SEMARAK ILMU
PUBLISHING
202103268166(003316878-P)

Journal of Advanced Research in Applied Mechanics

Journal homepage:
https://semarakilmu.com.my/journals/index.php/appl_mech/index
ISSN: 2289-7895

Enhancing Dynamic Behaviour of Exhaust Structure by Reconfiguration the Hanger Locations using Updated FE Model with Joint Modelling Strategy

Mohd Sahril Mohd Fouzi[1,2], Mohd Shahrir Mohd Sani[1,3,*], Meifal Rusli[4]

[1] Faculty of Mechanical and Automotive Engineering Technology, Universiti Malaysia Pahang Al Sultan Abdullah (UMPSA), 26600 Pekan, Pahang, Malaysia
[2] Kolej Komuniti Bagan Datuk, Lorong Lagenda 2, Medaniaga Lagenda, 36400 Hutan Melintang, Perak, Malaysia
[3] Automotive Engineering Centre, Universiti Malaysia Pahang Al Sultan Abdullah (UMPSA), 26600 Pekan, Pahang, Malaysia
[4] Mechanical Engineering Department, Faculty of Engineering, Andalas University, Kampus Unand Limau Manis, Padang 25163, Indonesia

ARTICLE INFO

Article history:
Received 23 October 2024
Received in revised form 24 November 2024
Accepted 1 December 2024
Available online 30 December 2024

Keywords:
Numerical analysis; joint modelling strategy; hanger location; exhaust; FE model updating

ABSTRACT

This paper presents numerical analysis study in enhancing the dynamic behaviour of exhaust structure through reconfiguration the hanger locations using reliable Finite Element (FE) model. The main objective of this study is to reduce the original amplitude (displacement) of test structure by relocating the hanger position. Initially, the test structure has been modelled with joint modelling strategy using existed element connectors in Finite Element Analysis (FEA) package, MSC. Patran such as CELAS, CBAR, CBEAM, and RBE2 to replicate welded joint in the FE model. Since FEA is a numerical prediction method which used the assumption and simplification during pre-processing stage, thus it's required to be validated with its measured test data through correlation process. This measured test data was acquired from Experimental Modal Analysis (EMA). Validation process between FEA and EMA showed that CBAR element connector is feasible to replicate welded joint in the FE model with 4.10 % of percentage error compared to the other element connectors. This FE model with CBAR element connector then treated with FE model updating technique to improve the agreement between numerical prediction result with its measured test data. The FE model updating technique has been implemented to alter certain dynamic properties with the assistance of design optimisation SOL200 function which available in MSC. Nastran. The discrepancy between numerical prediction and its measured test data successfully reduced from 4.10 % to 3.74 %. Next, the updated FE model has been used in final action of this study which is reconfiguration of hanger locations in enhancing dynamic behaviour of test structure. There are about 35 case studies with different configuration of hanger locations were analysed using modal frequency response analysis SOL111 in MSC. Nastran. From this analysis it showed that case study no. 9 has the smallest displacement with 0.77 mm compared to the others case study while its original hanger location has recorded 17 mm of displacement. This proposed method in this study is feasible to be implemented for different type of exhaust structure since it's more economic compared to field testing method which consumed more time, effort, and expenditure.

* *Corresponding author.*
E-mail address: mshahrir@ump.edu.my

https://doi.org/10.37934/aram.130.1.115

1. Introduction

As expected in the current automotive industry, automotive players are expressing intentions to replace conventional engines like gasoline and diesel with hybrid and electric motors. This shift aims to address the impeding crude oil shortage and global warming concerns. However, these efforts have been relatively negligible in recent years due to the greater quantity of conventional cars powered by internal combustion engines (ICE) sold compared to hybrid and electric vehicles. The reason behind this trend is the higher maintenance costs associated with hybrid vehicles, while electric vehicles face the challenge of insufficient charging facilities, preferred and relevant choice among buyers in the country.

In typical practice, every vehicle equipped with an ICE is constructed with an exhaust system. This system plays a crucial role as a sub-system within the vehicle, serving to filter harmful gaseous generated during the combustion process in the ICE and reduce unwanted noise before it disperses into the surroundings [1]. Attaching the exhaust system directly to the vehicle body is not feasible, as it would result in the transmission of vibrations to the cabin [2]. These vibrations originate from dynamic loads produced by the operating ICE itself and the unevenness of the road surface [3-6]. Unintentional vibration transfer can have a significant impact on the quality of noise, vibration, and harshness (NVH) experienced by the vehicle, as noise and vibrations within the passenger cabin can reduce comfort and lead to feelings of stress, fatigue, and insecurity [7]. To address this issue, the exhaust system is typically designed to be mounted on the vehicle’s body, usually underneath the chassis, using flexible hangers.

Because the hangers act as clamping devices to secure the exhaust structure to the vehicle’s body, NVH engineers must optimise their placement to minimise damage and increase durability [8]. As a result, modal analysis is a reliable method for understanding the dynamic behaviour of the exhaust structure at various natural frequencies by evaluating its mode forms [9]. Modal analysis can be performed numerically using the Finite Element Analysis (FEA) method and validated experimentally using experimental modal analysis (EMA), which entails measuring the actual response of the system.

Finite Element Analysis (FEA) is a vital technique in the engineering world, with multiple applications such as structural analysis, dynamic behaviour prediction, structural condition assessment, structural health monitoring (SHM), and damage detection [10]. FEA has been used in several research to perform modal analysis and explore the dynamic behaviour of test structures [11-15]. Various FE modelling strategies were adopted in their work [12], to accurately forecast the dynamic properties of sheet metal connections formed by friction stir welding (FSW). Due to their complexities and uncertainties, the behaviour of FSW joints has a substantial impact on the dynamic properties of the structure, emphasising the need of appropriately describing these joints in the FE model. To simulate jointed structures, existing element connectors in the FEA programme can be used during pre-processing stage. As proven by [11] in modelling welded thin-walled beams, such connectors include the rigid body element type 2 (RBE2), bar element (CBAR), beam element (CBEAM), and spring element (CELAS). Since the FEA data is based on numerical predictions created by computers, modal testing, also known as correlation, is used to verify and confirm the predicted data’s accuracy versus observed findings [16].

Verified results from numerical predictions are essential before the FE model can be used for further study. Thus, modal testing or EMA (Experimental Modal Analysis) should be carried out to gather modal data including natural frequency and mode shape. These variables will be used in FEA (Finite Element Analysis) to verify the outcomes that were expected [17]. According to [18], experimental modal analysis has become a well-known method in modal testing since the early 1970s, when the digital Fast Fourier Transform (FFT) spectrum analyser was first used. Software is used to accomplish the estimation of the modal parameters, also known as curve-fitting, which streamlines the extraction procedure [19].

The FEA method's extension has revealed inaccuracies in the obtained predicted results, particularly when dealing with complex structure. These inaccuracies arise from challenges in accurately modelling joints, boundary conditions, and damping. Consequently, the technology of model updating was developed to address this issue. Its goal is to maintain the precise representation provided by the FE model whole minimising errors within analytical models by adding observed dynamic test data [20]. In essence, modal updating entails changing some FE modal parameters [21].

