ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

A COMPARATIVE STUDY OF FINITE ELEMENT ANALYSIS FOR FRICTION STIR WELDING APPLICATION

Bahman Meyghani, M. Awang and S. Emamian
Department of Mechanical Engineering, Universiti Teknologi Petronas, Bandar Seri Iskandar, Perak, Malaysia
E-Mail: Bahman.malaki.2009@gmail.com

ABSTRACT

Friction stir welding (FSW) is a solid state welding technique that has been used in various industries for joining different materials which are difficult or impossible to be welded by conventional welding methods. Complexity of the geometry and a three dimensional character has made the FSW process complicated in comparison with other techniques. Therefore, theoretical study of FSW is challengeable and the governing equation prediction is challengeable. Finite element analyses of FSW can predict various parameters of the welding processes such as temperature profile, deformations, stresses, residual stress and forces. It can also help to investigate the material behaviour, which can be time-consuming by using experiments. The process complexity requires the choice of the best finite element software appropriate to the results to be predicted. This paper has compared different finite element analyses which have been done for the numerical simulation of FSW. The results showed that, ABAQUS®, ANSYS® and FLUENT® software have been the most common software in those papers which have focused on FSW modelling. In the terms of mechanical properties such as thermomechanical behaviour, strain, stress and friction simulation, models created using ABAQUS and ANSYS have achieved greater accuracy. Figures show that, FLUENT software for simulating material flow and fluid dynamic behaviour modelling has been the pioneer software.

Keywords: finite element analyses, friction stir welding, numerical simulation, ABAQUS, ANSYS, FLUENT.

INTRODUCTION

Joining technology plays a key role in structures' manufacturing. One of the most prominent techniques for joining materials in the recent years is FSW [1]. Finite element methods (FEM) are numerical techniques for solving approximate solutions to boundary value problems for differential equations. This method is using variational methods to reduce error functions and producing stable solutions. FE methods connect a lot of tiny straight lines which they can approximate a larger circle; FEM has procedural steps to connect small elements with finite length over many small sub domains for the approximation of complex equation within the domain under observation.

For resolving FE problems two methods have been proposed, explicit and implicit. The former is using for solving transient dynamic response by using direct integration procedure and the latter one is using to solve problems with static or dynamic equilibrium.

With the advance of non-linear FE analyses, different element types and solution algorithms have been proposed and simulated in specialized software such as ABAQUS, ANSYS and FLUENT. Thus, the question is raised for studying non-linear FEA if it is already available in many commercial packages. The answer lies in then fact that a commercial package is a black box program which provides an interface for simulations. The user has to make several choices in the model formulation i.e. the discretization of the mesh, the solution procedure, the element types, the solver tolerances and etc. The selection of these options has an extra ordinary impact on the ultimate result and sometimes can lead to non-physical and hence thus meaningless results.

For non-linear solid mechanics study, four different sources have been proposed, non-linear geometry, non-linear material, non-linear force boundary conditions and non-linear displacement boundary conditions. Figure-1 shows non-linearities diagram.

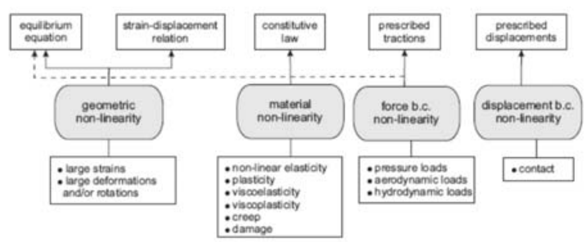

Figure-1. Non-linearities in solid mechanics.

There is an increasing need for reducing both time and cost in all industries. FEA has been proposed to solve the problems which are difficult to predict or expensive in experiments. One of the most common problems in the first step of analysis is to choose the right software according to the process. Therefore, the main objective of this research is to compare different software which has been used for modeling the FSW process to carry out the results that should be achieved.

THERMO MECHANICAL AND FRICTION MODELLING

A lot of studies have been designed for elucidating different aspects of thermomechanical phenomena during FSW.

FE modelling of FSW, using ABAQUS was successfully achieved by adopting a coupled Eulerian Lagrangian formulation and by considering the thermomechanical behaviour encountered during the process. The FE results validation was achieved. The

Eulerian based FE model was not only able to successfully predict the effect of rotational speeds on the peak temperature and thermal profiles in general, but was also capable of capturing the variation in thermal profiles between advancing and retreating sides [2].

FSW and friction stir spot welding (FSSW) in the plunge stage was investigated for finding thermomechanical behavior. A physically based, coupled thermomechanical model has been investigated during the FSSW of AA6082-T6 alloy [3]. High calculation time and distortion problems have been solved by using, Arbitrary Lagrangian Eulerian (ALE) technique, mesh remapping techniques and mass scaling method. The influence of the different parameters in the FSW on the temperature behavior was studied and figures showed that, interacting conditions are very challengeable at the tool work-piece interface. An improved methodology based on process variables impacting upon the system design has been introduced.

