JOURNAL OF ACOUSTICS AND VIBRATION RESEARCH (JAVR)
ISSN: XXXX-XXXX
VOL. 1, ISSUE 1, 2023, 76 – 89

Finite Element Modelling and Updating of Motorcycle Structure with Suitable Length of Connecting Element

K.M. Jelani[1] N. A. Z. Abdullah[1,], M. S. M. Sani[1,2], M.N. Abdul Rani[3] & M.A. Yunus[3]

[1]Faculty of Mechanical & Automotive Engineering Technology (FTKMA), Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.
[2]Automotive Engineering Centre (AEC), Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.
[3]Structural Dynamics Analysis & Validation (SDAV), School of Mechanical Engineering, Collage of Engineering, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Selangor, Malaysia

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

ABSTRACT

This paper attempts to present an appropriate way to model welded joints in a motorcycle structure using the finite element method. Joint modelling strategy was implemented in this study since the joint itself has a significant influence as local effect on dynamic behavior of a structure. The objective of the paper is to investigate the suitable length of connecting element for joints of the structure. The length of the gaps of the parts is a 3mm, 10mm and 15mm was created in 3D model of structure for this joint modelling purpose. The element connectors that available in the FEA software were utilized to model the welded at assembled parts on test structure in this study. Two locations were used on the FE model for modelling of welded joints such as CBAR element connector used to replicate spot weld joint. Ahead of the updating process, sensitivity analysis is made to select the most sensitive parameter for updating purpose. Optimization algorithm in MSC Nastran is used in FE model updating process. The results show that employing a gap connecting element as an updating parameter (3mm) offers greater convergence than using significant parameters. This paper concluded that suitable length of connecting element for joint structure with CBAR element connector is attainable to replicate the real welded motorcycle structure since it has the lowest discrepancy in correlation analysis.

Article History

Received: 03/10/2022

Revised: 02/04/2023

Accepted: 25/04/2023

Published: 31/05/2023

Keywords: Model updating; finite element method; parameterization; structural dynamics.

INTRODUCTION

A motorcycle structure built with a design that is difficult to join between parts with other parts with welding connections. The complex behavior of connecting elements plays a valuable role in characteristics of dynamic such as natural frequencies and mode shapes [1, 2]. The welded joints have turn out to be one of popular joints types in jointing the motorcycles structures because they can be easily disassembled, maintained and inspected. The finite element method has been used to develop the simulated finite element model of the motorcycles welded joints. The finite element modeling of structures depends upon various uncertain factors such as structural material properties, dimensions, boundary conditions, etc. The knowledge of experimental observations is important for an understanding there is always an error between the simulated and experimental. The experimental modal analysis has been studied over the latest few decades is used to measure the natural frequencies of the motorcycles structure of the welded joint specimen experimentally [3]. The FE model updating process using experimental data is presented. Previous studies have reported FE model updating procedure is performed using the NASTRAN optimization code (SOL200 [3]. It is carried out using NASTRAN optimization code. The procedure aims to adjust the uncertain properties of the FE model (from the weld joints) by minimizing the differences between the measured modal properties and the corresponding numerical predictions [4]. Result shows that the discrepancies of natural frequency between FEA and EMA are below than 10%. Sensitivity model updating is evaluated in order to make sure which parameters are significance in this structural dynamic modification. Young's modulus and density both materials are indicated significant parameters to do model updating [5].