Motivated by previous research, this study introduces a method for enhancing the dynamic behaviour with the reconfiguration of exhaust hangers' location using the updated FE model which treated with FE model updating technique and joint modelling strategy. Although several published works [2, 5-7, 22, 23] have explored the optimisation of exhaust hanger locations, none of them have utilised an updated FE model in the process. A joint modelling technique is also used in this study's FE model of the exhaust construction to choose the most trustworthy element connector that faithfully reproduce the welded joint model. An FEA package's modal frequency response analysis is used to optimise the hanger positions. Numerous case studies have been undertaken using different arrangements for exhaust hanger positions. The outcomes show that the displacement of the FE model of exhaust structure has been significantly reduced, dropping from 17 mm with the original hanger location to 0.77 mm with the optimised hanger location. The best hanger positions for certain applications can be found by using the suggested strategy in upcoming study for various exhaust system or structure types that include hangers.

2. Finite Element Modelling of The Test Structure

In this work, modal analysis was carried out suing a numerical prediction approach to ascertain the structure's dynamic behaviour. Using the FEA programme MSC. Nastran/Patran, important modal analysis parameters as natural frequency and mode shape were retrieved. Figure 2 illustrates the procedures used in this study. The test structure was too intricate to be directly represented in the FEA software, hence the FE model was initially produced using the CAD programme SolidWork. The FEA programme MSC. Patran was used to pre-process the FE model after it had been entirely built in CAD using the compatible format "Parasolid". Previous studies have used this strategy of building the FE model in CAD before importing it into the FEA programme [2, 24-26].

The FE model was meshed using CTRIA3 Shell elements during the pre-processing phase, yielding a total of 17,551 elements and 11,359 nodes. The material and physical attributes were then applied to the FE model using the meshing procedures, as described in Tables 1 and Table 2, respectively, as illustrated in Figure 1. In *Section 2.1 Joint Modelling Strategy* provides further information on the joint modelling strategy.

Table 1
Nominal value defined to each of the exhaust structure brackets

Properties		Nominal Value
Material properties	Poisson Ratio (v)	0.33
(Mild Steel 1010)	Density (ρ)	8000 kg/m^3
	Young's Modulus (E)	200 GPa
Shell properties	Bracket 1 (thickness)	0.0075 m
	Bracket 2 (thickness)	0.0084 m
	Bracket 3 (thickness)	0.0084 m

Table 2
Assigned nominal value for exhaust structure parts

Properties		Nominal Value
Material properties	Poisson Ratio (v)	0.29
(Mild Steel 304)	Density (ρ)	8000 kg/m^3
	Young's Modulus (E)	295 GPa
Shell properties	Pipe (thickness)	0.002 m
	Resonator (thickness)	0.0025 m
	Muffler (thickness)	0.003 m

Fig. 1. Exhaust structure FE model mesh in FEA

The SOL103 normal mode analysis was used analysis was used to analyse the FE model once the input properties of the pre-processing step were finalised. Using the MSC. Nastran Solver, the mathematical model was created and iterated in the bulk data file "bdf.". Using MSC. Patran, the normal mode analysis findings were acquired at the post-processing step. Important factors line natural frequency and mode shape may be recovered during post-processing. As further described in the correlation procedure, the projected FEA findings would next be evaluated and validated against the relevant data.

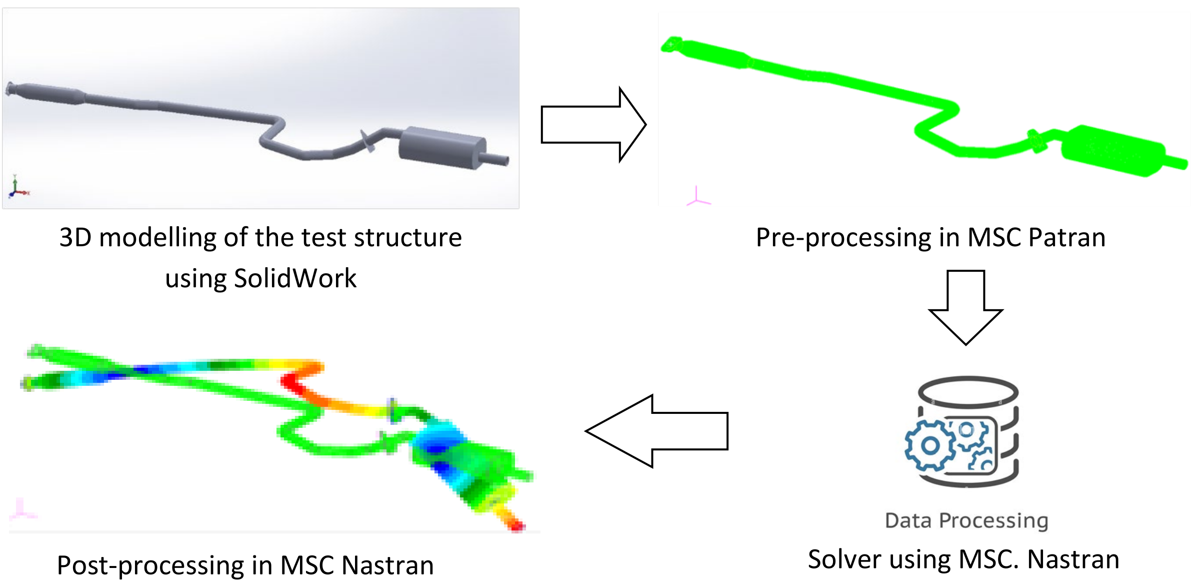


Fig. 2. Process flow of normal mode analysis SOL103 of FE model for exhaust structure

2.1 Joints Modelling Strategy

In this work, a joint modelling technique was used to construct the Finite Element (FE) model of the exhaust structure in order to properly reflect genuine connections, such as welded and bolted joints. Figure 3 shows where the joints are located on the test construction. There were three bolted joint points and a total of seven welded joints places. The weld joints on the test structure were modelled using a variety of element connectors from the FEA software, including RBE2, CBAR, CELAS, and CBEAM, as illustrated in Figure 4 to Figure 7. On the other side, the CBUSH element connection was used to simulate the bolt joints.

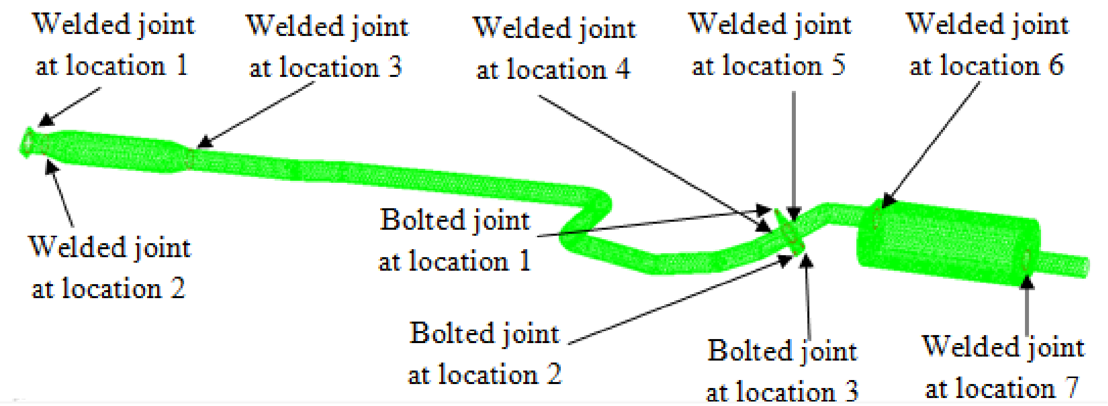


Fig. 3. 3D model of exhaust structure designed by CAD software

The FE model was prepared with a gap distance of 0.003 meters at each joint location to accommodate the application of element connectors RBE2, CBAR, CELAS, and CBEAM were used to represent the welded joint model. In contrast, the bolted joint model was replicated using 24 element connectors of CBUSH. The properties of each element connector were assigned in the FE model and are listed in Table 3.

