INTERNATIONAL JOURNAL OF AUTOMOTIVE AND MECHANICAL ENGINEERING (IJAME)
ISSN: 2229-8649 e-ISSN: 2180-1606
VOL. 19, ISSUE 1, 9543 – 9550
DOI: https://doi.org/10.15282/ijame.19.1.2022.17.0736

ORIGINAL ARTICLE

Strategies of Finite Element Modeling for Spot Welded Joints and its Modal Correlation with Experimental Data

N.A.Z. Abdullah[1], M.N.A.M. Asri[1] and M.S.M Sani[1,2,*]

[1]Advanced Structural Integrity & Vibrational Research (ASIVR), Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.
[2]Automotive Engineering Centre, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.

ABSTRACT – In building many complex engineering structures, there are many types of joining methods such as welding and fasteners that can be implemented. Modeling for joints in finite elements can be challenging as it sometimes has limiting factors that cause the prediction of the dynamic behaviour of the actual joints to be less accurate. This study aims to demonstrated several approaches of finite element modeling for spot-welded joints ad to analyse its accuracy through the correlation of modal data from experimental modal analysis. These modeling approaches are created by creating and manipulating the elements at the associated location of the spot weld joint on a top-hat beam structure. Four different approaches of spot weld modeling that uses the modeling strategies performed in other studies were created. The spot weld models are validated by comparing the modal properties of the tested structure which are obtained through finite element analysis and experiments. Model updating was performed on all models in order to observe the ability of model improvement in those different modelling approaches. The findings show that the model that uses solid elements has the lowest error compared to the model that uses beam elements. The model that uses multiple-beam elements shows the ability to be improved the most. The model that uses the simplest modeling approach using a single beam has the highest error and shows the lowest improvement after model updating. It was found that solid element is more suitable to model spot weld and the application of solid element for spot weld joints should be investigated in more types of analyses.

ARTICLE HISTORY
Received: 29th Sept 2021
Revised: 5th Feb 2022
Accepted: 4th Mar 2022

KEYWORDS
Top-hat structure;
Spot weld joint;
Modal analysis;
Model updating;
Joint modeling

INTRODUCTION

A spot-welded joint is produced by pressing together two metal sheets by electrodes with an electric current is passing through them. When proper welding parameters are applied, a weld nugget with a certain diameter will be created between those two metal sheets to connect them together and establish the joint [1]. The application of spot welding on sheet metal fabrication has been used in industry for decades. It is very important to have a reliable method for modeling spot weld connections as they are widely applied in many structures. Therefore, an efficient numerical method that accurately predicts its structural performance and integrity is highly needed.

Many approaches of modeling joints such as spot-welded joints for finite element analysis have been introduced in previous studies. As spot weld is very common in structure, the study of spot weld model is used in many types of analysis such as structural durability [2], structural dynamics analysis, weld failures and fatigues [3]–[6], optimization study and structural crash analysis. There are many proposed methods of modeling the spot weld for these many types of finite element analysis.

In the most common method to model the weld joints, the application of rigid elements was used [7], [8]. This method is mainly justified through the location of spot welded joints in actual structure only without any additional properties or character to define the joint behavior [9]. This rigid element implementation can be in either bar, shell or solid form. In addition, for rigid bar or element, it can be very difficult for large-scale analysis. This method needs the nodes to be aligned in a good manner so that the link representing the spot weld can be modelled at the correct position.

Thus, beam elements that enable the specification of properties on its elements are used for spot weld modeling are introduced in other studies [10], [11]. These elements were also used to model spot welds for studying the behavior of the weld strength [10], [12]. The application of beam elements was sometimes enhanced with different configurations. The simplest modeling method is the single beam model with the same diameter as the weld nugget. Other studies also demonstrate the application of multiple beam model, spoke beam/ umbrella beam model and spider web model [1]. For the spoke beam and spider web model, this method usually combined the beam element with shell and rigid element and therefore contain properties and parameters that can demonstrate the spot weld behavior in finite element analysis [13]. Another method of representing the spot weld using beam element is by joining together the element corresponding to nugget area by rigid body element. Then the rigid body elements are joined together by beam element. Even so, this method cannot capture stress concentration effectively. Although it was considered to increase the simulation cost and

***CORRESPONDING AUTHOR** | M.S.M. Sani | ✉ mshahrir@ump.edu.my

time, shell and solid element is also used to model the spot weld nugget [5]. Some studies proposed the application of spider web mesh on shell element with elastic beam element with six degrees of freedom is used to connect the center nodes of local spider web shell meshes [13].

