Latest Development of Different Simulation Approaches for Friction Stir Processing (FSP)

Z Zulkfli[1,a)] and N Fatchurrohman[1,2,b)]

[1]*Faculty of Manufacturing and Mechatronic Engineering Technology, Universiti Malaysia Pahang, 26600 Pekan, Pahang, Malaysia.*
[2]*Department of Industrial Engineering, Faculty of Engineering, Universitas Putra Indonesia YPTK Padang, 25221 Padang, Indonesia*

[b)]Corresponding author: fatchurrohman@ump.edu.my
[a)]zulkflizuhairah@gmail.com

Abstract. Friction Stir Processing (FSP) is a welding form for joining metals which by traditional methods are too difficult to weld. The three-dimensional existence during FSP makes it more complicated for experimental studies. It is also expensive and time consuming for experimental observations. In calculating the data during experiments, there might have some inaccuracies. Different simulations were used to solve the difficulties but also to improve accuracy and minimize costs. It should be noted that strong distortions of the mesh could occur in the numerical simulation as the existence of huge deformations in FSPed materials. The relationship between the deforming continuum of computing regions and grid of finite is determined by numerical technique selection. Other than that, the numerical approach describes the model's ability to resolve large mesh distortions and offers correct boundary and interface resolution. The best choice for numerical technique is the main considerations during process simulation. Lagrangian and Eulerian are different examples of algorithms for continuum mechanics. One of the other techniques suggested which is Arbitrary Lagrangian-Eulerian (ALE), modifying the above methods by combining them. A comparison between the various numerical methods to FSP simulation and the implementations of each tool in the FSP phase has been addressed in this review paper. Observations showed that the Lagrangian method is commonly used in the entire structure field to model thermal behavior. Eulerian approach is rarely used to model deformation and thermal behavior but typically used for modeling the liquid's flow. Finally, some critical issues and topics regarding FSP simulations remain to be addressed and prospects for future study are suggested.

INTRODUCTION

Friction stir processing (FSP) is a process built applying the Friction Stir Welding (FSW) theory and definition. FSW is a process which includes solid-state welding that The Welding Institute (TWI), UK, developed in 1991 [1]. Despite the fact that it is mostly used to weld aluminium alloys as well as combining lightweight and hard-to-weld materials [2-3], other metals too can be welded using this process such as titanium alloys, magnesium alloys and metal-matrix composites. By this process, identical or different materials with various thicknesses can be welded. Some of its advantages are it can be mechanized and automated, the distortion of the workpiece is poor and the mechanical characteristics are good [4]. In addition, FSW process reduces the energy usage, environmental-friendly and more economical.

At first, a cylindrical tool consisting of one shoulder and one pin rotates and will be plunged into specified material during FSP, thus moves across the welding joint [5]. The welded material shifts from the leading edge (front side) of the tool to the trailing edge (back side) throughout FSW process after the material was string as well as softened.

Therefore, the welding parameters at various welding stages should be studied in detail on how to analyze the process thermal behavior in order to improve welding conditions using the modeling approach as it may be a cost-effective technique [6]. To highlight, there are two key sources of FSP heat production which are friction force and plastic deformation [7].

The objective of this review article is to investigate the initial conditions and methods to continue with the FSP simulation analysis as different final results were obtained in different categories due to meshing types and sizes, software selection and various data used in previous researches. In this review paper, different FSP techniques have been found that can be implemented in the simulation that will come out with different kind of results. The initial selection which requires some flow and techniques to work on FSP simulation was justified in section 2. This entire analysis paper ends with the last part. To summarize all the information obtained, tables were built.

INITIAL SELECTION FOR FSP SIMULATION

Type and Size for Meshing

Mesh sizes can affect the accuracy in the simulation model as claimed by researchers. For that, there were several researchers who applied different forms and sizes of meshing to their modeling. Giving example, for each model case, two types of element's mesh sizes were applied: element size of 0.0005 produces 1119 mesh elements whereas element size of 0.0004 produces 3697 elements. In FSP content, the mesh was refined in the middle segment to consider FSP impact [10]. Next, 32,747 tetrahedral elements were meshed for tool while 100,000 tetrahedral elements were meshed with the blank sheet [11]. From another literature, the total elements and the total nodes were 2080 and 2870, respectively [12]. Moreover, the entire volume of AA5052 is meshed with a total of 18,000 elements. A finer mesh was utilized ahead the workpiece's processing line to achieve more detailed performance. Fine meshing then, ISO advanced size functioning Proximity and curvature were added for precision [16]. For the top side of material as well as infinite domains, authors had created free quadrilateral fine mesh. The tool parts which are shoulder and pin were meshed with a very thin triangular mesh [17]. Finally, for the modeling of the welding plate, a total of 3200 8-node 3D solid elements were used and three mesh sizes applied were coarse mesh, fine mesh and very fine mesh [18].