Table 3
Nominal value assigned for element connector of joint model

Properties		Nominal Value
CBUSH (bolted joint model)	Stiffness, K	7E+17 N/m
CELAS (welded joint model)	Stiffness, K	7E+17 N/m
CBAR (welded joint model)	Young's Modulus	1000 GPa
	Poisson Ratio	0.29
	Density	8000 kg/m^3
	Dimension (diameter)	0.001 meter
CBEAM (welded joint model)	Young's Modulus	1000 GPa
	Poisson Ratio	0.29
	Density	8000 kg/m^3
	Dimension (length x wide)	0.001 meter x 0.001 meter

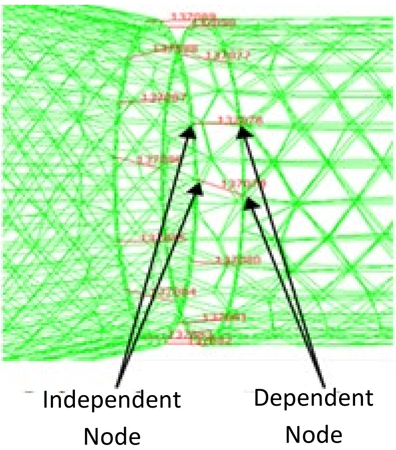


Fig. 4. RBE2 element connector [27]

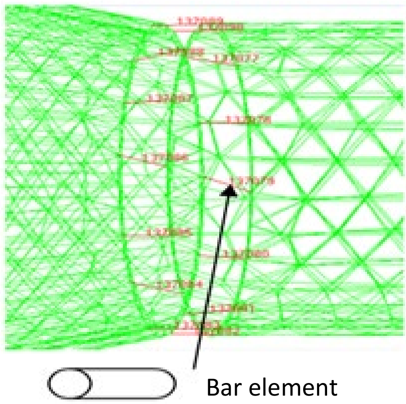


Fig. 5. CBAR element connector [27]

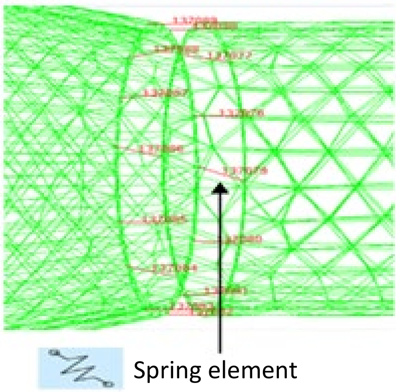


Fig. 6. CELAS element connector [27]

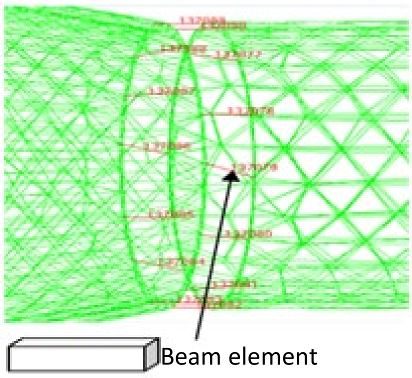


Fig. 7. CBEAM element connector [27]

3. Experimental Modal Analysis of The Test Structure

The modal testing technique, also known as Experimental Modal Analysis (EMA), was implemented in this work to collect the measured data. Figure 8 illustrate the setup for the experiment and the setting. The test structure was hung using elastic bungee rope to represent free-free boundary conditions. Figure 9 includes a list of the equipment used throughout the testing procedure. Following the roaming accelerometer approach with a fixed excitation point, measurements were performed using two triaxial accelerometers. The test structure, which was modelled as a wire frame structure as depicted in Figure 10, underwent a total of 66 measurement points. The wire frame structure was successfully imitated the mode shape from the taken measurements.

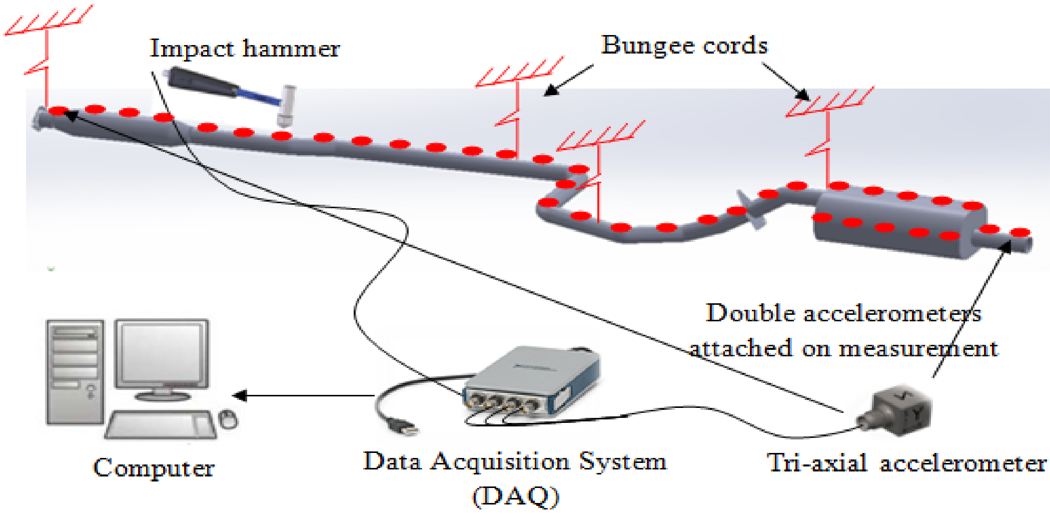
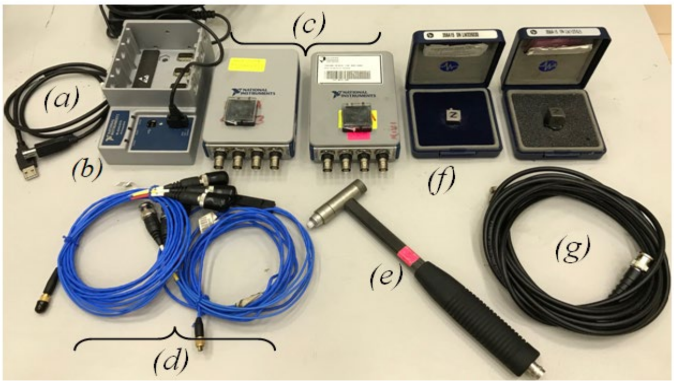