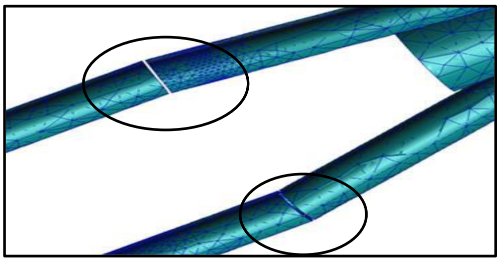
FINITE ELEMENT METHOD

Finite Element Modelling of Joint Structure

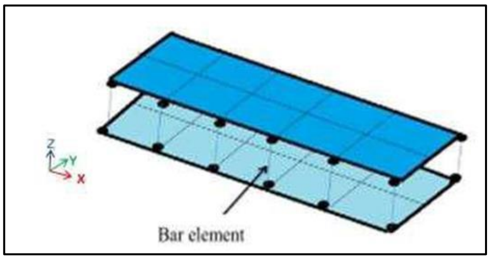
Previous studies have primarily concentrated on structural joints provide connection between structure element (beam, plate etc.) in order to construct a whole assembled structure [6]. There are many types of structural joints such as bolted joint, riveted joints and weld joints. Studies have found that on method of model updating on joints structure and discussed the guidelines to perform model updating for dynamic analysis purpose [1]. This paper firstly, define on some of existing finite element modelling works of joints structure according to an investigation suitable length of connecting element for joint modelling. Joint modelling strategy was implemented in this study since the joint itself has a significant influence as local effect on dynamic behaviour of a structure the element connectors that available in the FEA software were utilized to model the welded at assembled parts on test structure in this study[7]. The study examined the gaps of 3 mm, 10 mm and 15 mm was created in 3D model of structure for this joint modelling purpose. However, a number of studies show that significant differences do exist. This paper was undertaken to design two locations were used on the FE model for modelling of welded joints to investigate length cbar element connector used to replicate spot weld joint. Table 1 shows the material properties that include to the model. Figure 1 (a) and (b) show joint modelling using CBAR element of the structure.

Table 1. Material properties of stainless steel (SUB304)

Parameter	Value
Young Modulus (E)	200 GPa
Poisson's Ratio (ν)	0.29
Density (ρ)	7900 kg/m^3
Thickness (t)	0.002 m



(a)



(b)

Figure 1. (a)Two locations were used on the most reliable element in FE (b) CBAR element was selected as the FE model for modelling of welded joint.

Normal mode analysis is carried out using MSC. Nastran to determine the dynamic behavior of tested motorcycle structure with different length of connecting element test have been develop which stimulate a specific forming operation. The analysis simulates free-free boundary condition with no load or translational and rotational boundary conditions applied to any nodes on the structure [6]. The suitable length of connecting element for joints of the structure gaps of the parts is a 3 mm, 10 mm and 15 mm was created in 3D model of structure for this joint modelling purpose. Modal properties obtained from the FEA have been tabulated in Table 2, 3 & 4 difference length of connecting element for natural frequencies and mode shapes for each mode of interests. Previous studies have reported 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 2 [4] CBAR element is one example of joining provides in MSC Patran that able to withstand various kind of loads such as tension, compression and also bending forces. Researchers have studied the effect of after perform model updating, total error of the natural frequencies for CBAR model is improved significantly. Therefore, CBAR element was selected as the most reliable element in FE to represent motorcycle structure weld joint [8].

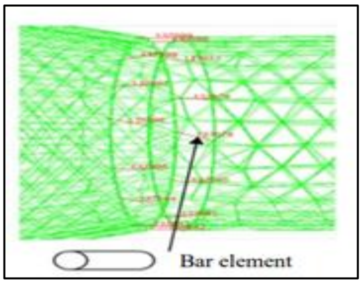


Figure 2. CBAR element is a straight prismatic element

Table 2. FEA have been tabulated in natural frequencies and mode shapes with 3 mm of CBAR

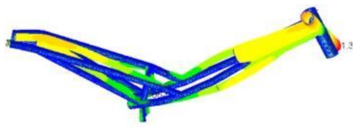
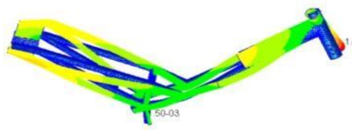
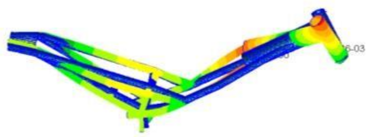
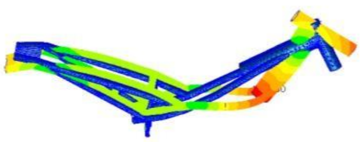
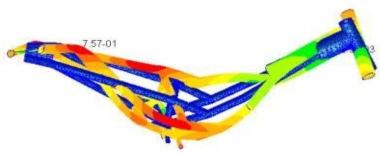
Mode	Natural frequency (Hz)	Mode shape of motorcycle structure
1	**239.84**	
2	**264.19**	
3	**380.67**	
4	**397.74**	
5	**519.32**	

Table 3. FEA have been tabulated in natural frequencies and mode shapes with 10 mm of CBAR

