Finite Element Modelling and Model Updating of Riveted Joint Structure

M A S Aziz Shah1, M A Yunus1*, M N Abdul Rani1, M. A. Zulkofli1
1Structural Dynamics Analysis and Validation, Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM), 40000 Shah Alam, Selangor, Malaysia

*Corresponding author: mayunus@salam.uitm.edu.my

Abstract. Nowadays, the applications of rivet joints are dramatically increasing because of its capability and efficiency to assemble the engineering structure such as aerospace and mechanical structures. However, every joint such as riveted joint has a significant effect to the overall dynamic behaviour of the assembled structure. This is because, the overall dynamic behaviour of the assembled structure is highly relying on the quality of the riveted joints. Generally, the riveted joints always introduce local effects such as surface contact, clamping force and slip to the surface of the assembled structure. Therefore, this paper is attempted to investigate the dynamic behaviour of the riveted joints structure experimentally and numerically. The experimental modal analysis method is used to measure the dynamic behaviour of the assembled structure. Meanwhile, the predicted data of the dynamic behaviour of the assembled structure is obtained by using finite element method. A few types of element connectors that available in finite element commercial software namely such as RBE, CBAR, CBEAM and CELAS have been investigated in order to represent the rivet joints in the assembled structure. In this study, the results obtained from the finite element analysis (in terms of natural frequencies and mode shapes) are then correlated with experimental counterpart in order to identify the level of the accuracy between element connectors that are used in modelling the rivet joints of the assembled structure. Finally, the reconciliation method using finite element model updating is used to minimise the discrepancies of the initial finite element model based on the most appropriate element connector to experimental data. The results reveal that CBEAM element connector has better capability and most appropriate comparing to other three element connectors to represent rivet joint in the assembled structure.

1. Introduction
In the advancement of the engineering structure the riveted joints have become the most implemented connection method to assemble thousands of sheet metal components in engineering industries such as aerospace and automotive structure. In comparison to the others joining methods such as welded, bolted or adhesive. The riveted joints are much more preferred due to its simplicity in joining process and can be available at very low cost [1-2]. One of the advantages of the riveted joints compared to other jointing methods is that the riveted joint has more flexibility than other type of joints where, the riveted joints can be used to assemble components with dissimilar type of materials. Moreover, the implementation of the riveting process is very fast and easy. Moreover, the riveted joints are having very high connections strength and also very efficient to be used in many industries such as in automotive or aircraft structures [3]. Even though rivet joint having a lot of advantages compared to other type of jointing methods, the presents of a local effects in the jointed structure are not easy to be included or
modelled in a large finite element model. This is because the finite element modelling is constructed based on assumptions and simplification on geometry and properties of a structure [4].

For instance, sheet metal substructures and the rivets can easily experience deformation during the riveting process. Furthermore, the quality of the riveted joints structure is depended with axis deviation in which the rivet’s axis must be equivalent or parallel with the hole’s axis of the structure which it may influence the deformation, stress and strain riveting. The simplification on these problems may lead to the inaccuracy in the prediction of the dynamic behaviour of riveted jointed structure. Previous studies suggested that, it is impractical to model the local effects in finite element modelling precisely because it requires a lot of computational efforts when performing analysis [5-6]. Therefore, the experimental method is used to analyse the dynamic behaviour of riveted joints structure namely experimental modal analysis (EMA). This experimental approach is required as to identify inaccuracies of approximation the initial finite element model. The discrepancies of results between experimental modal analysis and finite element analysis have led to the conciliation procedure through which the finite element model is altered in order to provide good agreement with the experiment result [7-9].

In this study, the dynamic behaviour of the riveted joints structure is investigated numerically and experimentally. Element connectors such as rigid body element (RBE), bar based element (CBAR), beam based element (CBEAM) and spring based element (CELAS) are investigated and their accuracy will be compared with the experiment data in term of natural frequencies and mode shapes. After that, finite element model updating will be applied in order to reduce relative error between the initial finite element model of the assembled structure with experimental data by improving the uncertainties of the input parameters and to ensure the validity of the developed finite element model.