Fig. 8. Free-free boundary condition setup modal testing of exhaust structure



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

Fig. 9. List of equipment used in EMA

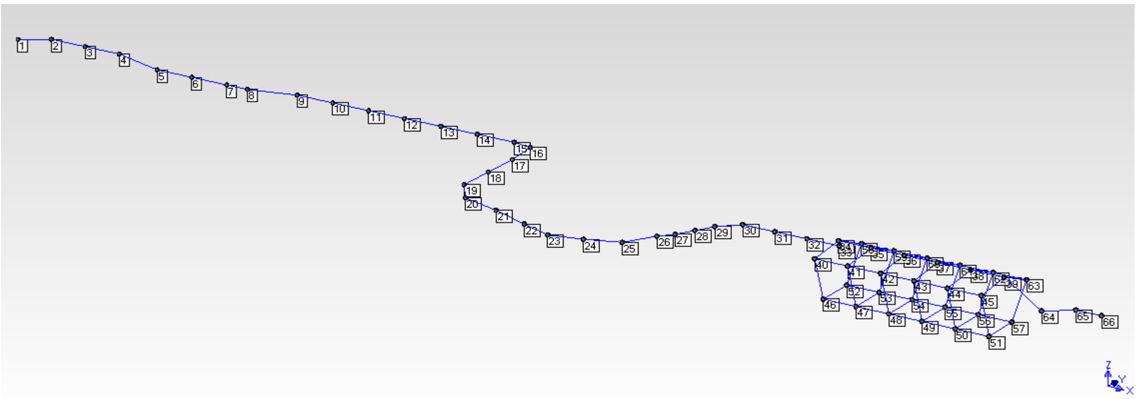


Fig. 10. Wireframe model in EMA software with 66 measurement points

The coherence graph presented in Figure 11 provides an indication of the measurement quality in this study. It demonstrates the level of consistency in the data. On the other hand, Figure 12 displays the computed frequency response functions (FRFs) obtained from the measurements. These FRFs are utilised in the curve-fitting process to extract the modal parameters, namely the natural frequency and mode shape.

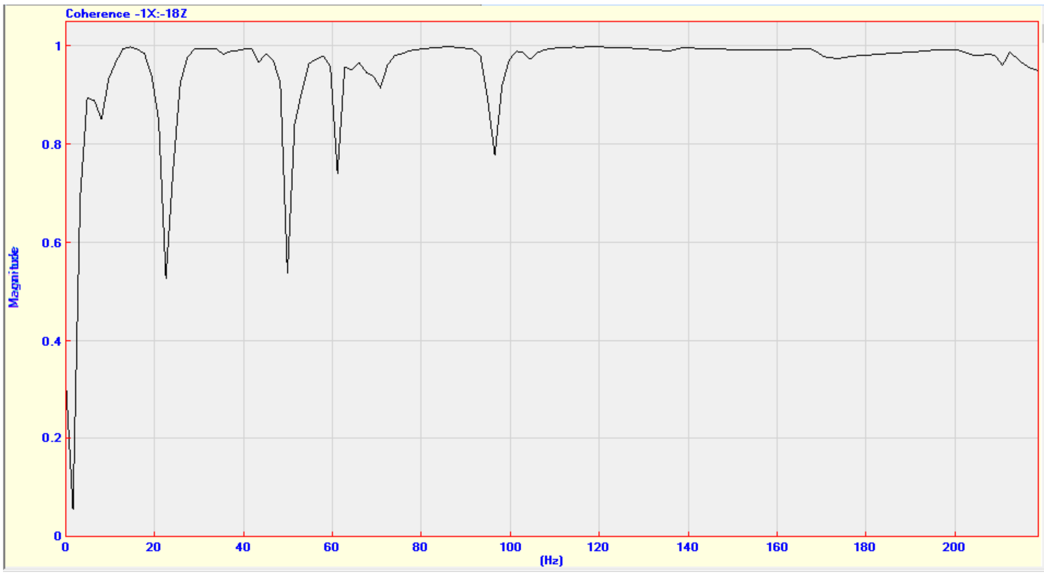


Fig. 11. Graph of coherence for modal testing

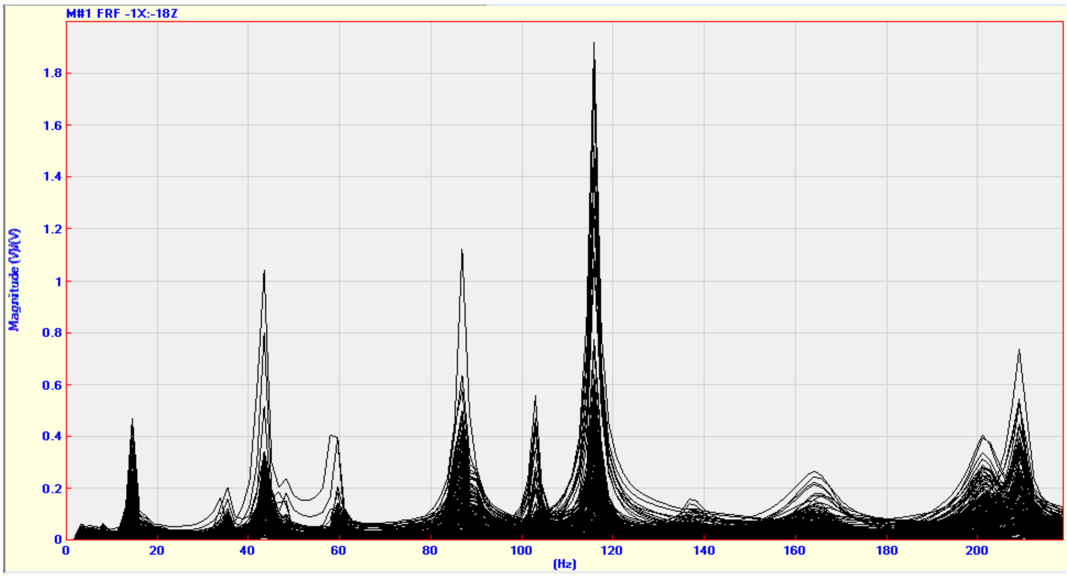


Fig. 12. Linear magnitude of frequency response functions (FRFs)

4. Result and Discussion

4.1 Result of Correlation between FEA and EMA

Since the result of FEA is numerical prediction process that used an assumption and simplification during pre-processing stage, hence it's required to be validated with its measured test data which been carried out in EMA to increase the integrity level. The correlation process was implemented in validating FEA with EMA. Through correlation process, result of FEA has been compared with measured test data in EMA as tabulated in Table 4 for mode shapes and Table 5 for natural frequencies (NF) in order to provide clear visual of correlation between FEA and EMA. Equation 1 was adopted to determine the percentage error which represents the level of disagreement between numerical predicted result in FEA and measured test data from EMA.

$$Percentage\ of\ error, (\%E) = \left|\frac{f^{FEA} - f^{EMA}}{f^{EMA}}\right| \times 100 \quad (1)$$

where f^{FEA} is natural frequency computed in FEA while f^{EMA} is natural frequency extracted from EMA. The CBAR element connector showed the lowest percentage error (*%E*) with 4.10 % as tabulated in Table 5. The other element connectors such as Equivalence (Equi.), RBE2, CBEAM, CELAS have the percentage of errors (*%*E) 7.21 %, 4.49 %, 5.37 %, and 26.78% compared with its measured test data from EMA. As the result from correlation process, CBAR element connector has been chosen for further action in this study (FE model updating process) since it most feasible to replicate the real welded joint for the test structure.