The spot weld model that uses solid elements is often used to analyse localized deformation as the geometry and its hardness gradient can be considered. This approach is also suitable for crash analysis as spot weld rupture can be studied [4]. Solid elements are considered as a more complicated but more realistic strategy to represent a weld nugget. The model of spot weld usually is created by using multiple solid elements placed at the nugget boundary and coupled to surrounding shell elements that represent the sheets that are welded together. It is suggested that mesh congruency at the weld nugget region is required and a fine mesh is generated with global mesh modification for better accuracy especially when the deformation and stress fields near the spot weld area is studied [13]. Usually, the solid elements associated with this approach is hexahedral. One or several hexahedral elements used in assembly can be created to represent a connection.

Some finite element software packages also have specific elements that can be used to model joints. Some studies have demonstrated the applications of connector element such as CBAR, CBEAM, CELAS and CWELD that are available in MSC. Patran. These modeling strategies were tested out and correlated by using modal data in order to verify its accurateness [7], [14]–[19]. A review by Palmonella et al. [20] discussed several different types of spot weld modeling and its application in the structural dynamics field. Some of them are the ones discussed earlier: the beam models (rigid and elastic beam), brick model or solid model, and umbrella model, which combine beams and shell elements. There are also ACM1 (area contact model 1) and ACM2 (area contact model 2) models which are using rigid elements. Also, there are two types of Salvini model. The first one consists of a circular clamped plate that simulates the welded plates and the other use a central beam that connect two plates and two sets of a radial beams: one pinned to the rest of the structure and one connected through two offsets to the central beam and the rest of the structure. CWELD model is a special shear flexible beam-type element with two nodes and 12 degrees of freedom.All these types of joint modeling (rigid element, beam element and solid element) were also discussed in other studies [21]–[24] with an additional model introduced which is the constraint model. The constraint model is created by constraint formulation rather than by elements assembly. However, these specific elements that can be used for modeling spot-welded joints are incompatible in several types of simulation such as buckling analysis. Buckling analysis for crash box structure is considered a popular topic of study for studying the crashworthiness performance of constructed crash box structure.

This paper aims to create a feasible model of spot-welded joints using finite element modeling and perform model updating through modal data of the tested structure. The model of the spot weld joints considers the application of the model to be analyzed not limited to a single software package only. Although only the dynamic behaviour of the model is studied in this work, the modeling methods used in this study can be applied not only for dynamic analysis but also for other simulations such as crash analysis.

FINITE ELEMENT MODELING AND ANALYSIS FOR SPOT-WELDED JOINTS

A top-hat beam structure with spot-welded joints was studied in this work. Several modeling approaches for spot weld joints that were mentioned in previous studies are created in this study. The application of one-dimensional elements such as beam elements, rigid and solid elements are mentioned previously. Consequently, four different approaches of modeling the spot weld nuggets are created. Figure 1 illustrates the details of the modeling approaches. Model A are modelled with CBAR standard MSC Nastran/Patran one dimensional element while in model B, the CBAR elements are enhanced with rigid elements (RBE2) around the nugget area. The definition of cross-sectional diameter for beam elements (CBAR) in both model A and model B is 5 mm, referring to the actual nugget diameter on real counterpart. On the other hand, multiple CBAR elements that are enhanced with rigid elements (RBE2) are used in model C. The cross-sectional diameter for each CBAR element in this model is 1 mm and they are placed at the centre and around the spot weld nugget area which has been assigned to be 5 mm according to its real counterparts. Therefore, the multiple beam elements are still located around the location of the welded area. Model D uses solid elements with tet-mesh while creating cylindrical shape representing the diameter of the spot weld nugget.

