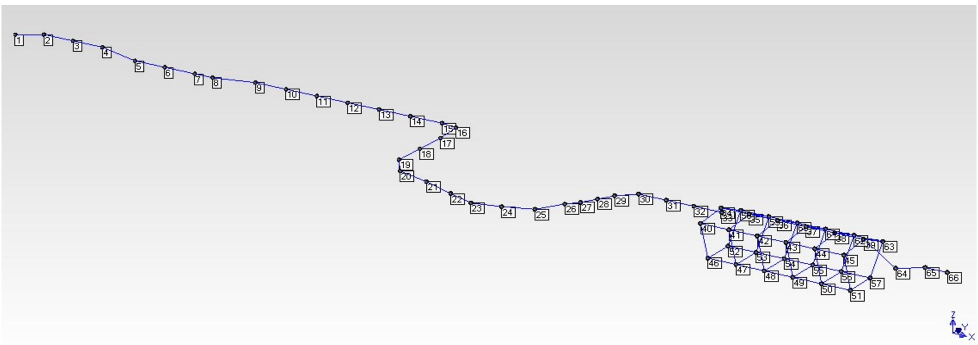


Figure 9. Wireframe structure in EMA software with 66 measurement points

4. Correlation between numerical result and measured test data

In the mean of trustworthiness of FE model, the numerical prediction result from FEA validated with measured test data through correlation process. Correlation was conducted in order to analyze the discrepancies existed between FEA and EMA [22]. As tabulated in Table 1, the correlation of mode shape from numerical prediction and its measured counterpart is draw together. The mode shape for first to sixth mode of FEA for each type of element connector compared with mode shape from EMA and showed close pattern except CELAS.

While in Table 2, the correlation of natural frequencies of FE model for exhaust structure compared with measured data from EMA. The discrepancy between numerical prediction result and measured test data calculated as shown in equation (1);

$$Percentage\ of\ error = \left|\frac{\lambda^{FEA} - \lambda^{EMA}}{\lambda^{EMA}}\right| \times 100 \tag{1}$$

where λ^{FEA} is natural frequency obtained from finite element analysis (FEA) method while λ^{EMA} is natural frequency value extracted from experimental modal analysis (EMA) technique. From the comparison made in Table 2, RBE2 element connector has the lowest value percentage of error from its measured counterpart, 6.98 % compared with CBAR 7.38 %, CBEAM 8.07 %, and CELAS 9.82 %. However, CBAR chosen for updating process since RBE2 did not contain any updating parameter because RBE2 is rigid body element. CBAR model has showed good agreement compared to others two elements; CBEAM and CELAS.

Table 1. Correlation of mode shapes between EMA, FEA and joint strategy elements.

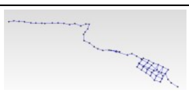
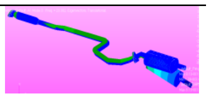
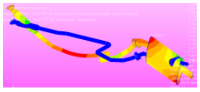
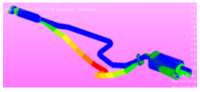
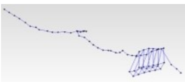
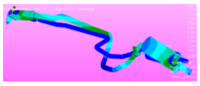
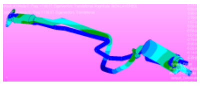
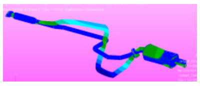
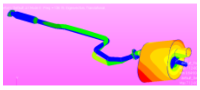
Mode	EMA	RBE2	CBAR	CBEAM	CELAS
1	 21.1 Hz	 18.322 Hz	 18.288 Hz	 18.245 Hz	 23.382 Hz
2	 46.9 Hz	 43.706 Hz	 43.548 Hz	 43.298 Hz	 52.731 Hz
3	 65.1 Hz	 57.413 Hz	 57.103 Hz	 56.550 Hz	 60.940 Hz
4	 76.3 Hz	 72.482 Hz	 72.180 Hz	 71.630 Hz	 76.232 Hz
5	 108 Hz	 102.57 Hz	 102.00 Hz	 100.87 Hz	 127.97 Hz
6	 123 Hz	 122.95 Hz	 122.34 Hz	 121.37 Hz	 136.15 Hz

Table 2. Correlation of Natural Frequency between EMA and FEA with joint strategies

Mode	Natural Frequency (Hz)								
	EMA	RBE2 Element	*(Error %)*	CBAR Element	*(Error %)*	CBEAM Element	*(Error %)*	CELAS Element	*(Error %)*
1	21.1	18.32	13.17	18.29	13.33	18.25	13.53	23.38	10.82
2	46.9	43.71	6.81	43.55	7.15	43.30	7.68	52.73	12.43
3	65.1	57.41	11.81	57.10	12.28	56.55	13.13	60.94	6.39
4	76.3	72.48	5.00	72.18	5.40	71.63	6.12	76.23	0.09
5	108	102.57	5.03	102.00	5.56	100.87	6.60	127.97	18.49
6	123	122.95	0.04	122.34	0.54	121.37	1.33	136.15	10.69
Total Average Error			6.98		7.38		8.07		9.82

