Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

Model Updating of Frame Structure with Bolted Joints

Model Updating of Frame Structure with Bolted Joints

M.H.N. Izham[1] and M.S.M. Sani[1,2*]
[1]Advanced Structural Integrity & Vibration Research (ASIVR), Faculty of Mechanical and Manufacturing Engineering, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia
[2]Automotive Engineering Centre, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia

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

Abstract. Experimental analysis is often viewed as an important source of reference compared to finite element due to the accuracy of giving reliable data. Currently, finite element analysis is widely used as an early adoption in development, hence trusting the finite element data is crucial for the user. Finite element analysis often shows discrepancies to the test result. The complexity of the joints (bolted) might cause the discrepancies to occur. This study aims to reduce the discrepancies between the experimental and numerical analysis on a frame structure with bolted joints by model updating. Model updating is a process of making adjustment to certain parameters of finite element model to reduce discrepancy between analytical predictions of finite element analysis (FEA) and experimental results. Modal properties (natural frequencies, mode shapes, and damping ratio) of a frame structure with bolted joints are determined using both experimental modal analysis (EMA) and finite element analysis (FEA). Both data obtained is correlated before optimising the properties with sensitivity analysis. Joint strategy of this paper is focusing on RBE2, CBAR and CELAS element. CELAS was selected to represent the bolted modelling due to its lowest percentage average of 2.03% compares to CBAR 6.55% or RBE2 3.56%. Selected parameters were identified by performing a sensitivity analysis and the discrepancies was reduced by performing model updating procedure.

1.0 Introduction

Structural systems or space frame structure is build up or assemble of many components. Components are commonly being axial element, with circular or rectangular section which all the components either can resist tension or compression. The joint or mechanical joint on structures plays an important role where it connects two or more components. Joint has a big impact to the structure related to strength which consuming 20-30% of the overall weight of the structure, which making the design is critical to the structure [1]. Engineers commonly used bolted joints as the solution to joint two or more components with the idea of low cost, easy to assemble/dissemble and the most importantly is suit to the design of using bolted joints. At the same time, some engineers are trying to reduce welding joints as good welding joints will need a special skill worker to execute a good joint and to maintain it consistency. Figure 1 shows the frame structure with bolted joints.

Figure 1. Frame Structure with bolted joints

As an early stage of investigation of dynamic characteristics finite element analysis (FEA) is commonly used method in current lifestyle. However, the results of numerical analysis vs experimental analysis have discrepancies [2] is a common topic to be discussed among researchers or engineers. Few ideas of why discrepancies happens on FEA, modelling structures and bolted joints [3] can be a big thing that impacting the results. The frequency of a structure can be significantly affected by the mass that might contribute by the inconsistency of the structure thickness due to manufacturing process. Common mistake by user is using a fixed connection at the bolted joints which causing a modelling error impacting the results [3]. There is other study on few most common errors in numerical studies, such as model structure errors, model parameters errors and model order errors [4].

Modelling a bolted joint is a challenging task where few researchers is focusing on the bolted joints modelling/strategy itself. A common mistake assuming a fixed connection on the bolted joints will lead to modelling error. There are two types of joints strategies which are by 3D modelling or assigning the joints using the FEA element that comes with the FEA software. Few types of modelling techniques are investigated on structure with bolted joints modelling. Consists of solid belt model, couple bolt model, spider bolt model and no bolt model. Experiment was conducted on large marine engine diesel which parts connected with long stay bolts. Among all four modelling techniques, solid belt model is recommended since it gives the best result and looking at computational time which it can save 62% computational time, 21% memory usage compares to solid belt model without scarifying the results badly, coupled bolt model is also recommended [5]. Couples of bolted joint modelling such as solid elements, line element and no bolt are discussed with pro and cons of the joint modelling. Result explains Solid always give a better accuracy, and line elements doesn't produce a bad result either. No bolt or fixed joints has a worse displacement with no properties of bolt been able to define as there is no modelling [6]. Joint element strategy, CFAST element is used for bolted joints, this technique allows the local effect to be modelled in a simple way [7]. A review paper focusing of model updating on joint structure discussed on bolt can be model as spring element (CELAS and CBUSH), connecter element (CBAR and CBEAM) and rigid connection element (RBE2) via NASTRAN software or it can also be modelled as thin layer [8]. Studies also shows, other researchers applied the element as part of their joints strategy with conclusion such as, CBAR and CBEAM are most suitable to use in welded joints compare to CTUBE [9] and spot weld joint modelling is best works with CWELD compare to CBAR and RBE2 [10].

Experimental modal analysis (EMA) has been used since early 1970's [11]. Experimental Modal Analysis (EMA) or also known modal testing or modal analysis is a process to determine the modal properties or vibration characteristic (natural frequency, damping ratio and mode shape) that permits a dynamic mathematical model to be formed. Structure is excited by a force using tool called impact hammer and the output response will be measured by accelerometers with the Fast Fourier Transform (FFT) analyzer, modal parameters will be extracted from the frequency response function (FRF) [11–13].