The real top-hat beam counterpart consists of 20 spot weld nuggets (10 on each flange) so the spot weld models that are explained above are created at the same location as their counterparts. Figure 2 shows the complete finite element model for the top-hat beam. The material properties assigned on the model are as illustrated in Table 1 which adopt the properties of six series aluminum alloy (AA6061). The crash box structure (excluding the elements constructed for joinings) is constructed using shell elements with an assigned thickness of 1.5mm. The number of elements and nodes constructed while considering the meshing size convergence. As for the joining elements, the high value of Young's modulus was assigned in order to provide the model with the stiffness of actual joints. The value of Young's modulus assigned is 1000 GPa.

Then, the modal properties for each created model were obtained by conducting finite element frequency analysis. The frequency analysis of all constructed models is conducted using finite element software package MSC Nastran/Patran under normal modes analysis (SOL103). All models are analysed under free-free boundary conditions (no loads or boundary conditions are assigned to the models). The first five vibrational modes are studied for analysis and correlations. The correlation of these five modes is done with the data gathered in experimental modal analysis.

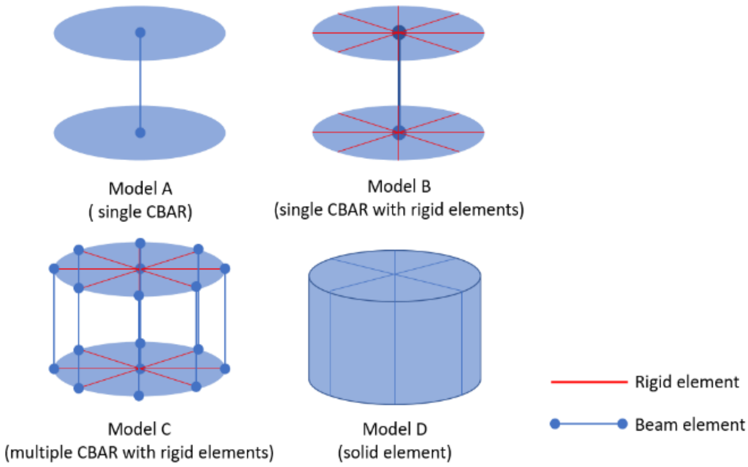


Figure 1. Illustration of spot weld finite element models.

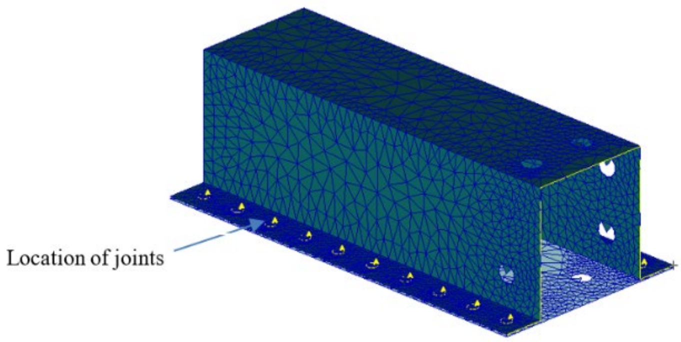


Figure 2. Finite element model of a top-hat beam with welded joints.

Table 1. Material properties (AA6061) assigned to a model of top-hat beam.

No.	Properties	Capability values
1	Young's modulus	68.9 GPa
2	Poisson's ratio	0.3
3	Density	2700 kg/m^3

EXPERIMENTAL MODAL ANALYSIS FOR DATA CORRELATION

Experimental modal analysis on the real counterparts was performed using impact hammer testing. The real top-hat beam was hung on elastic cables from the test rig in order to simulate the free-free boundary condition (see Figure 3). The impact hammer test was conducted by using a roving hammer set up while using two uniaxial accelerometers to record the vibrational signal from the structure. In order to simulate the mode shapes obtained through this experiment, an experimental model was created. There were 54 measurements points created on the experimental model. The correlation with the modal data obtained in finite element analysis and experimentally are tabulated in Table 2.

As depicted in Table 2, there is a significant error value for earlier mode correlation. For the first mode, the error is 8.14%, 14.02%, 14.16% and 13.1% for models A, B, C and D respectively. The correlation for the first mode shows very high error and therefore cumulating a high value of total error for all modes. Model A shows 24.95% of cumulative error and 4.99% of average error for each mode. This is the lowest value of error among all models. On the other hand, model C shows the highest value of total and average error which are 36.96% and 7.39% respectively. Based on the correlation, model A shows the highest potential as the most accurate model for modeling the spot weld joint based on the initial assumption of all the assigned properties to the model.

