DYNAMIC ANALYSIS OF FRICTION STIR WELDING JOINTS IN DISSIMILAR MATERIAL PLATE STRUCTURE

Siti Norazila Zaharia, Mohd Shahrir Mohd Sani*, Nurulakmar Abu Husain, Mahadzir Ishak, Izzuddin Zaman

aAdvanced Structural Integrity of Vibration Research (ASIVR), Faculty of Mechanical Engineering, University Malaysia Pahang, 26600 Pekan, Pahang, Malaysia
bAutomotive Department, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (UTM), 81310 UTM Johor Bahru, Johor, Malaysia
cFaculty of Mechanical Engineering, University Malaysia Pahang, 26600 Pekan, Pahang, Malaysia
dDepartment of Engineering Mechanics, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia

*Corresponding author mshahrir@ump.edu.my

Graphical abstract

Abstract

Friction stir welding (FSW) is a welding process that is widely used as a solid state joining process for producing welded structure of similar and dissimilar materials such as aluminum alloy, magnesium etc. FSW process has expanded rapidly in industries including aerospace, automotive and maritime due to several advantages compared to other fusion welding. In this paper, experimental modal analysis (EMA) and normal mode finite element analysis (FEA) of the FSW welded joint structure of materials AA6061 and AA7075 will be carried out to identify dynamic properties. Rigid Body Element (RBE2) in MSC NASTRAN/PATRAN is used to model the welds and their compatibility for representing FSW welded structure also being identified. Model updating is performed to minimize the discrepancy between EMA and FEA. Model updating will be acted as an optimization method and is being presented using the structural optimization capability. Finite model updating could be done in individual components and welded structure. RBE2 connecting element can be used to represent friction stir welding with good accuracy.

Keywords: Model updating; friction stir welding; finite element analysis; experimental modal analysis

Abstrak

Kimpalan putaran geseran (FSW) ialah sejenis kimpalan yang digunakan secara meluas sebagai salah satu proses penggabungan dalam keadaan pepejal bagi menghasilkan struktur kimpalan yang terhasil daripada percantuman bahan yang sama dan berbeza seperti aloi aluminium, magnesium dan lain-lain. Proses kimpalan putaran geseran telah berkembang secara mendadak dalam industri termasuklah aeroangkasa, automotif dan maritim disebabkan beberapa kelebihan yang ada pada FSW berbanding proses kimpalan yang lain. Dalam kertas kajian ini, analisis modal secara eksperimen (EMA) dan analisis mod normal unsur terhingga (FEA) dijalankan ke atas struktur (AA6061 dan AA7075) yang dikimpal untuk mengenal pasti sifat dinamik. Unsur jasad tegar (RBE2) dalam perisian MSC NASTRAN/PATRAN digunakan untuk merangka kimpalan dan kesesuaian unsur ini mewakili struktur kimpalan FSW juga dikenalpasti. Pengemaskinian model
1.0 INTRODUCTION

Friction Stir Welding (FSW) is a solid state joining technique, invented at The Welding Institute (TWI) in early 1990's. FSW has expanded rapidly since its first development and is being used widely in automotive, aerospace and maritime industries. FSW managed to catch public attention with the welding of the latest iMac by joining the front and back of computer's ultra-thin (5mm at the edge) enclosure and being remarked as a joining technology that enabled the creation of the "most advanced, most brilliant desktop" in Apple Inc.'s history.

The basic working principle of FSW is easy to implement and the detailed terminology to discuss FSW process was outlined in a paper by Threadgill[1]. A non-consumable rotating tool consisting of pin and shoulder plunges into adjoining edge of the workpiece and traversed along the joint line which resulting in heat generated through both friction and plastic deformation (Figure 1).

![Figure 1 FSW process](image)

FSW shows many advantages when compared to fusion welding. The most remarkable function of FSW is that this process enable alloys, that difficult or impossible to join using other conventional welding, to be welded. FSW also do not require filler material and shielding gas. This characteristics enable FSW to be considered as a "green" technology due to its energy efficiency and environmental friendly behavior. FSW can be applied to numerous types of joint such as square butt joint and lap joint as frequently used joint structure. FSW is the most compatible method for welding light metal alloys, aluminum (Al) based metal.

FSW welded structure do not only provide connection between the two materials but also crucially contribute to stiffness and dynamics properties of the structure. Due to these notable function, it is important to have required physical understanding on the behavior of the welds which can be achieved by numerical and experimental method.

Predictive work such as design, analysis and evaluation of welded structure are usually accomplished by computational method [3]. For a complex structure, it is always not practical to model the weld joints in detail. Only a simple but reliable finite element (FE) representation is needed in order to predict the behavior of welds. There are many previous research focused on modelling the weld joints [3-10]. However, to the author's best knowledge there is no published paper on modelling the dynamic properties of a structure with FSW. Experimental modal analysis (also referred as modal testing) has become a widely known and efficient technique to analyze dynamic behavior of structure. The data that been obtained from modal testing not only can be used to predict dynamic properties but also used as a validation of analytical models before they can be utilized for further detailed analysis.