This can be summarized that nodes' number is dependent on the element's shape and the order of interpolation. In addition, the choice of the element generally involves expert knowledge in theory plus knowledge of the application of finite element analysis (FEA). To decrease mesh deformation, remeshing techniques can be introduced. Using remeshing techniques, an increase within the computation time of CPU was observed and reduces the accuracy among results as approximation is needed to move the previous values from the distorted mesh into new values which can lose their process accuracy. Some guidelines for selecting the variable are suggested for different issues and evaluations.

Bahman et al. [1] highlights that users need to realize that various degrees of freedom are applied to each node depending on the area of the research solution such as the order of modeling space which are 1D, 2D, and 3D, rotation, displacement and temperature. Second, the formulation of numerical technique is different for different kind of families of elements. Integration which includes total and reduced integration. Loads can be implemented to shell and solid components such as surface pressure, body forces or gravity but distributions of line force can only be implemented to the shell and beam elements. All kinds of elements are geometrically suitable for non-linear analysis except for the small-strain shell portion. In general, material form that will be used for each element is not limited. In both explicit and regular solvers, many elements can be worked on.

FSP Simulation Software

Finite element software package that was selected was a main point in numerical simulation of FSP process. During FSP modeling, there are several points that need to be considered. Some software including ABAQUS, ANSYS, DEFORM-3D, MATLAB, SolidWorks and COMSOL Multiphysics are commonly used to simulate the FSP operation. Table 1 below shows different types of software packages that had been utilized to simulate FSP process along with its methods were described in the table.

TABLE 1. Different software methods according to categories.

Categories	Authors, Year	Software	Methods
Deformations	Fadi et al., (2013)	ABAQUS	Eulerian Lagrangian (CEL) coupling with adiabatic heat effect.
	Sandeepa et al., (2021)	ANSYS	Structural research has been carried out to show that the tool withstands operating conditions without loss.
Microstructure	Md Perwej et al., (2020)	DEFORM-3D	Combining thermo-mechanical model which Cellular Automaton (CA) with Laasraoui-Jonas (LJ).
	Asadi et al., (2012)	MATLAB	Relationship between parameters during FSP and grain size as well as hardness using the artificial neural network (ANN) nanocomposite.
	Ammouri et al., (2015)	DEFORM-3D	Used after being refined by FSP in the Zener-Hollomon parameter calculation.
Residual stress/Stress	Celena et al., (2016)	ANSYS	Simple analytical model is created.
	Ilesanmi et al., (2021)	SolidWorks	Combining the approach of Taguchi with Finite Element Analysis (FEA).
Temperature profile	Chen et al., (2013)	ANSYS	Centered on computational fluid dynamics. Thermo-mechanical coupled model.
	Ling et al., (2017)	DEFORM-3D	3D coupled thermal-mechanical model. Automatic conditions of contact and the adaptive technique of re-meshing.
	Bahman et al., (2020)	ABAQUS Altair Hyperworks	Programme combines the VDISP user-defined subroutine and configured input parameters.
	Siva et al., (2021)	ANSYS	ANOVA is used to survey the amplitude of the produced model.
	Vaira et al., (2018)	COMSOL	Build a mathematical model for peak temperature forecasting.
	Kishta et al., (2014)	ABAQUS	Eulerian formulation for the plate was allocated. Established rotation tool from Lagrangian formulation.
	Yaduwanshi et al., (2016)	ABAQUS	Gaussian distributed heat flux from plasma arc was used.

TABLE 1. Continue.

Categories	Authors, Year	Software	Methods
	Arora et al., (2012)	MATLAB	Developing implicit forms of heat equations of finite difference.
Material flow	Tutunchilar et al., (2012)	DEFORM-3D	Rigid visco-plastic FEM is derived from the theory of consistency of combinations, resulting in nonlinear equations.