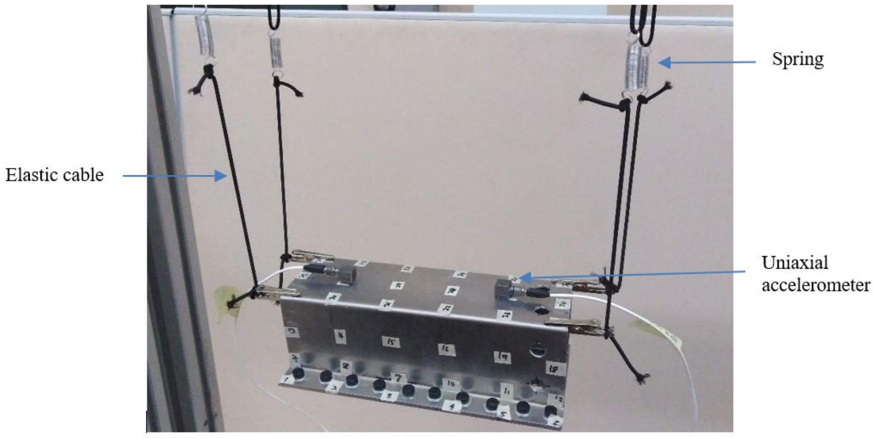


Figure 3. Tested top-hat structure under free-free boundary condition.

Table 2. Initial correlation of natural frequencies for all crash box models.

Mode	Natural frequencies (Hz)								
	Experimental	Model A	Error (%)	Model B	Error (%)	Model C	Error (%)	Model D	Error (%)
1	377	407.7	8.14	429.85	14.02	430.39	14.16	426.40	13.10
2	412	822.25	1.26	847.75	3.94	848.44	4.49	843.97	3.94
3	976	919.85	5.75	964.92	2.17	968.95	0.72	954.83	2.17
4	1180	1113.86	5.61	1288.62	7.64	1294.35	9.69	1270.18	7.64
5	1230	1177.27	4.29	1326.52	5.55	1327.28	7.90	1298.26	5.55
	Cumulative error (%)		24.95		33.32		36.96		32.40
	Average error (%)		4.99		6.66		7.39		6.48

FINITE ELEMENT MODEL UPDATING FOR REDUCING ERROR

Sensitivity of Updating Parameters

The model updating procedure was used to correct the less accurate the initially assigned properties. The process was conducted using MSC Nastran SOL 200 coding. The sensitivity coefficient for structural properties was calculated by using Eq. (1) below:

$$S_i = \frac{\delta\lambda_i}{\delta\theta} = \phi_i^T \left[\frac{\delta K}{\delta\theta} - \lambda_i \frac{(\delta M)}{\delta\theta}\right]\phi_i \tag{1}$$

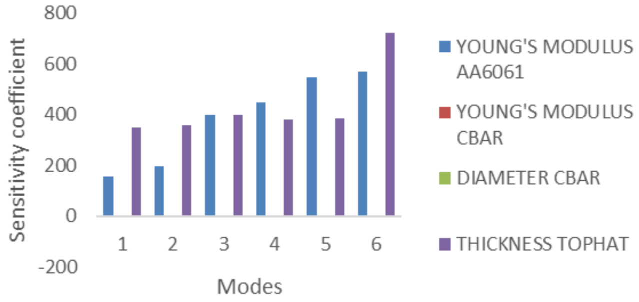
where S_i is the sensitivity matrix that represents the rate of change of eigenvalues λ_i with respect to the changes in parameter $\delta\theta$. When the parameters with a high value of sensitivity coefficients were obtained, the model updating procedure was conducted by setting an objective function as Eq. (2) to obtain the minimized value.