Table 4
Correlation of mode shapes computed in FEA compared with measured data extracted from EMA

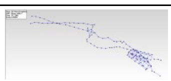
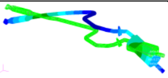
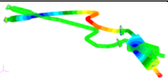
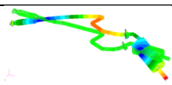
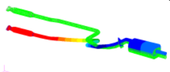
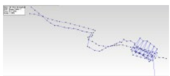
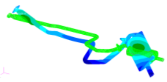
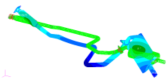
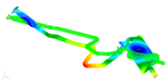
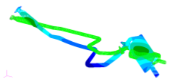
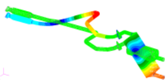
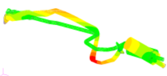
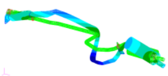
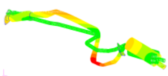
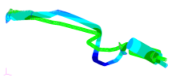
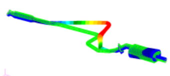
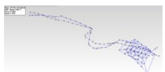
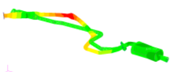
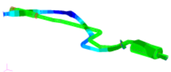
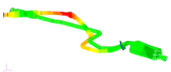
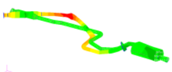
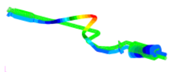
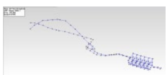
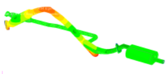
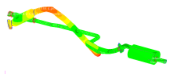
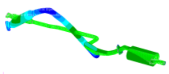
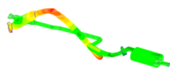
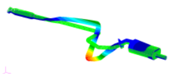
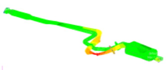
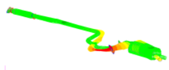
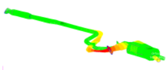
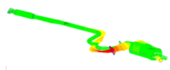
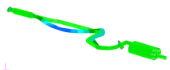
NF	EMA	Equivalence	RBE2	CBAR	CBEAM	CELAS
1	 14.10 Hz	 13.46 Hz	 14.70 Hz	 14.54 Hz	 14.04 Hz	 23.02 Hz
2	 43.70 Hz	 43.42 Hz	 41.33Hz	 40.26 Hz	 37.57 Hz	 53.14 Hz
3	 59.80 Hz	 73.79 Hz	 62.33 Hz	 61.44 Hz	 59.55 Hz	 63.72 Hz
4	 87.00 Hz	 91.50 Hz	 92.03 Hz	 91.69 Hz	 91.24 Hz	 71.48 Hz
5	 116.00 Hz	 113.20 Hz	 110.69 Hz	 110.06 Hz	 107.76 Hz	 140.55 Hz
6	 165.00 Hz	 176.74 Hz	 169.40 Hz	 164.41 Hz	 156.12 Hz	 214.97 Hz

Table 5
Comparison of Natural Frequencies (NF) for FE model of test structure with various joint element strategy computed in FEA with its measured data extracted from EMA

NF	Natural Frequency (Hz)										
	EMA	Equi.	(*%E*)	RBE2	(*%E*)	CBAR	(*%E*)	CBEAM	(*%E*)	CELAS	(*%E*)
1	14.10	13.46	4.54	14.70	4.26	14.54	3.12	14.04	0.43	23.02	63.26
2	43.70	43.42	0.64	41.33	5.42	40.26	7.87	37.57	14.03	53.14	21.60
3	59.80	73.79	23.39	62.33	4.23	61.44	2.74	59.55	0.42	63.72	6.56
4	87.00	91.50	5.17	92.03	5.78	91.69	5.39	91.24	4.87	71.48	17.84
5	116.00	113.20	2.41	110.69	4.58	110.06	5.12	107.76	7.10	140.55	21.16
6	165.00	176.74	7.12	169.40	2.67	164.41	0.36	156.12	5.38	214.97	30.28
Total Average Error			7.21		4.49		4.10		5.37		26.78

4.2 FE Model Updating

FE model updating technique has been implemented in this study for the purpose to reduce the discrepancies occurred between predicted result analysed in FEA with its measured test data extracted from EMA. FE model updating techniques have been categorised into two which are non-iterative (direct method) and iterative method [20]. For this study, optimisation algorithm SOL200 function that existed in MSC. Nastran was used to update the FE model of test structure using iterative technique. During the process of FE model updating, some of modal parameters were altered using SOL200 function [21]. As reported by [12, 28] in their studies, this technique have been successfully reduced the discrepancies between predicted numerical analysis of test structure with its measured counterpart.

4.2.1 Objective Function

The FE model updating technique is designed to minimise an objective function based on residuals between predicted numerical modal data (natural frequencies and mode shape) and its measured test data. When the discrepancy between values of the objective function (*J*) from successive iterations is appropriately small, convergence well-said to be occurred.

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

According to (2), w_j is a weighting coefficient for each mode while λ_j^{exp} is the j^{th} experimental eigenvalue and λ_j is the j^{th} eigenvalue predicted in numerical analysis process. The MSC. Nastran optimisation method has been adopted in this study during FE model updating process. The result of model updating of test structure has been tabulated in Table 6. It shown there are six (6) modes have been compared between initial FE model with joint element strategy using CBAR element connector and updated FE model. The percentage error successfully descends from 4.10 % to 3.74 % and it acceptable to be used in further action in this study.

Table 6
Comparison of Natural Frequency between Initial and Updated FE model with CBAR element connector compared to its measured counterpart in EMA

NF	Natural Frequency (Hz)				
	EMA	Initial CBAR Element	(*%E*)	Updated CBAR Element	(*%E*)
1	14.10	14.54	3.12	14.11	0.07
2	43.70	40.26	7.87	40.00	8.47
3	59.80	61.44	2.74	61.47	2.79
4	87.00	91.69	5.39	92.22	6.00
5	116.00	110.06	5.12	110.67	4.59
6	165.00	164.41	0.36	164.15	0.52
Total Average Error			4.10		3.74

4.3 Enhancement of Exhaust Hanger Location

Hanger is one of critical part in vehicle design which been used to attach the exhaust structure to the chassis. For current situation, there are various placement of hanger location in exhaust structure across different vehicle brands and models. Hence, it's crucial to identify the best location of the hanger in order to minimise the dynamic load experienced by the test structure. To evaluate the test structure amplitude (displacement), the modal frequency response analysis is implemented. Based on this method, the lowest displacement experienced by the proposed hanger configuration has been considered as ideal position in enhancing the hanger location.