In automotive, rail transportation and aerospace industries, AA7075 and AA6061 are the two most largely utilized structural material. Therefore for this present study, AA7075 and AA6061 Al alloys will be used as dissimilar materials to investigate dynamic properties of FSW welded structure. Finite element analysis and experimental modal analysis will be explained in the following section. Correlation of two different result will be discussed in next section. Then model updating will be performed to reduce the error between the two results.
2.0 DESCRIPTION OF STRUCTURES

2.1 Materials

The materials used in the study are AA6061 and AA7075 Al alloy flat plate with thickness 2mm. The physical and geometrical properties of these two flat plates are listed in Table 1.

<table>
<thead>
<tr>
<th>No.</th>
<th>Properties</th>
<th>Materials</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Length, ( l ) [mm]</td>
<td>200</td>
</tr>
<tr>
<td>2</td>
<td>Width, ( b ) [mm]</td>
<td>100</td>
</tr>
<tr>
<td>3</td>
<td>Density, ( \rho ) [kg/m³]</td>
<td>2820</td>
</tr>
<tr>
<td>4</td>
<td>Young’s Modulus, ( E ) [GPa]</td>
<td>72</td>
</tr>
<tr>
<td>5</td>
<td>Thickness, ( t ) [mm]</td>
<td>2</td>
</tr>
</tbody>
</table>

Table 1: Material and geometrical properties

2.2 Friction Stir Welding Process

AISI H13 tool steel with cylindrical pin profile was used as a welding tool. The tool consists of shoulder and pin with diameter 17.7mm and 5.80mm respectively. Backing plate and parallel bar were made of mild steel used to support and hold the specimen during welding process. FSW process was carried out using vertical milling machine. All the tools and machine involved in welding process are shown in Figure 2. For this study, plate AA6061 was placed on its advancing side due to its higher mechanical strength and tool pin was positioned at the center of joint line. Process parameters that being used in this study are rotational speed \( \omega = 1100 \) rpm, traverse speed \( v = 50 \) mm/min and tilt angle \( \alpha = 2^\circ \).

2.3 Finite Element Analysis (FEA)

Figure 3 shows the complete FE model of FSW welded structure which is modelled in MSC/NASTRAN using four noded shell elements (CQUAD4). The flat plane consists of 16 elements and 25 nodes. Nominal values for both flat plates are stated in Table 1 above. Neither constraints nor load were assigned to create free-free boundary state. Minimum frequency of 1 Hz was set to avoid the solver from calculating the six rigid body motions that having frequency less than 1 Hz. Rigid Body Element (RBE2) was used to represent the weld joint in the FE model. Five RBE2 elements was inserted with the middle nodes of each plate attached to another in a straight line.

Normal mode analysis (SOL 103) in MSC NASTRAN was performed to compute the modal data of the FSW welded structure. For this analysis, LANCZOS algorithm was used to analyze the plate because element size of the plate is fine and contains many degrees of freedom (DOF). Figure 4 shows five first natural frequencies and mode shapes. The first mode is shown at 146.84 Hz, 220.49 Hz for the second mode, and 252.52 Hz for the third mode, and 335.84 Hz for the fourth mode and 500.39 Hz for the fifth mode. Validation of the FE models are carried out by comparing modal parameters obtained from analytical models with the experimental counterparts that will be explained in the following section.

Figure 2 FSW tool and machining
2.3 Experimental Modal Analysis (EMA)

Modal testing or EMA is used to extract modal parameters such as natural frequency, mode shapes and damping ratio experimentally. For this study, impact hammer had been used as an excitation method. Due to its simple geometry, all flat plate was divided into 25 small grids point that represent the flat plate shape where at this point FRF was measured. A Kistler type uniaxial with sensitivity of 100mV/g used in this experiment was fixated at point 13.
The locations of impact point and measurement point were selected carefully to avoid any nodal points. All the equipments for modal testing illustrated in Figure 5 and the flat plate was supported by sponge in order to simulate free-free boundary condition. The flat plate was tested using one point hammer and one measurement point with only first five modes measured from this set of test. The mass of flat plate and accelerometer were neglected because mass loading effect was very minimal [11]. DasyLab software was used to measure the signal from impact hammer and accelerometer and convert it into FRF. Then, data block collected from DasyLab software will be transferred to MEScope software in order to extract modal parameters using curve fitting method. Natural frequency from the experiment is compared with the natural frequency of FEA and being listed in Table 2 together with the percentage error. In addition, Figure 6 shows the mode shapes extracted from experimental procedure.

Table 2 Natural frequencies of EMA & FEA

| Mode | I (EMA) Natural Frequency (Hz) | II (FEA) Natural Frequency (Hz) | III Error (%) = |(II-I)/I| |
|------|--------------------------------|---------------------------------|----------------|
| 1    | 149.00                         | 146.84                          | 1.45           |
| 2    | 210.00                         | 220.49                          | 4.50           |
| 3    | 255.00                         | 252.52                          | 0.97           |
| 4    | 345.00                         | 335.84                          | 2.66           |
| 5    | 508.00                         | 500.39                          | 1.50           |
From the table, it shows that the percentage error between predicted natural frequencies compared to measured data for less than 1 only occurred for mode 3 meanwhile for mode 1 and 5 are reasonably good with below 1.5 per cent error. Then for the rest of modes (mode 2 and 4), higher error are obtained which are larger than 2 per cent. This can concludes that the FE modelling is not accurate enough to represent the actual FSW structure. Therefore, model updating need to be carried out to reduce discrepancy between test results and prediction counterparts.