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

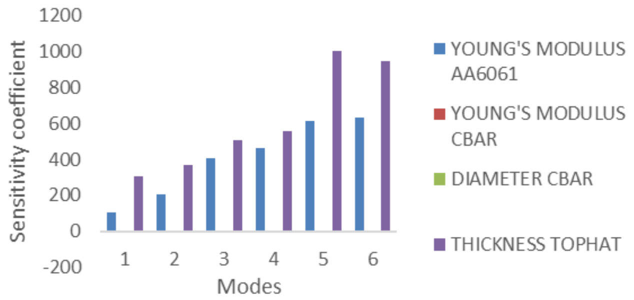
ω_i^e and ω_i^a are the value of natural frequencies obtained in experimental and finite element analysis respectively while W is the weighing factor for each mode. There are two main parameters that are associated with spot weld modeling. The first one is Young's modulus value. Initially, Young's modulus value was assigned as a high value in order to model the joint's stiffness. However, the exact stiffness value that is suitable to be assigned in order to replicate the actual stiffness of the real spot weld joints is uncertain. By conducting the model updating, an optimized value for the joint stiffness can be obtained. The other parameter in concern is the diameter of the spot weld. As diameter for the joining elements is assigned as properties in model A, model B and model C, it can be considered as an uncertain parameter too. Therefore, the sensitivity coefficient value for both of these parameters needs to be studied. The other parameters for the top-hat structure which are Young's modulus of the top-hat material (AA6061) and the top-hat thickness is included in the parameter sensitivity study. The thickness is included as possible parameters due to the variance of thickness reading taken from top-hat. This variance of thickness is possible due to the forming process of top-hat shape from sheet metal which undergoes shearing and bending process

The selection of sensitive parameters is based on the larger magnitude of the sensitivity coefficients as explained in references [8] and [25]. Figure 4 shows all the sensitivity coefficient generated based on the sensitivity analysis for all models (model A, B, C and D). Based on the figures, the most sensitive parameters for all models are Young's modulus of AA6061 and the top-hat thickness. These two parameters are included as updating parameters are not associated with the joints. Joint's parameters that are considered sensitive to be selected as updating parameters are the diameters of

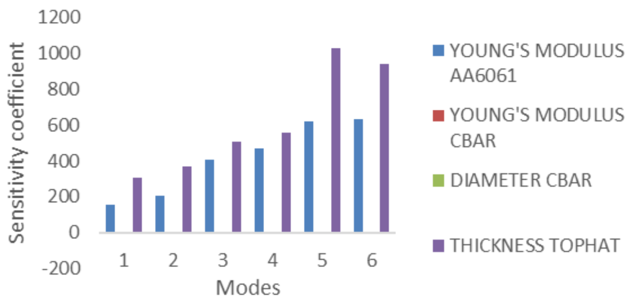
CBAR elements as the connector elements in models B and C. The sensitivity values for the CBAR's diameter in model A are not sensitive enough while in model D diameter of the connector is not a defined property and therefore cannot be updated. The other joint's parameter that is selected as updating parameters is the connector element's Young's modulus. However, this parameter is only sensitive in model D and not in another model.



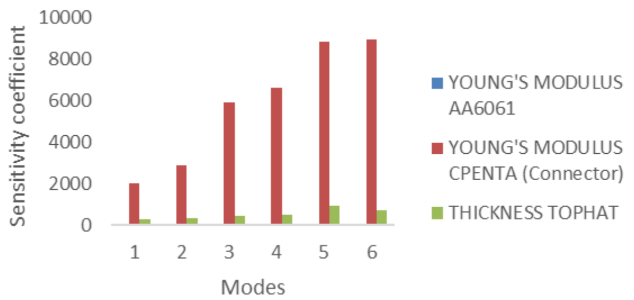
(a)



(b)



(c)



(d)

Figure 4. Parameter sensitivity coefficient for (a) model A, (b) model B, (c) model C, and (d) model D.

Updated Parameters and Modal Properties

As the model updating procedure was successfully applied on each model, the value of analytical natural frequencies for all models were deviated from their initial value. These updated values were obtained when the less accurate properties definition was corrected into a more optimized value. The changes of parameters value from the initial value are depicted in Table 3. From the table, Young's modulus of AA6061 can only be considered as updating parameter only for model A, B and C. As it is not sensitive in model D, there are no big changes on the parameter value and therefore not considered as updating parameter in model D. The Young's modulus of connector elements (CBAR for model A, B and C, and CPENTA for model D) is only sensitive for model D as the updated value the parameter only deviates in model D but not in other models. The connector diameter is only sensitive for models B and C so the value of the properties only deviates in those two models. The top-hat thickness is showing sensitivity in all models and is thus considered as updating parameter for all models. The summary of selected updating parameters is as depicted in Table 4.