Figure 13 represented the possible hanger layout in three (3) views; TOP, FRONT, and REAR in enhancing hanger locations with the original location and the proposed location. The locations were labelled as L1, L2, L3, L4 and L5 for original location while L1a, L1b, L2a, L2b, L3a, L3b, L4a, L4b, and L5a are proposed locations. The coordinate for each location (original and proposed) were tabulated in Table 7 due to x-axis, y-axis, and z-axis which been used during pre-processing stage for modal frequency response analysis for each case study. There are about 35 case studies were analysed using modal frequency response analysis SOL111 function in MSC. Nastran/Patran as summarized in Table 8.

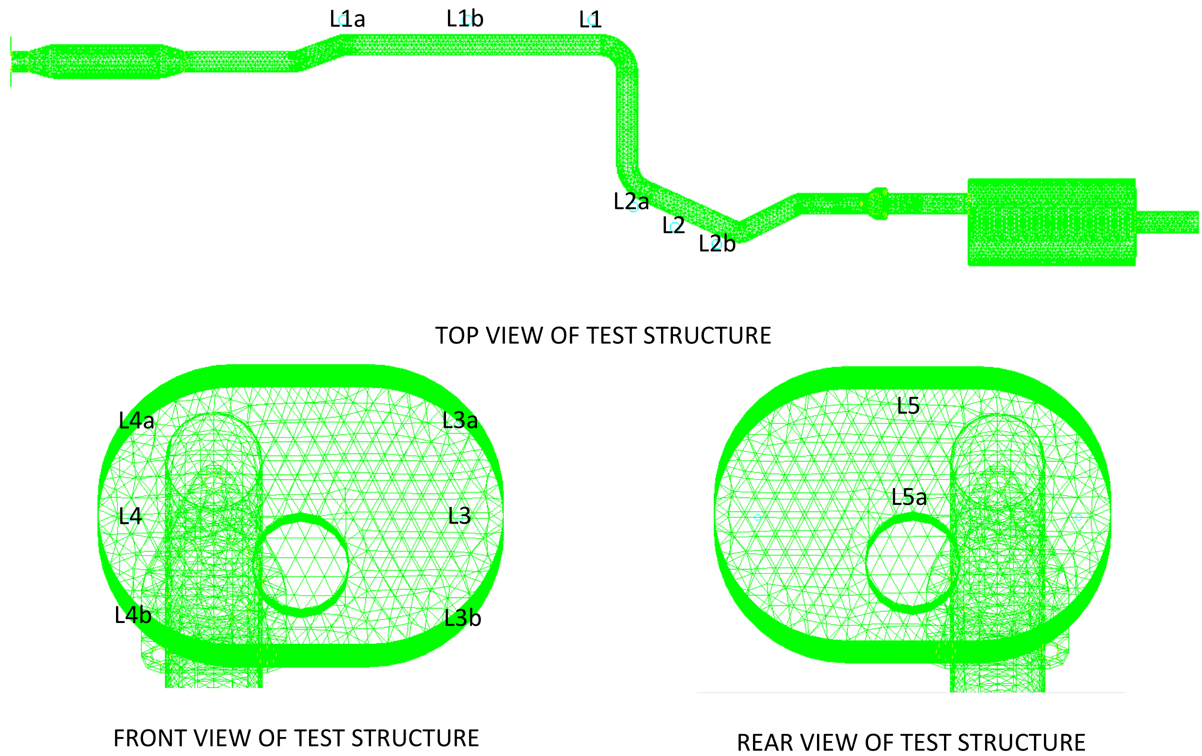


Fig. 13. Original and proposed hanger location on exhaust structure

Table 7
Proposed positions in optimising hanger locations of exhaust structure

Hanger location on exhaust structure (meter)							
Hanger	X-axis	Y-axis	Z-axis	Hanger	X-axis	Y-axis	Z-axis
L1	1.40	- 0.005	- 0.10	L2	1.60	- 0.005	0.40
L1a	20.80	- 0.005	- 0.10	L2a	1.50	- 0.005	0.35
L1b	1.10	- 0.005	- 0.10	L2b	1.70	-0.005	0.44
L3	2.29	0.12	0.47	L4	2.29	0.12	0.30
L3a	2.29	0.17	0.47	L4a	2.29	0.17	0.30
L3b	2.29	0.07	0.45	L4b	2.29	0.07	0.32
L5	2.73	0.18	0.39				
L5a	2.73	0.13	0.39				

Table 8
Proposed configurations of hanger locations of exhaust structure

No. of	Configuration of hanger locations				
Case Study	Hanger 1	Hanger 2	Hanger 3	Hanger 4	Hanger 5
1	H1a	H2	H3	H4	H5
2	H1	H2a	H3	H4	H5
3	H1	H2	H3a	H4	H5
4	H1	H2	H3	H4a	H5
5	H1	H2	H3	H4	H5a
6	H1b	H2	H3	H4	H5
7	H1	H2b	H3	H4	H5
8	H1	H2	H3b	H4	H5
9	H1	H2	H3	H4b	H5
10	H1a	H2a	H3	H4	H5
11	H1b	H2a	H3	H4	H5
12	H1b	H2b	H3	H4	H5
13	H1a	H2a	H3a	H4	H5
14	H1b	H2a	H3a	H4	H5
15	H1b	H2b	H3a	H4	H5
16	H1b	H2b	H3b	H4	H5
17	H1a	H2a	H3a	H4a	H5
18	H1b	H2a	H3a	H4a	H5
19	H1b	H2b	H3a	H4a	H5
20	H1b	H2b	H3b	H4a	H5
21	H1a	H2b	H3b	H4a	H5
22	H1b	H2b	H3b	H4b	H5
23	H1a	H2a	H3a	H4a	H5a
24	H1b	H2a	H3a	H4a	H5a
25	H1b	H2b	H3a	H4a	H5a
26	H1b	H2b	H3b	H4a	H5a
27	H1b	H2b	H3b	H4b	H5a
28	H1	H2a	H3	H4a	H5
29	H1	H2b	H3	H4b	H5
30	H1a	H2	H3a	H4	H5a
31	H1a	H2	H3a	H4	H5
32	H1b	H2	H3b	H4	H5a
33	H1b	H2	H3b	H4	H5
34	H1	H2a	H3	H4a	H5a
35	H1	H2b	H3	H4b	H5a

From Table 8, there are 35 case studies have been analysed with modal frequency response SOL111 with the proposed configurations of hanger locations in exhaust structure. There are five (5) hanger locations for each case studies based on the original number of hanger location. From the analysis, the result of displacement of test structure will be used to evaluate the hanger location. The result for original hanger location is displayed in Figure 14 with the peak of displacement has been recorded at 17 mm. From 35 case studies that undergo with modal frequency response analysis, case study no. 9 as depicted in Figure 15 has lowest displacement with 0.77 mm compared to the other case studies. In addition, from the graph of displacement versus frequency, its obvious showed that all hanger location in case study no. 9 closely align with each other compared to its original hanger location and other proposed hanger configuration.

With the lowest displacement it absolutely produced better dynamic behaviour of the test structure. The smallest displacement of test structure means the vibration transmitted and propagated along the test structure is acceptable and better for lifespan of the structure.