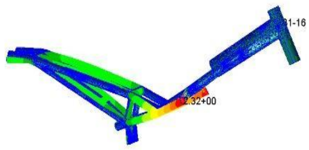
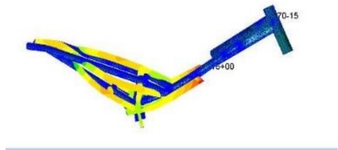
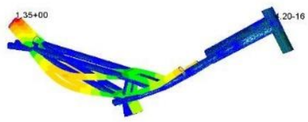
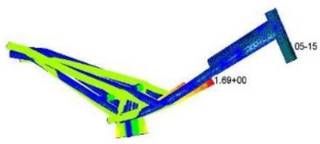
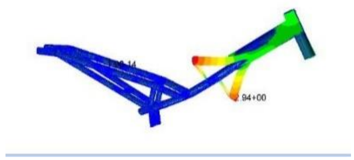
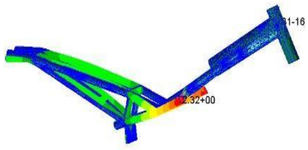
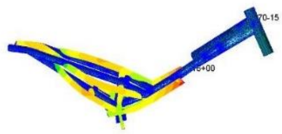
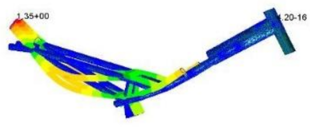
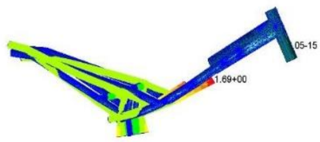
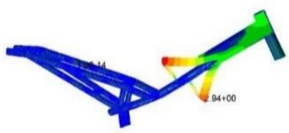
Mode	Natural frequency (Hz)	Mode shape of motorcycle structure
1	**399**	
2	**502**	
3	**509**	
4	**631**	
5	**742**	

Table 4. FEA have been tabulated in natural frequencies and mode shapes with 15 mm of CBAR

Mode	Natural frequency (Hz)	Mode shape of motorcycle structure
1	**399**	
2	**496**	
3	**508**	
4	**627**	
5	**722**	

EXPERIMENTAL MODAL ANALYSIS

Experimental Modal Analysis (EMA) Experimental Modal Analysis (EMA) or modal testing explains the details of method use on this paper to investigate the dynamic properties of the motorcycle chassis structure [9]. EMA is an approach to find the modal parameter of specified structure either through impact hammer testing, shaker testing or operational modal analysis [9]. In this study, focus is given to impact hammer test which claimed to be simpler and easier. Before conducting the impact hammer testing, a wireframe model of motorcycle chassis structure was created by using post-processing software. The model consists of lines and points which virtually represent the geometric shape of the motorcycle chassis structure as depicted Figure 3. In order to perform the impact hammer test, the structure of motorcycle chassis structure is hanged from a test rig by using elastic rope in order to put the structure under free-free boundary condition as demonstrated in Figure 4. The measurements were made using modal analysis software and several other equipment such as PCB 086D20 impact hammer with medium soft tip attached, 4- channel NI DAQ device, and a tri-axial PCB accelerometer as depicted in Figure 5 and Figure 6. Roving accelerometer method was adopted for the testing procedure, where one excitation point and 31 measurement points was assigned on the motorcycle chassis structure. Roving accelerometer test was done by creating initial disturbance on the motorcycle of structure at one fixed position while tri-axial accelerometer was roved straight other measurement points. The vibrational response was measured by using 4-channel NI DAQ device. The modal analysis software analytical data as shown in Figure 7 and Figure 8 used to extract the modal properties of the motorcycle of structure from the computed FRFs. Table 5 tabulated the result obtained from EMA for the natural frequency and mode shape.