Table 3. Changes of parameters value.

Model	Model A		Model B		Model C		Model D	
Updating parameter	Initial value	Updated Value	Initial value	Updated Value	Initial value	Updated Value	Initial value	Updated Value
Young's modulus AA6061 (GPa)	68.9	70.28	68.9	68.21	68.9	68.21	68.9	69.0
Young's modulus connector (GPa)	1000	1000	1000	1000	1000	1000	1000	67.65
Connector's diameter (mm)	5.0	5.0	5.0	5.2	1.0	0.93	NA	NA
Thickness top-hat (mm)	1.5	1.46	1.5	1.4	1.5	1.4	1.5	1.4

Table 4. Summary of updating parameters for each model.

Model	Updating parameter
Model A	Young's modulus AA6061 Top-hat thickness
Model B	Young's modulus AA6061 Connector diameter Top-hat thickness
Model C	Young's modulus AA6061 Connector diameter Top-hat thickness
Model D	Young's modulus connector Top-hat thickness

Table 5 shows the improved correlation of modal data after model updating is applied. Compared to the initial correlation shown in Table 2, the value of cumulative error and average error for all modes for all models are lowered. The cumulative error value is reduced as much as 1.12%, 14.82%, 18.66%, and 16.46% for models A, B, C and D respectively. The reduction of average error is 0.22%, 2.96%, 3.66% and 3.19% for models A, B, C and D respectively. Based on this reduction, the most error reduction is shown in model C while less reduction is shown in model A. Meanwhile, model D shows the lowest nominal value of error for both cumulative and average error. Although initially model A shows the lowest value of error, it was found that the model has the lowest ability to be improved as compared to the other models. In fact, model D which has the lowest value of error after updating can be considered as the best modeling method as it has a good ability to be improved using the model updating method. This error reduction is illustrated in Figure 5.

Table 5. Updated correlation of natural frequencies for all crash box models.

Mode	Updated natural frequencies (Hz)								
	Experimental	Model A	Error (%)	Model B	Error (%)	Model C	Error (%)	Model D	Error (%)
1	377	401.2	6.42	402.22	6.69	402.76	6.83	396.68	5.22
2	412	818.67	0.82	807.69	0.53	808.37	0.45	801.21	1.33
3	976	917.89	5.95	921.42	5.59	925.54	5.17	905.11	7.26
4	1180	1114.5	5.55	1221.34	3.50	1225.04	3.82	1203.24	1.97
5	1230	1167.4	5.09	1256.97	2.19	1259.35	2.39	1221.62	0.68
	Cumulative error (%)		23.83		18.50		18.66		16.46
	Average error (%)		4.77		3.70		3.73		3.29

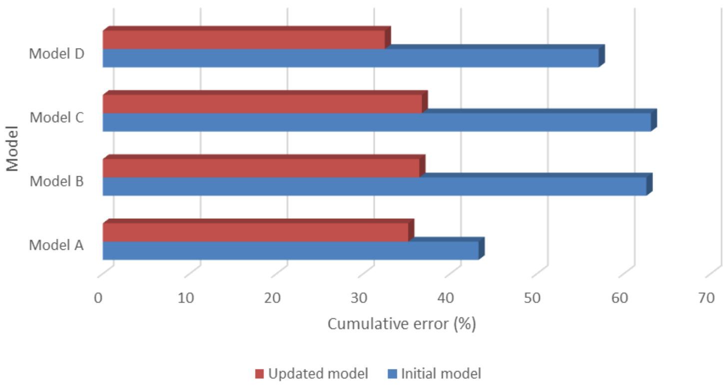


Figure 5. Changes of cumulative error in the initial and updated model.