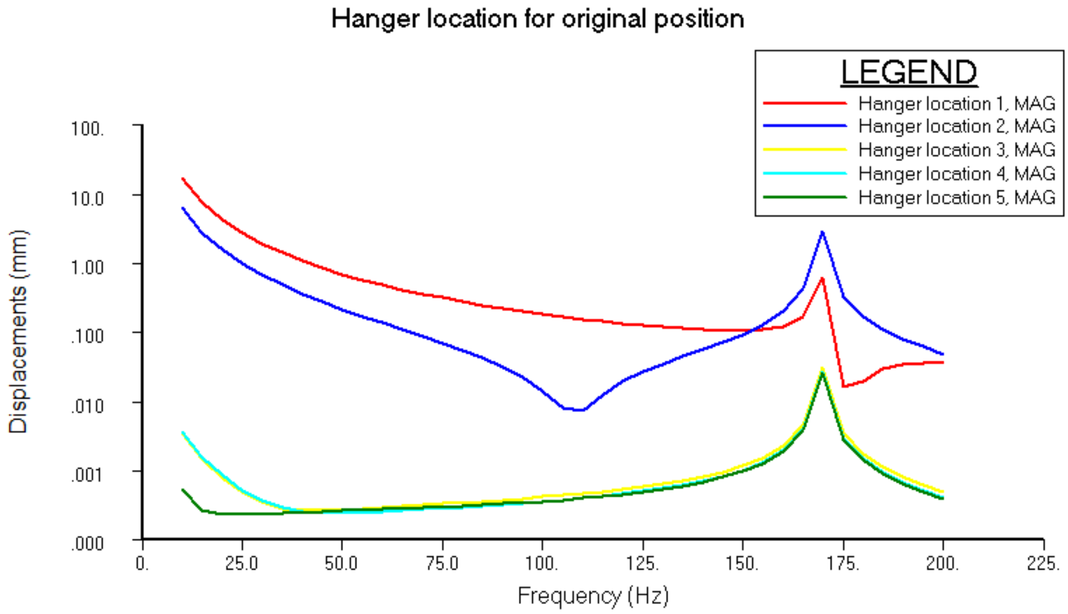


Fig. 14. Graph of displacement versus frequency for original hanger locations

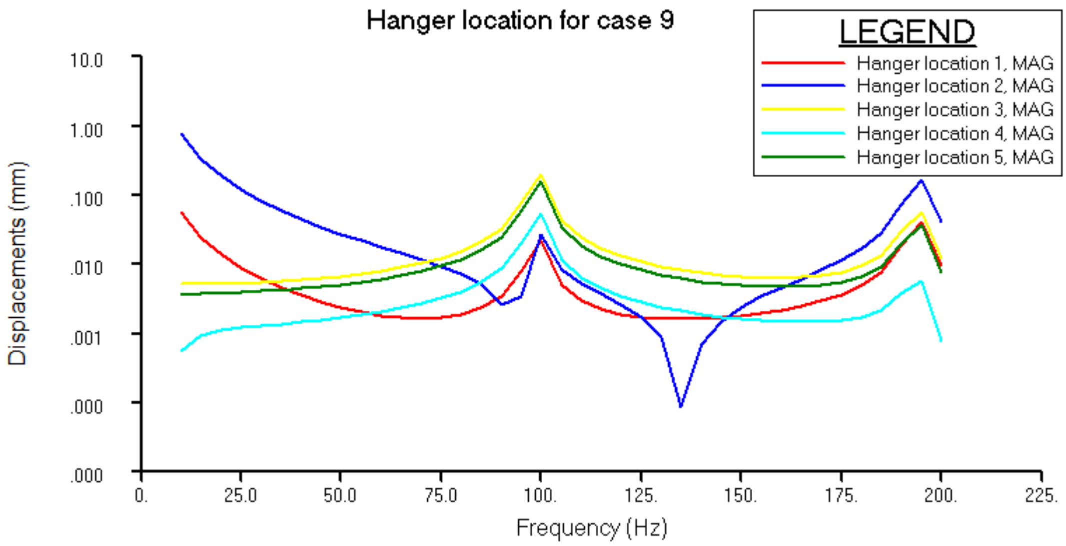


Fig. 15. Graph of displacement versus frequency for optimised hanger locations

Due to result portrayed in Figure 14, it's clearly showing the peak amplitude for the original hanger locations reached approximately 17 mm.

5. Conclusions

It can be concluded that the approach implemented in this study is feasible to enhance the dynamic behaviour of exhaust structure with the enhancement of hanger location. At early stage of this study, the test structure has been modelled using Computer Aided Design (CAD) package, SolidWork in 1 to 1 scale. Then, this 3D model has been imported into MSC. Patran for pre-processing stage such as meshing process, assignment of boundary condition, material input, and analysis type selection. During this pre-processing stage, joint modelling strategy has been adopted by using existed element connector such CBAR, CBEAM, CELAS, and CELAS to replicate the welded joint model, while CBUSH element connector to replicate bolted joint model.

Then, the test structure has been analysed with normal mode analysis SOL103 solution in MSC. Nastran to compute the dynamic properties such as natural frequency and mode shape that been used in FE model updating technique. Since the numerical prediction process in FEA used an assumption and simplification due to complexity of the test structure, the predicted result needs to be verified with its measured test data. This measured test data has been extracted from EMA with roving accelerometer method within 66 measurement points on test structure. The correlation process was implemented to observe the level of agreement between numerical prediction and its measured test data.

From the correlation process, FE model with CBAR element connector showed the most feasible model with 4.10% of percentage error compared with its measured counterpart. Then, this FE model with CBAR element connector was treated with FE model updating technique to improve the level of agreement between its measured counterparts. The percentage error successfully reduced from 4.10 % to 3.74 %. This reliable FE model with CBAR element connector then has been used in enhancing dynamic behaviour of exhaust by reconfiguration the hanger locations. This reconfiguration involved 35 case studies with the new proposed hanger locations. These 35 case studies have been analysed with modal frequency response SOL111 function in MSC. Nastran to evaluate the amplitude (displacement) produced by each case study.

From modal frequency response analysis, it has found that original hanger location recorded 17 mm displacement while case study no. 9 produced the lowest displacement with 0.77 mm compared to the other case studies. In summary, the suggested method in this study shows it's feasible to enhance the dynamic behaviour of test structure numerically by lowering the displacement with reconfiguration the hanger location. This method offered the effective way rather than physical modification and field testing of test structure which will be consumed a lot of effort, time and cost compared to numerically predicted.

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 in accomplished this research.