2.4 Model Updating Of The FSW Structure

Model updating is one of the methods used to improve correlation between finite element model and experimental results by changing the modeling parameters that have been assigned [12]. It is very crucial step in validation process by altering the values of parameter to make up a definite FE model for additional analysis. There are several methods for model updating [13-16] and for this study, optimization algorithm (SOL200) in NASTRAN software was used to perform model updating.

2.4.1 Formulation Of Updating Procedure

For equation of motion for undamped free vibration of a structure can be expressed as

$$\omega^2 [M] + [K] \ddot{u} = 0$$  \hspace{1cm} (1)

Where M and K are mass and stiffness matrices of the structure and is the modal displacement vector. $\omega_i^2$ is the eigenvalue and $\omega$ is the natural frequency of a structure. Simple first order Taylor series expansion that used in equation for NASTRAN can be expressed as

$$\omega_{n+1}^2 = \omega_n^2 + S_i(\delta\theta)$$  \hspace{1cm} (2)

In above equation, $S_i$ is sensitivity matrix (mxn) that denotes the rates of change of structural eigenvalues...
ω² with respect to the changes in parameters δθ which can be defined as [17]

\[ S_i = \delta \omega^2 \frac{\partial}{\partial \theta} = u_i^T (\delta \frac{\partial}{\partial \theta} - \omega^2 \frac{\partial}{\partial \theta}) u_i \]  

(3)

Therefore from the equation, it shown that any alteration made to the system parameters also can affect the modal properties (natural frequency, damping etc.) of the system. So, the updating parameters and modal properties should be chosen properly for updating procedure.

An error function, J between experimental modal data and analytical modal data is determined for minimization in the updating process. The process will be iterative until convergence is achieved when the error function between the two data is small enough. The error function can be derived from below equation

\[ J = \sum_{j=1}^{n} \left( \frac{\lambda_j}{\lambda_{j \text{ exp}}} - 1 \right)^2 \]  

(4)

Where λₗ is the jth predicted eigenvalue from FE model and λₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑₑ_electrical
4.0 CONCLUSION

This study show that the initial frequencies extracted from analytical method are having some error compare to experimental counterparts when RBE2 element is used to represent the modeling of FSW joint. More importantly in assigning the RBE2 connecting element, it is recommended to put a smaller displacement between the two plates. This is because it will influence the result of FEA. When modelling is settled, the discrepancy between this two results can be reduced using model updating. Before proceed to model updating, sensitivity analysis had been done first to select the right updating parameter. From the sensitivity analysis it shown that both material and geometrical properties play a vital role in model updating in minimizing the error of FE model. After model updating been executed, the error between numerical and test result become smaller. It can conclude that RBE2 can be used to represent FSW and model updating can be used to correct the value of parameter been assigned for FE modeling.

Acknowledgement

This research is fully supported by KTP grant, RDU141011. Special thanks to focus group of Advanced Structural Integrity of Vibration Research (ASIVR) and Universiti Malaysia Pahang for providing all the equipment used for this project. The author also wishes to acknowledge the financial support from Knowledge Transfer Programme (KTP) and Ministry Of Higher Education.

Table 3 Natural frequencies of EMA & FEA

| Mode | I (EMA) Natural Frequency (Hz) | II (FEA) Updated Natural Frequency (Hz) | III Initial Error (%) = |(II-I)/I| | IV Updated Error (%) = |(IV-I)/I| |
|------|-------------------------------|-----------------------------------------|-------------------------|-------------------|-------------------------|-------------------|
| 1    | 149.00                        | 147.20                                  | 1.45                    | 1.21              |
| 2    | 210.00                        | 220.76                                  | 4.50                    | 5.12              |
| 3    | 255.00                        | 253.31                                  | 0.97                    | 0.66              |
| 4    | 345.00                        | 336.97                                  | 2.66                    | 2.33              |
| 5    | 508.00                        | 501.89                                  | 1.50                    | 1.20              |
| TOTAL ERROR |                               |                                         | 11.08                  | 10.53              |

Table 4 Changes of the updating parameters for FE model

| Parameter | I Initial Value | II Updated Value | Changes (%) = |(II-I)/I| |
|-----------|-----------------|------------------|---------------|---------|
| Young's Modulus, $E_{AA6061}$ [GPa] | 72               | 67               | 6.94         |
| Young's Modulus, $E_{AA7075}$ [GPa] | 69               | 71               | 2.90         |
| Thickness of plate, $t_{AA6061}$ [mm] | 2.00            | 2.0030           | 0.15         |
| Thickness of plate, $t_{AA7075}$ [mm] | 2.00            | 2.0026           | 1.30         |

References