Figure 3. Hanging motorcycle structures under free-free boundary conditions

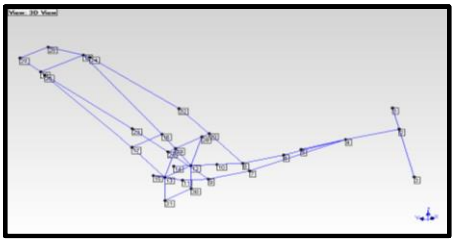


Figure 4. Wireframe chassis frame structure

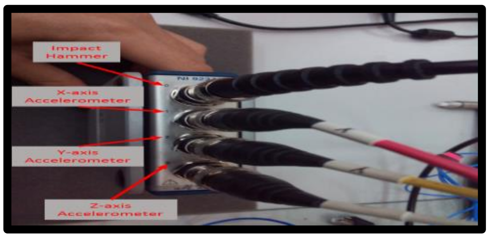


Figure 5 Connection of impact hammer and triaxial accelerometer

	Select Shape	Frequency (or Time)	Damping	Units	Damping (%)	Label	MPC
1	Yes	246	0.5	(Hz)	0.203	G-PLY-	0.0524
2	No	288	0.324	(Hz)	0.113	G-PLY-	0.216
3	No	383	0.509	(Hz)	0.133	G-PLY-	0.316
4	No	390	0.316	(Hz)	0.0809	G-PLY-	0.0548
5	No	516	0.0749	(Hz)	0.0145	G-PLY-	0.114

Figure 6 Setting for impact hammer and triaxial accelerometer

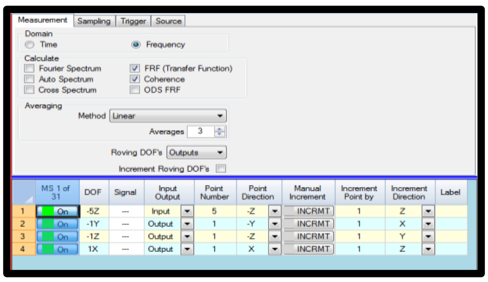


Figure 7 Frequency Response Function (FRF)

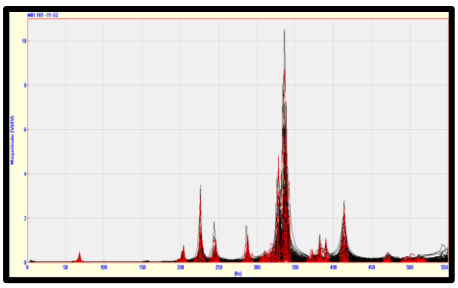
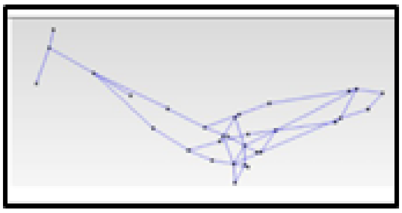
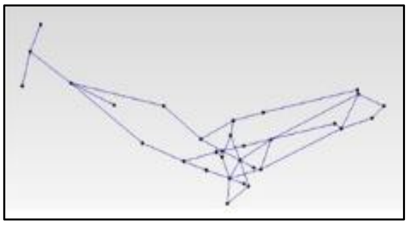
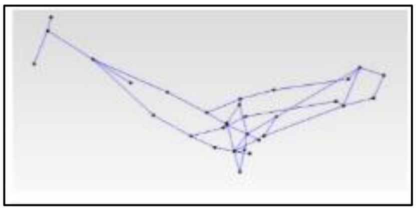
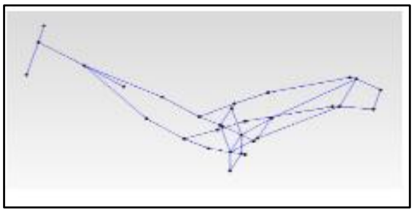
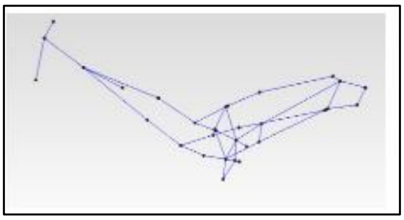


Figure 8 shape table after curve for fitting (FRF) for two acceleromete

Table 5. Natural frequencies and mode shapes formed by EMA for motorcycle structure.

Mode	Natural Frequency	Mode Shape
1	246	
2	288	
3	383	
4	390	
5	516	

After done with collecting the results from both, EMA and FEA, the differences would be determined in order to know about the discrepancies among the data obtained from both methods [11]. Table 6 shows the tabulated data for the comparison of natural frequencies results FEA and EMA. The percentage of error has been calculated by using Equation 1.