Model updating is a process to reduce the discrepancies between FEA and EMA. When the discrepancy is high, the predictions are often discussed which either the numerical or experimental setup that cause us a high difference in results. There are few model updated technique been proposed as a model updating method such as Bayesian method [2], sensitivity base model updating [14], Stochastic method by [15] and reduced order characteristic polynomial (ROCP) [16]. One of the iterative method is design sensitivity and optimization algorithm called SOL 200 by NASTRAN [17] and a study is found to utilize SOL 200 performing model updating with the CBAR element to represent the FSW joint [18]. SOL 200 is used as well to update the discrepancies on go kart chassis by updating the sensitive parameters which are young modulus and diameter of tube [19].

This paper aims to reduce the discrepancies between FEA and EMA with the most suitable joint element such as RBE2, CBAR and CELAS taking into considerations when simulating the joint on frame structure. Once suitable joint element is selected, model updating will be carried out using the sensitivity analysis.

2. Frame Structure with Bolted Joints

The structure used in this paper is divided into two which are L bars (4 pcs), straight/flat bars (16 pcs) together with 32 pcs of M6 screws and Nuts. M6 screws are tighten with control value of 8-9Nm. Dimension details of frame structure with the material properties is tabulate in table 1. Stainless steel (SUS304L) is used for this study. Frame structure is bolted to the ground making it as a fixed-free boundary condition. Frame structure model in MSC Nastran/Patran is shown in Figure 2.

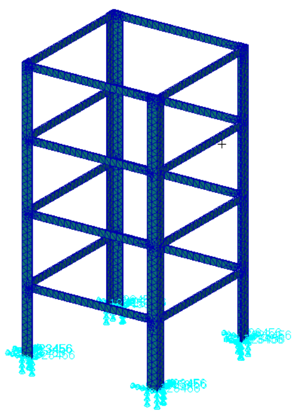

Figure 2. Frame structure of FEA 3D model

Table 1. Dimension and material properties of frame structure

Properties		Value
Material properties (SUS304L)	Young's modulus, E (GPa)	190
	Density (kg/m^3)	8000
	Poisson ratio	0.29
Physical properties	Length (mm)	1004
	Width (mm)	510
	Depth (mm)	510

3. Finite Element Analysis

Boundary condition and external forces were assigned to the model as the model was let to exist in fixed-free boundary condition. Calculation of modal properties in MSC.Nastran/Patran was done by using SOL 103, which is the solution for normal modes analysis. By using the normal modes analysis, the natural frequencies and mode shapes of the structure can be computed. In normal modes analysis, the equation of motion is stated as Eq. (1).

$$[M]\{\ddot{u}\}+[K]\{u\}=0 \quad (1)$$

where K and M are stiffness and mass matrices respectively. These system matrices are computed automatically by MSC Nastran/Patran, based on the geometry and properties of the FEA model. By assuming a harmonic solution, the equation from above can be reduced to an eigenvalue problem which is stated as below

$$[K-\lambda_i M]\{\phi_i\}=0 \quad (2)$$

where $\{\phi\}$ is the eigenvector (mode shape) corresponding to the eigenvalue λ (natural frequency). Each eigenvalue is proportional to a natural frequency and is corresponding to eigenvector. The eigenvalues are related to the natural frequencies as

$$f_i=\sqrt{(\lambda_i)}/2\pi \quad (3)$$

4. Experimental Modal Analysis

Experimental modal analysis is also known as modal testing or modal analysis. This is the process where dynamic characteristic will be collected. The excitation method use in this paper is impact hammer with roving tri-accelerometer. Below Figure 3 is showing the equipment used for impact hammer.

Figure 3. Impact hammer equipment

The portal frame structure is assigned and labelled into 60 grids points of excitation. VES. Figure 4 displays the points assigned on the frame structure with MEscope

IOP Conf. Series: Journal of Physics: Conf. Series **1262** (2019) 012023 doi:10.1088/1742-6596/1262/1/012023

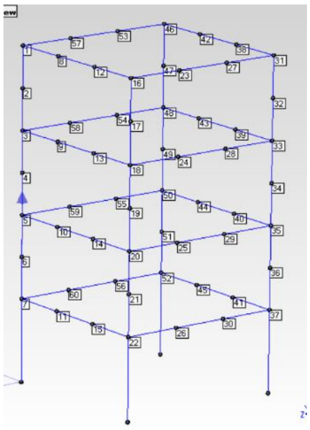

Figure 4. Frame structure with labelled grid points.