2. Experimental Modal Analysis (EMA)

Experimental modal analysis is a technique to define the dynamic behaviours of the structure from a vibration testing, in terms of natural frequencies, mode shapes and damping ratios [10-12]. The result from the experimental data will be compared with finite element model to identify the correlation between its results. If the results from the test structure is within the acceptable tolerance with predicted model, and then finite element model can be used in further analysis with higher degrees of confidence.

In this study, the riveted structure, which consist of two number of components, namely as 1) hat shape and 2) flat shape plate were tested. The hat shape and flat shape plate substructures were connected by 10 rivets with nominal 5 mm diameter for each rivet and were separated by 70 mm apart. The material used to fabricate the substructures of the structure are mild steel with overall dimension was 300 mm length, 240 mm wide and 5 mm thickness.

The experimental modal analysis was performed to the substructures and assembled structure by using impact hammer and roving accelerometers method as to measure the dynamic behaviour such as natural frequencies and mode shapes. Prior to the measuring process of assembled structure, the structure was setup under free-free boundary conditions whereby four sets of rubber band and strings were used to hang the structure from the specially designed clamps by attaching the strings to the holes on the structure and the other ends of the strings to the clamps. Meanwhile, the frequency bandwidth of interest was set from 1 – 400 Hz. Finally, all the data obtained were processes using LMS SCADAS data acquisition system. The experiment procedures were illustrated in Figure 1, Figure 2 and Figure 3.
3. Finite Element Modelling and Analysis

Finite element analysis is a numerical method that involve with the discretisation of the physical domain and the geometry of the structure into elements and in this work, the structure was designed as hat shape and flat shape plate connected by rivets joint.

In the pre-processing stage of the finite element modelling, the MSC PATRAN was used to develop the finite element model of the components and the assembled structure. The components and the assembled structure were modelled using CQUAD4 elements (shell) and the models were meshed into 5 mm meshing size based on the suitability of the meshes size recorded from mesh convergence test. In this study, four types element connectors namely rigid body element (RBE), bar based element (CBAR), beam based element (CBEAM) and spring based element (CELAS) were explored to represent the rivet joints in the modelling of the riveted joined structure. The material properties used in the finite element modelling was based on nominal properties of mild steel is listed in Table 1.
### Table 1. Material properties of the laser spot welded structure [13]

| Component     | Property          | Value | Unit     |
|---------------|-------------------|-------|----------|
| Hat shape plate | Young's Modulus  | 210   | GPa      |
|               | Poisson's Ratio   | 0.30  | Unitless |
|               | Mass Density      | 7700  | kg/m$^3$ |
| Flat shape plate | Young's Modulus  | 210   | GPa      |
|               | Poisson's Ratio   | 0.30  | Unitless |
|               | Mass Density      | 7700  | kg/m$^3$ |
| Rivet         | Young's Modulus   | 70    | GPa      |
|               | Poisson's Ratio   | 0.30  | Unitless |
|               | Mass Density      | 2770  | kg/m$^3$ |

In this research, normal mode analysis was calculated by using the normal modes analysis of MSC NASTRAN to identify the natural frequencies and mode shapes of the finite element models of the components and the assembled structure. The equation of motion that are used to discretise the system to a finite element model, also known as 2nd order differential equation is given as

$$M\ddot{x}(t) + C\dot{x}(t) + Kx(t) = f(t)$$

where $M$, $C$ and $K$ are symmetric matrices of mass, damping and stiffness. Meanwhile $\dot{x}$, $\ddot{x}$ and $x$ represent the vector of accelerations, velocities and displacement respectively and $f(t)$ is vector of external forces. The structure that been used in this research were considered having light damping and the effect of damping can be theoretically neglected in finite element modelling. As a result, for the undamped free vibration analysis, the equation (1) can be simplified as