A finite element model has been predicted and presented different aspects of FSW such as, contact and material behavior [4]. Welding stages has successfully simulated as a fully thermomechanical model using continuum solid mechanics. Temperature, stress and strain field are quantified by the FE model.

For large scale FSW structures the equivalent load method which uses inherent strain methodology, has been proposed [5]. ANSYS has been used in this simulation. The results showed good agreements with experiments. In the study a well-organized method has been used for simulating deformations and residual stress of aluminum alloy Al6061-T6 sheets.

A FE analysis of the process has been investigated using Abaqus/Explicit. An ALE formulation was used for mesh destruction. A thermomechanical analysis for showing the heat generation and temperature fluxes through plastic dissipation in stir (Sheppard rigid viscoplastic constitutive model). Norton frictional contact model was used for the frictional heat dissipation at the interfaces. For understanding the welding behavior, material flow in the pin area showed using tracers [6].

Finite element code using ABAQUS was employed. The effect of using various thermal and boundary conditions for the heat conduction and plastic deformation were measured by utilizing a rigid viscoplastic material behavior [7]. Eulerian formulation and Johnson-Cook material law was used as well. The model was calculated the compressibility of the material with the help of the elastic response for aluminum. Coulomb’s law was used for modeling the contact forces, making the contact condition highly solution dependent.

Johnson–Cook material law has been used for studying different manufacturing processes in a FE model, because manufacturing processes play a key role in metallic materials plastic deformation. But, when the model used in the engineering analyses hot working processes faced serious shortcomings. The main aim of the study was to mix fundamental principles of physical metallurgical with related kinetics to alter Johnson–Cook strength model. It could be used in the analyses of metal hot working and welding processes [8].

A 3D FE model for simulating the thermomechanical analysis during FSW of different aluminum alloys was studied by using ABAQYS [9]. Different working conditions have been used for experiments. The temperature changes and the residual stresses in different positions of the welding throughout the welding process were measured during the process. Thus, these changes are measured by using hole drilling method. Results showed that tool rotation speed plays a key role in the residual tensile stress value. However, changing feed rate speed has impact on residual transverse stresses distribution. Besides, the experiments and results showed that welding fixtures has an important effect on the residual stress profiles and the levels related to them.

By applying techniques such as ALE, Johnson-Cook material law and Coulomb’s law in Abaqus/Explicit a 3D FE model was successfully measured. Numerical results showed a prominent influence from the friction generated heat. Tool slip rate depends on the work-piece material which was shown as the portion of heat. But, material velocity was related to the heat generated by plastic deformation [6].

Both the classical Coulomb’s law and the Norton’s friction law are compared together with the corresponding heat flux produced due to friction dissipation. [10].

ABAQUS was used with the combination of the user subroutine which was compiled by FORTRAN code for the description of a modified coulomb law. Eight node thermo-mechanical brick elements were used for mesh generation of the work-piece. For convenience of mesh generation with brick elements, a circular work-piece with the radii of 30mm is considered. The model has been validated for the temperature and material flow during FSW of AA6061 [11].

MATERIAL FLOW MODELING

In a systematic research, a coupled thermomechanical FE model has been investigated [11, 12]. Results indicated that, shoulder rotation can speed up the material flow near the welding surface. The effect of material deformation and temperature field on the microstructure evolution was studied. Observation has shown that texture and appearance in the joints have good correlation with equivalent plastic strain distributions on the top surface of the work-piece and the temperature field was symmetric around the welding path. The material flows changed with varying work-pieces thicknesses.

A FE model using ABAQUS software was calculated an elastic plastic, coupled thermomechanical model for modeling the FSW of aluminum 7075 [13]. Results of the horizontal direction showed that, two material patters migration are produced: in the first one the material is rotated with the tool and deposits the tentative cavity behind the pin; in the second one, the material transfers in the mode of laminar flow.

Based on the FSSW structure a coupled thermomechanical visco-plastic FE analysis was measured

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

[14] and was calibrated. A good agreement with experiments was found and was employed for validating more effective strain distribution during the welding area.

Both material flow modeling and coupled thermomechanical modeling in the FSW process was presented using COMSOL software and was used for assembling of Sc-modified aluminum alloy (7042-T6) extrusions [15]. The FE model showed that the surface material rotated from the retreating side into the welding area where it is added within situ material. The surface material was warmer as compared to in situ material due to frictional contact with shoulder. Therefore, the final microstructure was made-up of a mixture of materials having various temperature readings. By applying FE analysis for thermal analysis details acquired through differential scanning calorimetry, a technique was developed to form the onion rings within the weld zone.