5. Data Correlation

Data obtained from the FEA and EMA is correlated to analyze the discrepancies, where discrepancies between both results will be compared and the percentage error will be captured. It has been discussed before, the results of FEA often have discrepancies between the experimental [2]. Three joints elements have been taken into considerations to select the most reliable element to represent bolted joints on frame structure. Table 2 summarized the average percentage of error for CBAR, RBE2 and CELAS.

Table 2. Percentage of error between EMA and FEA for 3 types of joint modelling

Modes	Natural Frequency (Hz)						
	EMA	FEA					
		CBAR	Error (%)	RBE2	Error (%)	CELAS	Error (%)
1	39.6	38.69	2.30	38.99	1.54	39.45	0.38
2	71.7	68.21	4.87	68.25	4.81	70.65	1.47
3	84.4	74.06	12.25	87.49	3.66	80.47	4.66
4	90.4	84.28	6.77	94.24	4.24	91.84	1.59
Average Error (%)			6.55		3.56		2.03

The results reveal that, RBE2 would not be suitable for updating since RBE2 does not present any material properties. Unlike CBAR and CELAS, both elements contain material properties allowing the updating process to be carried on. Average percentage error for CBAR and CELAS are respectively 6.55% and 2.04%. The objective of this study is to reduce the discrepancies between FEA and EMA, looking at the percentage error, by fact CELAS average percentage error is more attractive hence CELAS is chosen to represent the bolted joint with frame structure to perform updating procedure.

6. Finite Element Model Updating

Model updating is a process adjusting selective parameters in finite element to reduce the results discrepancies between FE and EMA [20]. The target of correlation is to minimize the percentage of error in between FE and EMA. This can be completed by performing the model updating. Approximation subject to the simple first-order Taylor series expansion was used in SOL 200 in MSC.Natran in order to change the vector λ of eigenvalues based on the vector θ of structure updating parameters as stated in Eq. (4).

$$\lambda_{i+1} = \lambda_i + [S_i](\delta\theta) \tag{4}$$

where $[S_i]$ is a sensitivity matrix of i-th iteration, which signify the rate of change of the structural eigenvalues λ_i with respect to changes in $\delta\theta$.
The expression for the eigenvalue sensitivity S_i can be stated as Eq. (5).

$$S_i = \frac{\delta\lambda_i}{\delta\theta} = \phi_i^T \left[\frac{\delta K}{\delta\theta} - \lambda_i \frac{\delta M}{\delta\theta}\right] \phi_i \tag{5}$$

The objective function for the prediction error is defined as Eq. (6).

$$g(x) = \sum_{i=1}^{n} W \left(\frac{\omega_i^a}{\omega_i^e} - 1\right)^2 \tag{6}$$

where ω_i^e and ω_i^a are the experimental and finite element natural frequencies respectively; W is the weighting factor for each mode.

The number of updating parameters was kept to be less than number of modes to be updated in order to avoid ill-conditioning problem in updating procedure. The sensitivity analysis was carried out in the first place in the interest of ensuring that the selected updating parameters were meaningful. The prediction of objective function $g(x)$ in updating procedure was set to obtain the minimized value.

Therefore, the procedure was kept going until convergence was accomplished when the values of $g(x)$ is sufficiently small. Sensitivity analysis is to determine the sensitive parameters to be used for updating the structure with bolted joints. Finite element model updating only works to adjust the uncertainties and assumptions of modelling parameters [21-22]. Table 3 shows the sensitivity matrix for updating.

Table 3. Sensitivity matrix for updating

Output Type	Young Modulus (structure)	Young Modulus (Joint)	Density (structure)	Poison Ratio (structure)
NF1	11.55	2E-08	0.52	39.41
NF2	17.36	1.6E-08	2.49	59.27
NF3	18.23	1.6E-09	0.00046	62.21
NF4	18.33	9.2E-09	0.023	62.57

Based on the facts tabulated by the sensitivity matrix, Young modulus and position ratio of the structure are the most sensitive among all 4 parameters, but not to ignore the density parameter which contributes to reduce the discrepancies between FEA and EMA. Joint young modulus gives an extremely low value; this will not give any significant change to the natural frequency. Table 4 display the error between initial and updated results of FEA.

Table 4. Comparison of discrepancies between initial and updated results.

Mode	EMA (Hz)	Initial FEA		Updated FEA	
		Nat Freq (Hz)	Error (%)	Nat Freq, (Hz)	Error (%)
1	39.6	39.45	0.38	39.92	0.81
2	71.7	70.65	1.47	71.51	0.26
3	84.4	80.47	4.66	81.42	3.53
4	90.4	91.84	1.59	92.91	2.78
Average Error (%)			2.03		1.85

The results above show 3 out of 4 natural frequency improved except for mode 1 natural frequency. Average percentage error reduced from 2.03% to 1.85% and this proved model updating with SOL 200 able to reduce the discrepancies between FEA and EMA. However, Table 5 explain the changes in value of parameter prior to model updating

Table 5. Updated value of parameters.