CONCLUSION

Modeling and its improvement of structure with spot weld joint were carried out in this study by using validation of modal data from experimental modal analysis. Four different modeling strategies were implemented and model updating method was applied to each constructed model. Initially, it was found that the model that uses a single beam element to model the weld joint shows the lowest error and can be considered as the most appropriate model for the spot weld joints. However, after model updating was applied it was found that the application of single beam elements, despite being the simplest and the easiest method of modeling spot weld joints, has the highest error and shows the lowest percentage of error improvement after model updating is applied. In addition, properties of the joint element in this type of model were not selected as updating parameters as they have a low sensitivity coefficient. On the other hand, the model that uses beam elements with enhanced rigid elements shows high error initially but shows the highest error reduction after model updating was applied. It was also found after correlation of modal properties of each model obtained in finite element analysis and experimental modal analysis that the model which uses the solid elements has the lowest error before and after updating. The ability for improvement using model updating was also appropriate. Thus, this type of model can be considered as the most suitable method for modeling the spot weld model in terms of modal correlations. It can be suggested that the application of the modeling method using solid element can be further investigated in ensure its accuracy in other types of analysis.

ACKNOWLEDGEMENT

The authors would like to thank the Ministry of Higher Education for providing financial support under Fundamental Research Grant Scheme (FRGS) No. FRGS/1/2017/TK03/UMP/02/19 (University reference RDU170123) and Universiti Malaysia Pahang for laboratory facilities as well as additional financial support under Internal Research grant PGRS 1903150.

REFERENCES

[1] S. Xu and X. Deng, “An evaluation of simplified finite element models for spot-welded joints,” *Finite Elem. Anal. Des.*, vol. 40, no. 9–10, pp. 1175–1194, Jun. 2004, doi: 10.1016/j.finel.2003.08.006.

[2] Y. Choi, J. Kim, Y. W. Park, and S. Rhee, “Development of a spot weld analysis model that incorporates strain rate,” *Int. J. Precis. Eng. Manuf.*, vol. 13, no. 2, pp. 245–251, 2012, doi: 10.1007/s12541-012-0030-6.

[3] E. H. J. Lamouroux, D. Coutellier, N. Doelle, and P. Kuemmerlen, “Detailed model of spot-welded joints to simulate the failure of car assemblies,” *Int. J. Interact. Des. Manuf.*, vol. 1, no. 1, pp. 33–40, 2007, doi: 10.1007/s12008-007-0006-4.

[4] S. Patil and H. Lankarani, “Numerical Prediction of various failure modes in spotweld steel material,” *SAE Int. J. Transp. Saf.*, vol. 6, no. 1, pp. 29–38, 2018, doi: 10.4271/09-06-01-0003.

[5] S. Bhuvaneswaran and R. Padmanaban, “Prediction of spot weld fatigue life using finite element approach,” *Mater. Today Proc.*, Feb. 2021, doi: 10.1016/j.matpr.2020.12.816.

[6] W. Zhang, W. Jiang, X. Zhao, and S. T. Tu, “Fatigue life of a dissimilar welded joint considering the weld residual stress: Experimental and finite element simulation,” *Int. J. Fatigue*, vol. 109, pp. 182–190, Apr. 2018, doi: 10.1016/j.ijfatigue.2018.01.002.

[7] M. S. M. Fouzi, K. M. Jelani, N. A. Nazri, and M. S. M. Sani, “Finite element modeling and updating of welded thin-walled beam,” *Int. J. Automot. Mech. Eng.*, vol. 15, no. 4, pp. 5874–5889, Dec. 2018, doi: 10.15282/IJAME.15.4.2018.12.0449.

[8] S. N. Zahari *et al.*, “Dynamic analysis of friction stir welding joints in dissimilar material plate structure,” *J. Teknol.*, vol. 78, no. 6–9, 2016, doi: 10.11113/JT.V78.9148.

[9] F. Mainnemare *et al.*, “A numerical analysis to investigate the spot weld local influence,” doi: 10.1051/matecconf/201816521006.

[10] V. Devar, M. Arunadevi, and C. P. S. Prakash, “Finite element analysis of lap joint based on various geometrical parameters to study the behavior of weld strength,” *Mater. Today Proc.*, Mar. 2021, doi: 10.1016/j.matpr.2020.12.1226.

[11] N. A. Z. Abdullah, M. S. M. Sani, and M. S. Salwani, “Study of modal properties and crashworthiness performance of crash box numerical model with different joining modeling strategies,” in *IOP Conference Series: Materials Science and Engineering*, Feb. 2021, vol. 1062, no. 1, p. 012008, doi: 10.1088/1757-899X/1062/1/012008.