Material flow during the stirring and mixing process were examined in a FE model [8]. Coupled Eulerian/Lagrangian analysis has been used for the process for finding thermomechanical behavior, however; temperature was made independent to make an impact upon the mechanical features of sheets by material properties which are temperature dependent. A modified classical Johnson–Cook model was used. However, the tool material was formed with an isotopically linear elastic material. The results obtained due to material flow throughout the process were matched to the experimental ones.

A FE thermomechanical simulations of the FSSW were presented for AA5083 H18 sheets using commercially available FVM code [16] and Eulerian formulation was used for simulations. A fluid dynamic code STAR-CCM+ has been used for simulation. The simulation was done under a steady state condition. Results showed that simulation accuracy increased sharply by inclusion of the proper thermal properties of the back side plate. To find the impact of the probe geometry on the material flow and weld strength in the FSSW process, STAR-CD CFD code was used under non-steady conditions.

A FE model of the process has been presented by using ANSYS FLUENT. The model was discussed about the real geometry for the rotation tool. Material parameters' influence on the temperature was studied [17]. Furthermore, the impact of pin geometry and the shoulder geometry on material flow performance was introduced. It was found that, by reducing the probe angle or width of the screw groove, material flow velocity within weld is expanded. When the left screw pin within the tool rotates clockwise, the material direction flow near the pin is downward, however; direction of flow near the thermal mechanical affected zone is upward, evidence show that material flow is opposite for a right screw pin. The research work related to the material flow shows valued details for material behavior and helps to gain enhancements in tool design and proper choice for FSW parameters. Material flow was presented and described by fluid dynamic CFD code using FLUENT for simulating 2D or 3D metal flow in FSW processes [18-22].

TEMPERATURE DISTRIBUTION MODELING

A FE fully coupled thermal mechanical using ALE method has been presented for simulating FSW. The inelastic heat generation is encompassed in the model [6, 23] and the temperature fields were simulated. In this study tool rotational speeds of 600 and 800 r/min have been studied. The results indicated a good agreement between the FE model and experiments in terms of temperature distribution.

A coupled thermomechanical FE simulation including the plunge phase of a modified refill FSSW was developed. The model has been presented in Abaqus/Explicit. Results included deformation, temperature, stress, and the strain distribution. A nice correlation between the temperatures measured in comparison with the experiment was found. Besides, in terms of stress and plastic strain numerical model was capable to estimate in a better way [24].

Temperature profile and energy dissipation history of a FE model of the FSSW process using a Abaqus/Explicit code was used for simulating a 3D coupled thermal stress [25]. A Johnson–Cook rate dependent material model has been used for elastic–plastic work-piece deformations.

FSW process was simulated in the ABAQUS. DFLUX subroutine has been used for the simulation. Both thermal and strain rate analytical models were used to validate the process [26]. In the model grain size and yield strength were estimated and producing the transition rules, relating them to temperature, strain-rate and strain observed. From the CAFE model, grain size prediction and yield strength distribution was validated with the experiments.

The evaluation of the thermomechanical behavior and final micro structure of different aluminum AA6061-T6 and AA5086O were predicted [27]. ABAQUS has been used for finding thermomechanical behavior during both similar and dissimilar FSW. The simulation was predicted by employing ABAQUS. The mechanical properties of the work-piece and microstructures were studied. Results showed that different strengthening mechanisms in aluminum alloy AA5086 and AA6061 in complex behaviors with respect to the welded cross section hardness. It was found that hardness variation in AA5086O aluminum plate joints depends on both recrystallization and generation of the fine grains in weld nugget. But, the hardness variations of both AA6061/AA6061 and AA6061/AA5086 plate are affected by subsequent aging the welding.

Cellular automata coupled finite element (CAFÉ) model was applied for grain size distribution prediction in a FSW and the effect of the weld faults on the sheet formation. Simulation of the FSW procedure was simulated by using the ABAQUS and with the help of the subroutine DFLUX. Thermal and strain rate analytical models to the elements constituting the FSW blanks [26]. The final grain size and yield strength prediction was studied by relating them to temperature, generating transition rules, strain-rate and strain evolved. Both grain size and yield strength distribution were predicted from the CAFÉ model and results showed good agreements with the experiments in all conditions.

In a 3D FE analysis both corresponding thermal and time varying material field was analyzed using ABAQUS. The heat generated between the tool and sheets was simulated. Numerical simulation was done by the help of the user defined subroutine DFLUX for the moving heat source load. Copper sheets have been embedded in the welding path of aluminum 2024 as a marker and their final positions after the FSW was examined using metallographic techniques [28]. A 3D FE model was simulated regarding to the visualized material flow patterns to conduct both the temperature profile and material flow. Velocity contour was evaluated for the plastic material flow near to the tool. The prediction results and nugget zone size matched with experiments.