Parameter	Initial value (i)	Updated value (ii)	Deviation |(ii-i)/i|
Young Modulus of structure (GPa)	190	192.1	0.01
Density of structure (kg/m^3)	8000	7600	0.05
Poison Ratio of structure	0.29	0.287	0.01

7. Conclusion

This study was undertaken to correlate the natural frequency between FEA and EMA on frame structure with bolted joints. Few joint strategies have been introduced to this structure with bolted joints are CBAR, RBE2 and CELAS. The average percentage of error being tabulated, CELAS element is selected as a reliable model for bolted joint and the sensitivity analysis with SOL 200 was executed to improve the discrepancies. Based on the sensitivity matrix, it is found that Young Modulus and poison ratio gives a significant number impacting the natural frequency and as well as the density. The discrepancy between EMA and FEA has been brought down prior to model updating.

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 equipment used for this work Fundamental Research Grant Scheme (FRGS/1/2017/TK03/UMP/02-19) – RDU 170123.

References

[1] Tien T L 1999 Space Frame Structures in Structural Engineering Handbook.
[2] Marwala T, Mdlazi L and Sibisi S 2007 Finite Element Model Updating Using Bayesian Approach *22th Int. Modal Anal. Conf.*, vol. 42 (1) 1-7
[3] He K and Zhu W D 2009 Finite Element Modeling of Structures with L-shaped Beams and Bolted Joints: Model Updating and Predictive Modeling Approaches ASME 133 1–18.

[4] Mottershead J E and Friswell M 1993 Model Updating In Structural Dynamics: A Survey *Journal of Sound and Vibration* 167(2) 347–375.
[5] Kim J, Yoon J C and Kang B S 2007 Finite element analysis and modelling of structure with bolted joints *Appl. Math. Model* 31(5) 895–911.
[6] Montgomery J 2002 Methods for Modeling Bolts in the Bolted Joint *ANSYS User's Conf.* 15.
[7] Yunus M A 2011 Finite Element Modelling and Updating of Structure of Sheet Metal with Bolted and Welded Joints 227.
[8] Zahari S N, Zakaria A A R, Sani M S M and Bujang I 2016 A review on Model updating of joint structure for dynamic analysis purpose MATEC Web of Conferences 74 (00023)
[9] Abdullah N A Z, Sani M S M and I Zaman 2017 Correlation of Structural Modal Properties of Go-kart Frame Structure using Different Type of Joint in Finite Element Modeling *Adv. Sci. Letter* 23(11) 11315–113193.
[10] Nazri N A and Sani M S M 2017 Finite element normal mode analysis of resistance welding jointed of dissimilar plate hat structure *IOP Conf. Ser. Mater. Sci. Eng* 257(1) 012059
[11] Schwarz B J and Richardson M H 1999 Experimental Modal Analysis *J. Sound Vib*.
[12] Avitabile P 2001 Experimental Modal Analysis *J Sound Vib*. 35(1) 1–15.
[13] Larbi N and Lardies J 2000 Experimental Modal Analysis of a Structure Excited By a Random Force *Mech. Syst. Signal Process*. 14(2) 181–192.
[14] Bakir P G, Reynders E, and De Roeck G 2007 Sensitivity-based finite element model updating using constrained optimization with a trust region algorithm *J. Sound Vib*. 305(1-2) 211–225.
[15] Abu Husain N, Khodaparast H H and Ouyang H 2012 Parameter selection and stochastic model updating using perturbation methods with parameter weighting matrix assignment *Mech. Syst. Signal Process*. 32 135–152.
[16] Li W L 2002 A New Method for Structural Model Updating and Joint Stiffness Identification *Mech. Syst. Signal Process* 16(1) 155–167.
[17] Fouzi M S M, Jelani K M, Nazri N A and Sani M S M 2018 Finite Element Modelling and Updating of Welded Thin-Walled Beam International Journal of Automotive and Mechanical Engineering (IJAME) 15(4) 5874-89
[18] Zahari S N, Sani M S M, Husain N A, Ishak M and Zaman I 2016 Dynamic analysis of friction stir welding joints in dissimilar material plate structure J. Teknol. 78
[19] Abdullah N A Z, Sani M S M, Hussain N A, Rahman M M and Zaman I 2017 Dynamics properties of a Go-kart chassis structure and its prediction improvement using model updating approach International Journal of Automotive and Mechanical Engineering (IJAME) 14 3887-97
[20] Mottershead J E, Link M and Friswell M I 2011 The sensitivity method in finite element model updating: A tutorial Mech. Syst. Signal Process. 25 2275–96
[21] 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 90 (01021)
[22] Sani M S M and Ouyang H 2016 Identification of Nonlinearities in Joints of a Wing Structure MATEC Web of Conference 83 (03006)