During bending or twisting, Finite Element Modeling (FEM) enables accurate visualization of structures. The distribution of stresses and displacements are often demonstrated by these methods. A short summary on various commercially available software packages were presented in above section detailing their methods in performing FSP process including plastic deformation, microstructure, stress, thermo-mechanical behavior and material flow.

In short, ANSYS and ABAQUS software are commonly used for the simulation of deformations, temperature distributions and heat transfers. It has been found that DEFORM-3D demonstrates more detailed achievements in mechanical properties and material flow.

Eulerian, Lagrangian Methods

In accordance with structural and solid elements, the Lagrangian method is primarily used. The Lagrangian technique was considered among researchers to be able to achieve precise results on a global scale [1]. The nodes shift with the workpiece following the material's deformation as stated earlier during the process. In this system, no material passages were found between elements making this technique ideal for studying the process when small mesh distribution at the outer welding zone. By using the Eulerian formulation, the material flows. In addition, equation that the authors used for the thermo-mechanical investigation of the process was the Eulerian formulation. The influence of temperature on the viscosity was not considered in the model as well as the transient temperature too. There is gap identified between the temperature findings from the experiments and numerical model. The results showed that viscosity has a heuristic behavior for welding similar workpiece and also a temperature independent parameter.

Niyati Raut et al. [3] suggested that numerical simulation is more fitting leading the subject to branch out as multiple modeling approach such as Lagrangian formulation and Eulerian formulation, Arbitrary Lagrangian Eulerian formulation (ALE) and dynamics of computational fluid (CFD). Such methods are suggested to simulate the FSW operation. The Eulerian formulation incorporates fixed mesh through material travels while mesh deforms with material points in the Lagrangian formulation. Fluid dynamics-related models also use the classical Eulerian formulation approach. For Eulerian formulation, satisfaction of steady state conditions is important. Whereas the classical technique in solid mechanics is the Lagrangian formulation. Despite the great deformation in elements, the Lagrangian formulation is usually used to model the unsteady-state phase.

Material Input Parameters

Accuracy of simulated models can be directly affected by the material data since the temperature during the FSW process varies from room temperature to the material's melting point. The material input parameters can give a direct impact on the model's accuracy while being simulated. The newest advances using temperature dependent material input or fixed parameters including the final findings are summarized in Table 2.

TABLE 2. Findings on simulations.

Authors, Objectives, Material Data	Findings
Chen et al. [8] To simulate the generation of heat and material flow pattern. Density AA6061 = 2700 kg m^3 (constant)	Simulated temperature on retreating side (RS) was lower than the advancing side (AS). The heat flux distribution was nearly axisymmetric around the tool.
Fadi et al. [9] To describe the shape of the plasticized zone as well as void's presence in the weld. Coefficient of friction = (0.3, 0.58 and 0.8)	Frictional contact value and tool features directly affected the formed void size. Higher the coefficient of friction contributes to the development of smaller void sizes.
Celena et al. [10] To predict the gradient of resulting stress.	There was a strong difference as the material used for modeling does not plasticize in expected way during the tear test.
Ling et al. [11] To investigate the tool tilt angle effect during FSW. Density AA6061 = 2700 kg m^3 (constant) Shear friction factor = 0.4–0.6 (constant)	Worm hole defects appear with a tilted angle of zero occur in the weld. When a tilt angle of 2◦ is used, no such defects are found.
El-Sayed et al. [12] To determine the peak temperature for two different tool pin profiles. Young's modulus = 72 GPa, Poisson's ratio = 0.33	Peak temperatures for threaded tool higher than tapered tool when using rotational values at (400 and 630) rpm while welding speed values at (50, 100 and 160) mm/min.
Bahman et al. [13] To analyze thermal behavior when FSW a curved surface.	After 3 s, peak temperature of nearly 300 °C (shear zone expansion) comes out when increase the heat output.
Siva et al. [14] To determine thermal history for different parameters. Young's modulus = 70 GPa, Poisson's ratio = 0.33	Peak temperature produced from FSP AA5052 alloy is 531.754 C, (84% of material's base melting point) at TRS 1200 rpm, TS 30 mm/min and SD 12 mm.
Aude et al. [15] To study the influence of heat treatment.	In simple traction, critical interfacial stress for debonding was defined to be 435 MPa but only 250 MPa when shear is controlled by damage.
Sandeepa et al. [16] To study the thermal analysis in choosing a tool.	Least value of heat flux distribution makes triangular pin as the best profile.
Vaira et al. [17] To estimate peak temperature of Mg AZ91 for various parameter combinations.	Peak temperature was proportional to tool's rotational speed as well as diameter of the shoulder, but inversely proportional to movement speed.
Kishta et al. [18] To investigate the impact of parameters on thermal profiles. Young's modulus = 72 GPa, Thermal conductivity = 0.49 KJ/(Kg·K), Density AA5083 = 2650 kg/m^3	New Eulerian-based FE model has also been used to study the difference on thermal profiles between AS and RS.