Table 6. Comparison of natural frequencies with error percentage length of connecting joint structure

Mode	Natural Frequency (Hz)						
	EMA	Initial FEA (3mm)	Error (%)	Initial FEA (1.0cm)	Error (%)	Initial FEA (1.5cm)	Error (%)
1	246	239.84	2.50	399	62.1	399	62.1
2	288	264.19	8.26	502	74.3	496	72.2
3	383	380.67	0.61	509	32.8	508	32.6
4	390	397.74	1.98	631	61.8	627	60.7
5	516	519.32	0.64	742	43.7	722	39.9
Total average			2.80		54.94		53.5

$$\%\ Error = \left| \frac{f_{experimental} - f_{theoretical}}{f_{theoretical}} \right| x100 \quad (1)$$

Then, the total average error would be calculated in order to proof that does the result of EMA obtained using more FEA lengths would be better compared to three of result. The result from three of length connecting element shows error for length 3 mm better one compared both 10 m and 15 mm. The percentage error of 3 mm length shown 2.80%, which result shows that the discrepancies of natural frequency between FEA and EMA are below than 10 %. However, the percentage for 10 mm and 15 mm of connecting length about 54.94% and 53.5%. The effect of various factors causes the result to be more than 10% error. The connecting element attached to the structure shows that the handle is low deformation, causes the cut structure to not connect properly. This causes the natural frequency to affect the dynamic behaviour. The mode shape in the simulation shows that the structure between the parts where the connecting element is installed does not move together, among the factors that can be detected is that the length of connecting between the structures exceeds 3 mm. Through the FEA results, this study tells us that connecting installed over 3 mm will be weak and not hold the installed part firmly.

FE MODEL UPDATING

Even though the percentage of error is 10% less, model updating of the FEA results is still being carried out because it could be a big help in during the analysis, either imperfection in experimental setup or modelling the chassis frame. So, in this paper, the FEA natural frequency would update by referring EMA as a benchmark. However, before proceeding into model updating, a sensitivity analysis should be done first. From this analysis, the most sensitive parameters among Young's Modulus, E, density, p and Poisson Ratio, v which mostly would affect the updated FEA could be discoverable [12]. Consequently, the sensitivity analysis is necessary to run in advance, so that only the most sensitive parameter will be selected. The sensitivity analysis performed in this study and represented in Table 7. There are three parameters; Modulus's Young (E), Density (ρ), and Poisson's Ratio (ν) in sensitivity matrix form. Modal updating of the FEA result is done on 2 case studies about 5 mode 3 parameter's and 5 mode 2 parameters. Referred to the table 8, it can be seen that all the three parameters show the significant value which means that they are sensitive for model updating work. Hence, the model is being updated using three parameters; Young's Modulus (E), density, (ρ) and Poisson ratio, (ν). The Table 8, it can be seen that all the two parameters show the significant value which means that they are sensitive for model updating work Young's Modulus (E) and density, (ρ). Hence, the model is being updated using three parameters; Young's Modulus (E) and density, (ρ).

Table 7. Sensitivity matrix analysis coefficient analyzed for five modes and three parameters

Mode	Young modulus (E) GPa	Density (ρ) kg/m^3	Poisson ratio (ν)
1	118	-118	-10.4
2	131	-131	-2.4
3	187	-186	0.6
4	195	-195	-1.74
5	255	-255	-4.02

Table 8. Sensitivity matrix analysis coefficient analyzed for five modes and two parameters

Mode	Young modulus (E) GPa	Density (ρ) kg/m^3
1	118	-118
2	131	-131
3	187	-186
4	195	-195
5	255	-255

Based on Table 9 and 10, the updating process is conducted by using 5 modes and 3 parameters of interest and the error value decreased by 0.32%. Then for comparison 5 modes 2 parameters the updating process of interest and the error value decreased by 0.26%.

Table 9. Correlation between EMA with initial and updated FE model with CBAR element connector (5 mode 3 parameters)

	Natural Frequency (Hz)				
Mode	EMA	Initial FEA (CBAR)	Error (%)	Updated CBAR	Error (%)
1	246	239.84	2.50	241	2.03
2	288	264.19	8.26	266	7.63
3	383	380.67	0.61	379	1.04
4	390	397.74	1.98	396	1.53
5	516	519.32	0.64	517	0.19
Total average			2.80		2.48

Table 10. Correlation between EMA with initial and updated FE model with CBAR element connector (5 mode 2 parameters)

	Natural Frequency (Hz)				
Mode	EMA	Initial FEA (CBAR)	Error (%)	Updated CBAR	Error (%)
1	246	239.84	2.50	241	2.03
2	288	264.19	8.26	266	7.63
3	383	380.67	0.61	378	1.30
4	390	397.74	1.98	396	1.53
5	516	519.32	0.64	517	0.19
Total average			2.80		2.54

DISCUSSION

The model updating process has been done for several times in between the minimum and maximum boundary limit of updating parameters' value until the value of natural frequencies converged. After updating is completed, the optimum value of Young's Modulus (E) and density, (ρ) was tabulated in Table 11.