$$M\ddot{x}(t) + Kx(t) = 0$$

The equation (2) can be solved by assuming the harmonic solution in the form of

$$x = \phi sin\omega t$$
where $\omega$ and $\phi$ are the mode shape and natural frequency of the system. If the differentiation of the assumed harmonic solution is performed and substituted in equation (2), the equation of motion yields and simplified to the following

$$ (K - \omega^2 M) \phi = 0 \quad (4) $$

The natural frequencies and mode shapes of the laser spot welded structure can be predicted by solving the equation (4) using finite element commercial software such as MSC NASTRAN.

4. Finite element model updating

Model updating is one of the methods used to improve the correlation of finite element models and the test structure by correcting the invalid assumptions of the model to an acceptable level of accuracy and the process is carried out by tuning some uncertain model parameters systematically. However, it is difficult to select the right parameters that sensitive to the change of modal properties. This is because the selection of the parameters should be justified by engineering understanding of the structure and the number of selected parameters should be kept to a minimum in order to avoid ill-condition problems.

The selection of the most influential parameters was performed using sensitivity analysis and can be expressed as,

$$ S = \phi_i^T \left[ \frac{\partial K}{\partial \theta_j} - \lambda_i \frac{\partial M}{\partial \theta_j} \right] \phi_i $$

where, $S$ shows the sensitivity matrix, $K$ and $M$ are the stiffness and mass matrices respectively, while $\phi$, $\lambda$ and $\theta$ represent eigenvector, eigenvalue and parameter respectively. Furthermore, $i$ indicates the $i$-th eigenvalue and $j$ is the $j$-th parameter.

In order to perform model updating, the NASTRAN optimisation code (SOL 200) is used in which it allows the objective function ($J$) to be minimised by adjusting the eigenvalues of the initial finite element model until the objective function is converged. The objective function can be expressed as,

$$ J = \sum_{i=1}^{n} W_i \left( \frac{\lambda_i^{fe}}{\lambda_i^{exp}} - 1 \right)^2 $$

where, $\lambda_i^{exp}$ is the $i$-th experimental eigenvalue and $\lambda_i^{fe}$ is the $i$-th predicted eigenvalue from the finite element model and $n$ is the number of eigenvalues involved in the updating procedure.

5. Results and Discussion

In this study, the first five natural frequencies and mode shapes of the riveted joints structure were calculated experimentally and numerically. The accuracy of the finite element models of the structure are using four different types element connectors which to represent the rivet joint namely RBE2, CBAR, CBEAM and CELAS was investigated. The results shown in Table 2 consists the comparisons between experiment data and initial finite element data. From the table, it shows that CELAS based FE model obtained the most relative total error which is 320.72%, meanwhile total relative error of RBE based FE model is 21.86%, CBAR based FE model is 31.3% and the less total relative error is CBEAM based FE with 31.22 %. The results in Table 1 clearly shows that the CELAS based FE model is the highest relative total error. The results suggest that the CELAS element connector is not suitable to be represent as rivet joints. This is because CELAS element cannot produce adequate rigidity to the assembled structure. Furthermore, the inaccuracy in the finite element results is happen due to the influence of the local effects in the rivet joints that correspond badly to the sheet metal structure [14-16]. Moreover, the invalid assumptions of the input parameters on the structure such as Young’s
modulus, Poisson’s ratio, sheet metal thickness and diameter of rivets also effect the dynamic behaviour of the riveted joints structure.

Based on the result in Table 2, RBE based FE model has shown the lowest error in comparison with the other element connectors. However, CBEAM based FE model was selected as the most appropriate element connector to represent riveted joint in the assembled structure. This is because RBE element connector do not have any material properties and on top of that, the RBE properties is based on infinite stiffness configuration [17-19]. This shortcoming makes RBE based FE model not suitable to be used in model updating. Therefore, CBEAM based FE model is more preferable. In order to reduce discrepancies of the CBEAM based FE model with experimental data counterpart, finite element model updating was performed by minimising the objective function with the experimental data was used as benchmark. The result of the dynamic behaviour of the updated finite element model of the structure is shown in Table 3. The result shown that model updating method managed to reduce the total relative error of the CBEAM based FE model from 31.22% to 16.44%. The modal assurance Criterion (MAC) values of the updated finite element model also shown a good improvement.