Ilesanmi et al. [19] To create a function that correlates distortion as an independent processing parameter. Young's modulus = 69 GPa, Poisson's ratio = 0.33	Maximum distortion obtained from experiment was greater than the maximum distortion expected from simulation. It was sufficiently acceptable to ensure that it did not affect the welded joint's structural integrity.

TABLE 2. Continue.

Authors, Objectives, Material Data	Findings
Yaduwanshi et al. [20] To study various aspects of P-FSW between aluminum and copper.	Maximum temperature obtained at nearest thermocouple position on copper side is ~316–378 K higher than the aluminum side which helps to reduce the plasticized state temperature difference.
Md Perwej et al. [21] To analyse the mechanical, microstructure and metallurgical properties of FSW samples.	SZ experienced large amount of heat and plastic deformation that producing fine grains. The temperature as well as effective strain rise and decline with variation of *v*.
Arora et al. [22] To estimate the final average grain size and grain growth rate of FSPed specimen.	Presence of fine precipitate particles produced in situ on Mg alloy contributed to the growth structure of fine grains via Zener pining effect at grain limits.
Asadi et al. [23] To analyse hardness and grain size.	For training and testing patterns on hardness and grain size, correlation coefficients (R2) were used nearly unity.
Ammouri et al. [24] To predict temperature and strain rate values to calculate Zener–Hollomon parameter.	Coefficients a and b described the average difference of grain size with Z-parameter of twin roll cast Mg alloy, AZ31B which are -0.23 and 8.79, respectively.
Tutunchilar et al. [25] To describe the defect types, material flow, temperature distribution and effective plastic strain in the weld area.	Tunneling defect formation occurs at AS and at the root of SZ. At AS and closer to the top surface, powder agglomeration occurred. Both simulation and experiment showed that better distribution of powder came out from moving the tool towards RS.

To summarize, the literature's fixed material input parameters are mostly limited to lower plastic deformations and temperatures. Large plastic deformations and more fluctuations in temperature are anticipated to be occurred while undergo FSP process. Temperature-dependent material input parameter used was found to be higher to maintain the models' accuracy. In addition, the elastic modulus was obtained to be one of the most relevant input parameters since the mesh distribution problem can be solved by an accurate description of it. It is important to accurately consider thermal expansion, thermal conductivity, specific heat, yield, density and tensile strengths as they are temperature-dependent parameters.

CONCLUSIONS

Experimental analysis has been studied throughout the FSP process. Through observations from experiment, time and costs required were quite high. The numerical modeling approach of the FSP work is now very popular among researchers due to reducing the time and costs as reflected in numerous published literatures covering this subject. Different modeling methods for FSP were examined in this report. Different simulation methods have been defined for simulating the operation, which are Lagrangian and Eulerian. At the global level, testing via numerical analysis specifically when taking into account the entire structure. However, it can be locally studied by further exploring the process deeply or within the affected region from heat. Next, Lagrangian approach is ideal for structural component modeling. By combining the Lagrangian and Eulerian technique, ALE was defined as a suitable method at the local level. There are many possibilities for future works such as the correct usage of material data which are temperature dependent parameters. Furthermore, numerical technique for various thicknesses of materials can boost interest among researchers. In contrast with joining similar materials or those materials that have slight variations, the joining between different materials is typically more difficult. Numerical analysis for joining different materials will draw enormous research interests too. Lastly, the modeling of welding joint on curved material would be an interesting topic for further FSP research.

ACKNOWLEDGMENTS

The authors gratefully appreciate Universiti Malaysia Pahang and Ministry of Higher Education Malaysia for providing the supports in technical and financial aspects through Fundamental Research Grant Scheme FRGS/1/2019/TK03/UMP/02/17 (RDU 1901140).