References
[1] Wei, Zhigang, Sarat Das, Ryan Barr, Greg Rohrs, Robert Rebandt, Xiao Wu, and HongTae Kang. *Development of lightweight hanger rods for vehicle exhaust applications*. No. 2017-01-1709. SAE Technical Paper, 2017. https://doi.org/10.4271/2017-01-1709
[2] Shojaeifard, M. H., S. Ebrahimi-Nejad R, and S. Kamarkhani. "Optimization of exhaust system hangers for reduction of vehicle cabin vibrations." *Automotive Science and Engineering* 7, no. 1 (2017): 2314-2325.
[3] Rajadurai, M. S., R. Kavin, and Rejinjose Prabhakaran. "A system approach to dynamic characteristics of hanger rod in exhaust system." *International Journal of Innovative Science, Engineering & Technology* 3 (2016).
[4] Lupea, Iulian. "An exhaust system lumped model–identification and simulation." *ACTA TECHNICA NAPOCENSIS-Series: APPLIED MATHEMATICS, MECHANICS, and ENGINEERING* 59, no. 4 (2016).
[5] Wenzhu, Wang, Liu Gang, Cheng Mianhong, and Wei Jun. *Hanger location layout of automotive exhaust system*. No. 2015-01-2318. SAE Technical Paper, 2015. https://doi.org/10.4271/2015-01-2318
[6] Nyein, Ookyaw, Aminudin Bin Abu, and Aung Lwin Moe. "Numerical analysis of vehicle exhaust system to determine hanger location using root mean square value." *ARPN J Eng Appl Sci* 10 (2015).
[7] Waghmare, Vishwas Shantaram, Vaibhav Salunkhe, Rakesh Nagrani, and Vishal Guttal. "Analysis of exhaust system mounting scheme." *SAE International Journal of Vehicle Dynamics, Stability, and NVH* 1, no. 2017-26-0291 (2017): 89-96. https://doi.org/10.4271/2017-26-0291
[8] Gaonkar, Chetan D. "Modal analysis of exhaust system to optimize mounting hanger location." *International Journal of Engineering Research & Technology (IJERT)* 4, no. 03 (2015). https://doi.org/10.17577/IJERTV4IS030050

[9] Balaji, D. Jai, P. V. Srihari, and Veeranna B. Sheelvanth. "Model frequency analysis of automotive exhaust system." *Int J Mech Eng Robot Res* 3 (2014).
[10] Sarmadi, Hassan, Abbas Karamodin, and Alireza Entezami. "A new iterative model updating technique based on least squares minimal residual method using measured modal data." *Applied mathematical modelling* 40, no. 23-24 (2016): 10323-10341. https://doi.org/10.1016/j.apm.2016.07.015
[11] Fouzi, M. S. M., K. M. Jelani, N. A. Nazri, and Mohd Shahrir Mohd Sani. "Finite element modelling and updating of welded thin-walled beam." *International Journal of Automotive and Mechanical Engineering* 15, no. 4 (2018): 5874-5889. https://doi.org/10.15282/ijame.15.4.2018.12.0449
[12] Zahari, Siti Norazila, Mohd Shahrir Mohd Sani, and Mahadzir Ishak. "Finite element modelling and updating of friction stir welding (FSW) joint for vibration analysis." In *MATEC Web of Conferences*, vol. 90, p. 01021. EDP Sciences, 2017. https://doi.org/10.1051/matecconf/20179001021
[13] Sani, M. S. M., N. A. Nazri, and D. A. J. Alawi. "Vibration analysis of resistance spot welding joint for dissimilar plate structure (mild steel 1010 and stainless steel 304)." In *IOP Conference Series: Materials Science and Engineering*, vol. 238, no. 1, p. 012017. IOP Publishing, 2017. https://doi.org/10.1088/1757-899X/238/1/012017
[14] Nazri, N. A., and M. S. M. Sani. "Finite element normal mode analysis of resistance welding jointed of dissimilar plate hat structure." In *IOP Conference Series: Materials Science and Engineering*, vol. 257, no. 1, p. 012059. IOP Publishing, 2017. https://doi.org/10.1088/1757-899X/257/1/012059
[15] Izham, M. H. N., N. A. Z. Abdullah, S. N. Zahari, and M. S. M. Sani. "Structural dynamic investigation of frame structure with bolted joints." In *MATEC Web of Conferences*, vol. 90, p. 01043. EDP Sciences, 2017. https://doi.org/10.1051/matecconf/20179001043
[16] Abdullah, N. A. Z., M. S. M. Sani, M. M. Rahman, and I. Zaman. "Correlation of numerical and experimental analysis for dynamic behaviour of a body-in-white (BIW) structure." In *MATEC Web of Conferences*, vol. 90, p. 01020. EDP Sciences, 2017. https://doi.org/10.1051/matecconf/20179001020
[17] Zahari, S. N., A. A. R. Zakaria, MS Mohd Sani, and I. Zaman. "A review on model updating of joint structure for dynamic analysis purpose." In *MATEC Web of Conferences*, vol. 74, p. 00023. EDP Sciences, 2016. https://doi.org/10.1051/matecconf/20167400023
[18] Schwarz, Brian J., and Mark H. Richardson. "Experimental modal analysis." *CSI Reliability week* 35, no. 1 (1999): 1-12.
[19] Avitable, Peter. "Experimental modal analysis (a simple non-mathematical presentation)." In *Annual Technical Meeting-Institute of Environmental Sciences and Technology*, vol. 46, pp. 434-448. 2000.
[20] Modak, S. V., T. K. Kundra, and B. C. Nakra. "Comparative study of model updating methods using simulated experimental data." *Computers & structures* 80, no. 5-6 (2002): 437-447. https://doi.org/10.1016/S0045-7949(02)00017-2
[21] Mottershead, John E., Michael Link, and Michael I. Friswell. "The sensitivity method in finite element model updating: A tutorial." *Mechanical systems and signal processing* 25, no. 7 (2011): 2275-2296. https://doi.org/10.1016/j.ymssp.2010.10.012
[22] Yang, Yuliang, Yu Yang, Ying Sun, Zhihong Dong, and Yunquan Zhang. *Optimization of vehicle exhaust system hanger location*. No. 2016-01-0448. SAE Technical Paper, 2016. https://doi.org/10.4271/2016-01-0448
[23] Bötke, Artun, and Deniz Yazgaç. "Exhaust mount placement optimisation with comparable methods." In *23rd International congress on sound & vibration (ICSV23)*. 2016.
[24] Vinutha, B. S., and K. Hemanth. "Study of static and frequency responsible analysis of hangers with exhaust system." *IJASRE* 3 (2017).
[25] V. Kumbhar and D. C. Patil."Structural optimization & durability analysis of VW Bettle cross exhaust muffler system," *International Research Journal of Engineering and Technology (IRJET)* 4, no. 7 (2017).
[26] N. Vaddankeri, N. G. S. Udupa, and Venkategowda."Stress analysis of exhaust system in automobiles," *International Journal for Scientific Research & Development (IJSRD)* 4, no. 3 (2016): 383-385.
[27] Fouzi, M. S. M., M. S. M. Sani, and Yusrizal Muchlis. "Finite Element Modelling and updating of welded joint for dynamic study of exhaust structure." In *IOP Conference Series: Materials Science and Engineering*, vol. 469, no. 1, p. 012099. IOP Publishing, 2019. https://doi.org/10.1088/1757-899X/469/1/012099
[28] Husain, N. Abu, H. Haddad Khodaparast, A. Snaylam, S. James, G. Dearden, and H. Ouyang. "Finite-element modelling and updating of laser spot weld joints in a top-hat structure for dynamic analysis." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 224, no. 4 (2010): 851-861. https://doi.org/10.1243/09544062JMES1787