Table 2. Comparison different types of element connectors.

| Mode | Test | RBE FE (Hz) | CBAR FE (Hz) | CELAS FE (Hz) | CBEAM FE (Hz) |
|------|------|-------------|--------------|---------------|---------------|
|      |      | Error (%)   | Error (%)    | Error (%)     | Error (%)     |
| 1    | 153.72 | 149.95     | 2.45         | 146.68        | 4.58          | 146.73        | 4.55          |
| 2    | 159.78 | 157.38     | 1.50         | 156.24        | 2.22          | 156.24        | 2.22          |
| 3    | 228.74 | 224.13     | 2.02         | 220.65        | 3.54          | 220.86        | 3.45          |
| 4    | 321.47 | 291.09     | 9.45         | 284.40        | 11.53         | 284.65        | 11.45         |
| 5    | 331.34 | 310.01     | 6.44         | 300.10        | 9.43          | 299.67        | 9.56          |
|      | Total Error | 21.86     | 31.30        | 320.72        | 31.22         |

Table 3. Comparison between the measured, predicted and updated results of the structure.

| Mode | Test | Initial CBEAM FE (Hz) | Error (%) | MAC | Updated CBEAM FE (Hz) | Error (%) | MAC |
|------|------|-----------------------|-----------|-----|-----------------------|-----------|-----|
|      |      |                       |           |     |                       |           |     |
| 1    | 153.72 | 146.73                | 4.55      | 0.85| 149.85                | 2.52      | 0.92|
| 2    | 159.78 | 156.24                | 2.22      | 0.81| 158.14                | 1.03      | 0.90|
| 3    | 228.74 | 220.86                | 3.45      | 0.84| 224.17                | 2.00      | 0.89|
| 4    | 321.47 | 284.65                | 11.45     | 0.74| 301.91                | 6.09      | 0.88|
| 5    | 331.34 | 299.67                | 9.56      | 0.79| 315.42                | 4.80      | 0.88|
|      | Total Error | 31.22     |             |     | 16.44                 |           |     |

6. Conclusions

In this paper, the dynamic behaviour of a riveted joints structure was analysed experimentally and numerically. The main objective was to identify the most appropriate element connector to represent the rivet joint accurately in the assembled structure. The discrepancies of the initial finite element model and the test structure was determined by comparing natural frequencies and mode shapes of the structure. The result shows that CBEAM element was the best element connector to represent as rivet joints in the assembled structure. Therefore, the initial CBEAM based FE model was used in finite element model updating method to reduce the discrepancies with experimental data. Based on the result obtained, it is recommended to use CBEAM element connector to represent as rivet joints. However, there is still an improvement that can be made in the future such as the inclusion of local effects in the finite element modelling that may improve the prediction result of the assembled structure.
Acknowledgments
The authors wish to acknowledge the Malaysia Ministry of Higher Education (MOHE), Research Management Institute (RMI) and Institute of Graduate Studies of Universiti Teknologi MARA (UiTM) for providing financial support for this research through the fundamental research grant scheme (FRGS) 600-IRMI/FRGS 5/3 (172/2019). They would also like to extend their sincere gratitude for the support and help given by, Prof. Hadariah Bahron and all SDAV members.