5. FE Model Updating of Exhaust Structure

There may be differences between the FE models and real structures resulting from inaccuracy in modelling and various uncertainties, including approximation of boundary condition, incorrect initial assumptions of geometry and material properties, and limitations of modelling structural connections [13]. Thus, primary aim of FE model updating is to reconcile a FE model by improving the design parameters of the model in the light of experimental data to an acceptable of accuracy [18]. Selection of the updating parameter is a vital aspect of the FE model updating process such as geometric and material properties of test structure. However, the parameters selected should be kept to a minimum to avoid-ill conditioning problem [23]. Consequently, the sensitivity analysis is necessary to run in advance, so that only the most sensitive parameter will be chose. The sensitivity analysis performed in this study and represented in Table 3. There are three parameters; Modulus's Young (E), Density (ρ), and Poisson's Ratio (v) in sensitivity matrix form. From Table 3, the most sensitive parameter is
Density (ρ) and followed by Modulus's Young (E). The positive or negative signs just indicated the direction of the vector not to parameter value. In addition, obviously the less sensitive parameter in this study as tabulated in Table 3 is Poisson's Ratio (v).

Table 3. Sensitivity matrix for three parameters

Mode	Modulus's Young (E)	Density (ρ)	Poisson's Ratio (v)
1	9.0550	- 10.489	- 0.5972
2	21.429	- 24.882	- 0.2693
3	28.077	- 32.757	- 2.6236
4	35.493	- 41.318	- 1.5457
5	50.060	- 58.512	- 4.0707
6	60.188	- 70.082	- 4.3396

5.1. Objective Function

An objective function based on residuals between the experimental modal data (i.e. natural frequencies, mode shape, etc.) and its numerical prediction is set for minimisation in the updating procedure. The procedure continues until convergence accomplished when the difference between values of the objective function (J) from consecutive iterations is sufficiently small.

$$J = \sum_{j=1}^{n} w_j = \left(\frac{\lambda_j}{\lambda_j^{exp}} - 1 \right)^2 \tag{2}$$

From (2), w_j is a weighting coefficient for each mode while λ_j^{exp} is the jth experimental eigenvalue and λ_j is the jth eigenvalue predicted in numerical prediction process. The optimization process applied using optimization algorithm in MSC Nastran and result obtained from the process tabulated in Table 4. There is comparison between initial and updated of CBAR model for five modes. The discrepancy between FEA and EMA is reduced significantly with the reduction of percentage of error from 8.74 % to 3.45 %. The new design variable for updated parameters is displayed in Table 5.

Table 4. Correlation of Natural Frequency between EMA with Initial and Updated FE with CBAR element connector

Mode	Natural Frequency (Hz)				
	EMA	Initial CBAR Element	*(Error %)*	Updated CBAR Element	*(Error %)*
1	21.1	18.29	13.33	20.03	5.09
2	46.9	43.55	7.15	47.56	1.41
3	65.1	57.10	12.28	62.58	3.86
4	76.3	72.18	5.40	78.92	3.43
5	108	102.00	5.56	111.71	3.44
Total Average Error			8.74		3.45

New design variable of updated parameters are 1.0963 for Modulus's Young (E), 0.91923 for density (ρ) and 0.89971 for Poisson's Ratio (v) from the initial value. The deviation of new design variable from its initial value calculated as equation (3);

$$Deviation, \Pi = \left|\frac{\alpha - \beta}{\beta}\right| \tag{3}$$

Table 5. Updated parameters value

Parameters	Initial Value (α)	Updated Value (β)	S.I. Unit	Deviation (Π)
Modulus's Young (E)	195	213.78	GPa	0.09
Density (ρ)	8000	7353.84	kg/m^3	0.09
Poisson's Ratio (v)	0.29	0.261	-	0.11

6. Conclusion

As a conclusion, this study is presented the FE modelling approach of welded joint for numerical prediction of dynamic behaviour for exhaust structure. Four types of element connectors; RBE2, CBAR, CBEAM, and CELAS have been adopted to replicate welded joint. The precision of these joints modelling has been evaluated through correlation process using natural frequency and mode shape data obtained from FEA and EMA. Correlation process has been revealed that CBAR model showed outstanding capability to represent welded joint model as precise as real structure compared to other three weld joint models. CBAR model also containing updating parameter compared to RBE2 model that did not has any updating parameter since it represent rigid body element. Then, model updating has been implemented to improve the correlation between numerical prediction and its measured counterpart. Ahead of updating process, sensitivity analysis has been carried out to choose the most sensitive parameters only for the process. Consequently, updated CBAR model has showed significant improvement from its initial result; with apparently reduction of percentage of error between predictions result with its measured data.

Acknowledgments
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 equipmentused for this work Fundamental Research Grant Scheme (FRGS/1/2017/TK03/UMP/02-19) – RDU 170123. Special thanks for Ministry of Education (MOE) under Higher Education and JabatanPendidikanPoliteknik (JPP) for financial assistance, support and study leave via HadiahLatihan Persekutuan (HLP) scholarship.