REFERENCES

1. B. Meyghani, M. B. Awang, S. S. Emamian, M. K. Mohd Nor, and S. R. Pedapati, Metallurgy Journal, 0–23 (2017).
2. P. Asadi, M. K. B. Givi, and M. Akbari, Int. J. Mech. Mater. Eng. 10:20 (2015).
3. N. Raut, V. Yakkundi, and A. Vartak, "A numerical technique to analyze the trend of temperature distribution in the friction stir welding process for titanium Ti 6Al 4V" (Materials Today: Proceedings, 2020).
4. C. Hamilton, M. Stanisiaw, and S. Dymek, Metall. Mater. Trans. B 46, 1409-1419 (2015).
5. S. Kumaraswamy, V. Malik, S. Devaraj, V. Kumar, and L. Avinash, "A study of shear friction factor in friction stir welding for developing a finite element model and its importance in the context of formation of defect free and defective weld" (Materials Today: Proceedings, 2020).
6. B. Ahmad, A. Galloway, and A. Toumpis, J. Manuf. Process. 34, 625–636 (2018).
7. R. Jain, S. K. Pal, and S. B. Singh, J. Manuf. Process. 23, 278–286 (2016).
8. G. Chen et al., Comput. Mater. Sci. 79, 540–546 (2013).
9. F. Al-Badour, N. Merah, A. Shuaib, and A. Bazoune, J. Mater. Process. Tech. 213, 1433–1439 (2013).
10. C. A. Lipscomb, A. Fortier, F. Kong, S. Das, N. Kumar, and R. S. Mishra, Mater. Sci. Eng., A 673, 178–184 (2016).
11. L. Long, G. Chen, S. Zhang, T. Liu, and Q. Shi, J. Manuf. Process. 30, 562–569 (2017).
12. M. M. El-Sayed, A. Y. Shash, and M. Abd-Rabou, J. Mater. Process. Tech. 252, 13–24, (2018).
13. B. Meyghani, M. Awang, and C. S. Wu, J. Adv. Join. Process. 1, 100007 (2020).
14. R. S. Chintalu, R. Padmanaban, and R. V. Vignesh, "Finite element modelling of thermal history during friction stir processing of AA5052" (Materials Today: Proceedings, 2021).
15. A. Simar, A. Mertens, S. Ryelandt, F. Delannay, and L. Brassart, Mater. Sci. Eng., A 723, 324–333 (2018).
16. K. L. N. S. Sarma, A. Srikanth, and B. Venkateshwarlu, "Methodological approach for best tool geometry determination in friction stir welding process" (Materials Today: Proceedings, 2020).
17. R. V. Vignesh and R. Padmanaban, "Modelling of peak temperature during friction stir processing of magnesium alloy AZ91" (IOP Conference Series Materials and Science Engineering, 2018), pp. 012019.
18. E. E. M. Kishta, F. H. Abed, and B. M. Darras, Engng. Trans. 62, 4, 313–328 (2014).
19. I. Daniyan, K. Mpofu, B. Ramatsetse, and H. S. Phuluwa, "Numerical modeling and validation of Aluminium Friction Stir Welding (FSW) process during railcar manufacturing" (Procedia CIRP 97, 2021), pp. 7–14.
20. D. K. Yaduwanshi, S. Bag, and S. Pal, Mater. Des. 92, 166–183 (2016).
21. P. Iqbal, A. Tripathi, R. Jain, R. P. Mahto, S. K. Pal, and P. Mandal, Int. J. Mech. Sci. 185, 105882 (2020).
22. H. S. Arora, H. Singh, and B. K. Dhindaw, Mater. Sci. Eng., A 543, 231–242 (2012).
23. P. Asadi, M. K. B. Givi, A. Rastgoo, M. Akbari, V. Zakeri, and S. Rasouli, Int. J. Adv. Manuf. Technol. 63, 1095–1107 (2012).
24. A. H. Ammouri, G. Kridli, G. Ayoub, and R. F. Hamade, J. Mater. Process. Technol. 222, 301–306 (2015).
25. S. Tutunchilar, M. Haghpanahi, M. K. B. Givi, P. Asadi, and P. Bahemmat, Mater. Des. 40, 415–426 (2012).