References
[1] A. L. L. da Silva et al., “Fatigue characterization of a beam-to-column riveted joint,” *Eng. Fail. Anal.*, vol. 103, pp. 95–123, Sep. 2019.
[2] L. Huang, H. Guo, Y. Shi, S. Huang, and X. Su, “Fatigue behavior and modeling of self-piercing riveted joints in aluminum alloy 6111,” *Int. J. Fatigue*, vol. 100, pp. 274–284, Jul. 2017.
[3] M. A. Yunus, M. N. A. Rani, M. S. M. Sani, and M. A. S. A. Shah, “Finite element model updating of riveted joints of simplified model aircraft structure,” *AIP Conf. Proc.*, vol. 1952, no. 1, p. 2013, 2018.
[4] M. S. A. Sulaiman, M. A. Yunus, M. N. A. Rani, A. M. Saman, Z. Mohamed, and M. A. S. A. Shah, “Characterisation of the dynamic characteristics of a complex jointed structure,” in *MATEC Web of Conferences*, 2017.
[5] M. A. S. A. Shah, M. A. Yunus, A. M. Saman, M. N. A. Rani, A. R. Bahari, and C. Peter, “Finite element model updating of laser stitch welded structure,” *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 506, p. 012010, 2019.
[6] H. H. Khodaparast et al., “Fuzzy finite element model updating of the DLR AIRMOD test structure,” *Appl. Math. Model.*, vol. 52, pp. 512–526, Dec. 2017.
[7] Ø. W. Petersen and O. Øiseth, “Sensitivity-based finite element model updating of a pontoon bridge,” *Eng. Struct.*, vol. 150, pp. 573–584, Nov. 2017.
[8] V. Panwar, P. Gupta, A. K. Bagha, and N. Chauhan, “A review on studies of finite element model updating and updating of composite materials,” *Mater. Today Proc.*, vol. 5, no. 14, pp. 27912–27918, Jan. 2018.
[9] M. A. S. Aziz Shah, M. A. Yunus, M. N. Abdul Rani, N. A. Tormidi, A. A. Mat Isa, and W. I. I. Wan Iskandar Mirza, “Closed mode separation of the symmetrical structure,” *J. Mech. Eng.*, vol. 5, no. 3, pp. 88–97, 2018.
[10] M. A. Yunus, M. A. S. Aziz Shah, M. N. Abdul Rani, W. I. I. Wan Iskandar Mirza, and S. Kushairi, “Experimental and finite element analysis of laser stitch welded structure,” *J. Mech. Eng.*, vol. 4, no. 3, pp. 155–162, 2017.
[11] F. Adel, S. Shokrollahi, M. Jamal-Omidi, and H. Ahmadian, “A model updating method for hybrid composite/aluminum bolted joints using modal test data,” *J. Sound Vib.*, vol. 396, pp. 172–185, May 2017.
[12] J. M. W. Brownjohn, A. Raby, J. Bassitt, A. Antonini, E. Hudson, and P. Dobson, “Experimental modal analysis of British rock lighthouses,” *Mar. Struct.*, vol. 62, pp. 1–22, Nov. 2018.
[13] M. A. Yunus, “Finite element modelling and updating of structure of sheet metal with bolted and welded joints,” University of Liverpool, 2011.
[14] J. Q. Yin, Z. Q. Wang, Y. G. Kang, and N. Li, “Finite element analysis of axis deviation between rivet and plate hole,” *Appl. Mech. Mater.*, vol. 321–324, pp. 188–191, 2013.
[15] X. C. He, “Sheet material property effects upon dynamic behavior in self-pierce riveted joints,” *Mater. Sci. Forum*, vol. 675–677, pp. 999–1002, 2011.
[16] S. M. S. Hassin.M,N, Yunus.M.A, Abdul Rani.M,N, “Experimental and finite element analysis of riveted joints structure of a simplified model of aluminium crash box,” *J. Eng. Appl. Sci.*, vol. 11, pp. 1451–1455, 2016.
[17] N. M. Hassin, M. A. Yunus, A. Radzi, A. Ghani, N. A. Rani, and S. Kasolang, “Using updated model for the crash analysis,” *Int. Congr. Sound Vib.*, pp. 1–8, 2016.

[18] 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

[19] 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