[12] M. Zaoui and A. Hadjoui, “Finite element method based modeling of resistance spot-welded mild steel,” *Lat. Am. J. Solids Struct.*, vol. 13, no. 11, pp. 2141–2151, 2016, doi: 10.1590/1679-78252978.
[13] T. Ho-Nguyen-Tan and H. G. Kim, “A novel scheme for modeling and analysis of spot-welded shell structures using interface shell elements,” *Comput. Struct.*, vol. 239, Oct. 2020, doi: 10.1016/j.compstruc.2020.106340.
[14] N. A. Z. Abdullah, M. S. M. Fouzi, and M. S. M. Sani, “Computational modal analysis on finite element model of body-in-white structure and its correlation with experimental data,” *Int. J. Automot. Mech. Eng.*, vol. 17, no. 2, pp. 7915–7926, Apr. 2020, doi: 10.15282/ijame.17.2.2020.10.0591.
[15] R. Omar, M. N. Abdul Rani, and M. A. Yunus, “Representation of bolted joints in a structure using finite element modeling and model updating,” *J. Mech. Eng. Sci.*, vol. 14, no. 3, pp. 7141–7151, Sep. 2020, doi: 10.15282/jmes.14.3.2020.15.0560.
[16] C. Peter *et al.*, “Finite element modeling and analysis for dynamic investigation of a laser spot welded hat-plate structure under initial stress influence,” doi: 10.1088/1757-899X/506/1/012065.
[17] M. S. M. Sani, N. A. Nazri, M. N. A. Rani, and M. A. Yunus, “Dynamic analysis of i cross beam section dissimilar plate joined by TIG welding,” in *AIP Conference Proceedings*, Apr. 2018, vol. 1952, no. 1, p. 020086, doi: 10.1063/1.5032048.
[18] W. I. I. W. I. Mirza *et al.*,, “Correlating finite element model of a car spot-welded front-end module in the light of modal testing data,” *Int. J. Automot. Mech. Eng.*, vol. 17, no. 2, pp. 7974–7984, Apr. 2020, doi: 10.15282/ijame.17.2.2020.16.0597.
[19] S. N. Zahari, M. S. M. Sani, and M. Ishak, “Finite element modeling and updating of friction stir welding (FSW) joint for vibration analysis,” in *MATEC Web of Conferences*, Dec. 2016, vol. 90, p. 01021, doi: 10.1051/matecconf/20179001021.
[20] M. Palmonella, M. I. Friswell, J. E. Mottershead, and A. W. Lees, “Finite element models of spot welds in structural dynamics: Review and updating,” *Computers and Structures*, vol. 83. pp. 648–661, 2005, doi: 10.1016/j.compstruc.2004.11.003.
[21] F. Zhang, H. Xu, and X. Fang, “Failure behavior and crash modeling of resistance rivet spot welding (RRSW) for joining Al and steel in vehicle structure,” *Int. J. Crashworthiness*, pp. 1–18, 2020, doi: 10.1080/13588265.2020.1786269.
[22] J. K. Sønstabø, D. Morin, and M. Langseth, “Static and dynamic testing and modeling of aluminium joints with flow-drill screw connections,” *Int. J. Impact Eng.*, vol. 115, no. November 2017, pp. 58–75, 2018, doi: 10.1016/j.ijimpeng.2018.01.008.
[23] J. K. Sønstabø, D. Morin, and M. Langseth, “Macroscopic modeling of flow-drill screw connections in thin-walled aluminium structures,” doi: 10.1016/j.tws.2016.04.013.
[24] P. K. Taraphdar, R. Kumar, C. Pandey, and M. M. Mahapatra, “Significance of finite element models and solid-state phase transformation on the evaluation of weld induced residual stresses,” *Met. Mater. Int.*, pp. 1–15, Jan. 2021, doi: 10.1007/s12540-020-00921-4.
[25] N. Abu Husain, H. Haddad Khodaparast, and H. Ouyang, “Parameter selection and stochastic model updating using perturbation methods with parameter weighting matrix assignment,” *Mech. Syst. Signal Process.*, vol. 32, pp. 135–152, Oct. 2012, doi: 10.1016/j.ymssp.2012.04.001.