Table 11. Updated Value of parameters based on design variable

Design Variable						Changes
Parameter	Lower Bound	Upper Bound	Value	Initial Value (i)	Value Updated (u)	[(u-i)/i]
Young modulus (E) GPa	0.95	1.05	1.01	200	202	0.01
Density (ρ) kg/m^3	0.90	1.10	1.01	7900	7979	0.01
Poisson ratio (μ)	0.96	1.04	1.04	0.29	0.3	0.03

CONCLUSIONS

This paper described parameter selections in the stochastic model updating that may be applied in future studies to quantify variability in the dynamics of structures [3]. The study has been conducted to three different length of connecting element of motorcycles structures, i.e., gap connecting element 3 mm. 10 mm and 15 mm by using same of parameters (i.e., the thickness and material properties). The findings indicate that one of gap connecting element (3mm) as the updating parameters provides better convergence than those updated by using parameters. As some general guidelines, the selection of parameters should be made by choosing the most sensitive parameters to the response of the system. This can be easily achieved by finalize out a simple sensitivity analysis. The research has been shown result after updated 3 of parameter error 2.48% compare updated 2 of parameter error 2.54%. The selection of sensitivity parameters should also because the output errors measured can be obtained. Percentage of error reduced 0.32% for 3 parameter and for 2 parameter 0.26% to below 11% after some modification was doing on motorcycles structure

ACKNOWLEDGEMENTS

Authors would like to thank to Universiti Malaysia Pahang (UMP) and Institut Kemahiran Tinggi Perda (PERDA-TECH) to appreciate the funding body and other supporting members.
.

REFERENCES

[1] Zahari, S., et al. *A review on model updating of joint structure for dynamic analysis purpose*. in *MATEC Web of Conferences*. 2016. EDP Sciences.

[2] Sani, M., et al. *Dynamic Correlation Technique and Model Updating on Go Kart Chassis Structure*. in *Proceedings of EnCon2008, 2nd Engineering Conference on Sustainable Engineering Infrastructures Development & Management*. 2008.

[3] Husain, N.A., et al. *FE model updating for damage detection–application to a welded structure*. in *Key Engineering Materials*. 2009. Trans Tech Publ.

[4] Fouzi, M., M. Sani, and Y. Muchlis. *Finite Element Modelling and updating of welded joint for dynamic study of exhaust structure*. in *IOP Conference Series: Materials Science and Engineering*. 2019. IOP Publishing.

[5] Sani, M., N. Nazri, and D. 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*. 2017. IOP Publishing.

[6] Kim, J., J.-C. Yoon, and B.-S. Kang, *Finite element analysis and modeling of structure with bolted joints.* Applied mathematical modelling, 2007. **31**(5): p. 895-911.

[7] Fouzi, M. and M. Sani, *Finite Element Modeling and Updating for Dynamic Study of Exhaust Structure*, in *Advancement in Emerging Technologies and Engineering Applications*. 2020, Springer. p. 305-313.

[8] Zahari, S.N., M.S.M. Sani, and M. Ishak. *Finite element modelling and updating of friction stir welding (FSW) joint for vibration analysis*. in *MATEC Web of Conferences*. 2017. EDP Sciences.

[9] Jelani, K., et al. *Dynamics investigation on motorcycle chassis based on Finite Element (FE) modelling and updating*. in *IOP Conference Series: Materials Science and Engineering*. 2019. IOP Publishing.
[10] Abdullah, N., M. Asri, and M.S.M. Sani, *Strategies of Finite Element Modeling for Spot Welded Joints and its Modal Correlation with Experimental Data.* International Journal of Automotive and Mechanical Engineering, 2022. **19**(1): p. 9543-9550.
[11] Zaman, I., et al. *The effects of bolted joints on dynamic response of structures*. in *IOP Conference Series: Materials Science and Engineering*. 2013. IOP Publishing.
[12] Zang, C., S. Ma, and M. Friswell. *Structural model updating with an improved parameter selection method [C]*. in *Proceeding of the 25th International Conference on Noise and Vibration Engineering, Leuven*. 2012.