References
[1] Waghmare V S, Salunkhe V, Nagrani R, and Guttal V 2017 "Analysis of exhaust system mounting scheme" *SAE International Journal of Vehicle Dynamics, Stability, and NVH* vol. 1 pp. 89-96 doi:10.4271/2017-26-0291
[2] Shojaeifard M H, Ebrahimi-Nejad R, and Kamarkhani S 2017 "Optimization of exhaust system hangers for reduction of vehicle cabin vibrations" *International Journal of Automotive Engineering* vol. 7 pp. 2314-2324 doi:10.22068/ijae.7.1.2314
[3] Rajadurai M S, Kavin R, Rejinjose, Prabhakaran, and Rajeshraman 2016 "A system approach to dynamic characteristics of hanger rod in exhaust system" *International Journal of Innovative Science, Engineering & Technology* vol. 3
[4] Pan G Y and Cao D Q 2016 "Vibration characteristics analysis of exhaust system based on optimization of hanger position" *4th International Conference on Sustainable Energy and Enviromental Engineering (ICSEEE 2015)*
[5] Wenzhu W, Gang L, Mianhong C, and Jun W 2015 "Hanger location layout of automotive exhaust system" doi:10.4271/2015-01-2318
[6] Wei Z, Das S, Barr R, Rohrs G, Rebandt R, Wu X*, et al.* 2017 "Development of lightweight hanger rods for vehicle exhaust applications" doi:10.4271/2017-01-1709
[7] Vinutha B S and Hemanth K 2017 "Study of static and frequency responsible analysis of hangers with exhaust system" *IJASRE* vol. 3
[8] Yang Y, Yang Y, Sun Y, Dong Z, and Zhang Y 2016 "Optimization of vehicle exhaust system hanger location" doi:10.4271/2016-01-0448
[9] Zahari S N, Sani M S M, and Ishak M 2017 "Finite element modelling and updating of friction stir welding (FSW) joint for vibration analysis" *MATEC Web of Conferences* vol. 90 doi:10.1051/ matecconf/20179001021
[10] Izham M H N, 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* vol. 90 p. 01043 doi:10.1051/ matecconf/20179001043
[11] Mottershead J E and Friswell M I 1993 "Model updating in structural dynamics: a survey" *Journal of sound and vibration* vol. 167 pp. 347-375
[12] Abdullah N A Z, Sani M S M, Husain 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)* vol. 14 pp. 3887-3897 doi:10.15282/ijame.14.1.2017.6.0316
[13] Sarmadi H, Karamodin A, and Entezami A 2016 "A new iterative model updating technique based on least squares minimal residual method using measured modal data" *Applied Mathematical Modelling* vol. 40 pp. 10323-10341 doi:10.1016/j.apm.2016.07.015
[14] Avitable P 2000 "Experimental modal analysis (a simple non-mathematical presentation)" *Annual Technical Meeting-Institute of Environmental Sciences And Technology* vol. 46 pp. 434-448
[15] Mottershead J E, Link M, and Friswell M I 2011 "The sensitivity method in finite element model updating: A tutorial" *Mechanical systems and signal processing* vol. 25 pp. 2275-2296 doi:10.1016/j.ymssp.2010.10.012
[16] Kumbhar V and Patil D C 2017 "Structural optimization & durability analysis of VW Bettle cross exhaust muffler system" *International Research Journal of Engineering and Technology* vol. 04

[17] Vaddankeri N, Udupa N G S, and Venkategowda 2016 "Stress analysis of exhaust system in automobiles" *International Journal for Scientific Research & Development* vol. 4

[18] Rani M N A, Kasolang S, Othman M H, Yunus M A, Mirza W I I W I, and Ouyang H 2016 "Finite element modelling and modal based updating for the dynamic behaviour of a laser spot welded structure" *ICSV 2016-23rd International Congress on Sound and Vibration: From Ancient to Modern Acoustics* pp. 1-8

[19] Ewins D J 1984 "Modal testing: theory and practice" vol. 15

[20] Schwarz B J and Richardson M H 1999 "Experimental Modal Analysis" *CSI Reliability Week, Orlando, FL*

[21] Carne T G and Stasiunas E C 2006 "Lessons learned in modal testing—part 3: Transient excitation for modal testing, more than just hammer impacts" *Experimental Techniques* vol. 30 pp. 69-79 doi:10.1111/j.1747-1567.2006.00046.x

[22] Abdullah N A Z, Sani M S M, Rahman M M, and Zaman I 2017 "Correlation of numerical and experimental analysis for dynamic behaviour of a body-in-white (BIW) structure" *MATEC Web of Conferences* vol. 90 p. 01020 doi:10.1051/matecconf/20179001020

[23] Husain N A, Khodaparast H H, and Ouyang H 2010 "FE model updating of welded structures for identification of defects" *International Journal of Vehicle Noise and Vibration* vol. 6 pp. 163-175