A thermomechanical analysis for the steady state conditions of stainless steel has been modeled using an Eulerian [29] and viscoplastic self-consistent methods. In the model the texture was developed in sheets. Material behavior was found from the velocity gradients from the streamlines in the material flow field. Heat profile was studied near the interface between the tool and sheets. Results showed that in the vicinity of the boundary between the stir zone and the TMAZ the viscosity changes sharply.

Based on thermomechanical processing of metals a FE viscoplastic metal flow and the temperature profile in the FSW were determined [29]. Momentum equations, mass conservation, and energy were resolved. Spatially variable thermos-physical properties and non-Newtonian viscosities were used. In terms of metal flow Non-Newtonian viscosity was used according to the temperature, strain rate, and temperature dependent material properties. Both viscosity and strain were tested in the light which is existing literature on thermomechanical behavior. Results showed that, heat asymmetry and mass flow are important in increasing the welding speed and rotational speed. A good agreement with the experimental was observed in a numerical simulation of temperature profile, cooling rates, and the geometry of the TMAZ.

SUMMARY

The result of this comparative study demonstrates that the software of ABAQUS/Explicit uses an explicit integration scheme to solve highly nonlinear transient dynamic and quasi-static analyses. Hence, in the tool wear simulation and thermo mechanical simulation ABAQUS software has had the best accuracy with the experiments, because of its capabilities in defining material law and failure models. The simulation of FSW process is a complicated solution especially where the interaction between structures and fluids is considered. This software has a strong feature for choosing variety of elements such as Eulerian, Lagrangian and coupled Eulerian Lagrangian, which can help simulating the cases with the large plastic deformations. Arbitrary Lagrangian–Euleraian (ALE) technique has been used in majority of numerical studies for modifying the mesh distraction. This applicable technique is only available in the ABAQUS software. In addition, writing subroutines is available in the ABAQUS software which helps to modify non-linear behaviors for different parts of the modelling such as mesh destruction and interaction. According to the complexity of the FSW process, there is a need for using variety of elements when the mechanical properties such as fatigue behavior, deformation and tensile stress are desired.

One of the main drawbacks of ABAQUS is its capability to undo the work in case of error and / or change in the modeling of any geometry, because ABAQUS works with instances in the assembly module which is not available in the ANSYS feature. Thus, in ABAQUS the model creates one part and makes multiple instances of it. If there is a change in the part, it will affect all of the instances which can be helpful for optimizing the tool geometry.

Moreover, Users can write their own scripts in Python to automate or make their calculation parametric. They can also write their own subroutines. Furthermore, in the ABAQUS, there is a supportive library of user-material models (UMATs/VUMATs) for general-purpose finite element modeling of all polymer systems including, thermoplastics, thermo sets, elastomers, foams, filled plastics, and biomaterials. This variety of material behaviors are applicable in the FSW process, because this method of welding is usually uses for joining those materials which are difficult to weld by conventional welding methods. This library provides for the ABAQUS users the advanced material models, as if they built into the ABAQUS Standard and Explicit solvers, so the engineers can perform accurate FE simulations with less expertise in material model software development. In addition, an optimization application that enables semi-automatic extraction of pertinent material parameters from experimental data for all material models is available in ABAQUS.

Evidence shows that, FLUENT software has been used to predict the impact of fluid flows on the product throughout design and manufacturing. The complexity of FSW model is in the kinematics, tool geometry and the mechanisms that the weld is created by them. The limitation of the material flow is in the region enveloping the tool and process variables in this area are coupled in a manner which excludes an independent treatment. As an instance, viscosity is highly reliant on both the strain rate and temperature; which both of them reliant on the frictional condition at the welding line. The frictional condition at the welding boundary is strongly reliant on viscosity of the material, producing a closed circle relationship. Plastic dissipation, viscous heating and frictional heating at near the tool surface produce steep temperature gradients in both the work material and the tool itself near their interface. The steep process variable gradients in this area are compared with the lack of such gradients throughout the remainder of an FSW model. The FLUENT software's unparalleled fluid flow analysis capabilities can be used to optimize and design new equipment and to troubleshoot existing installations. The FSW is a multi-phase process; therefore, FLUENT

ARPN Journal of Engineering and Applied Sciences

software can give valuable insight into the material flow behavior. Advanced solver technology module provides fast, accurate CFD results, flexible moving and deforming meshes, and superior parallel scalability. User-defined functions allow the implementation of new user models and the extensive customization of existing ones.

CONCLUSIONS

Finite element methods allow detailed visualization of where structures bend or twist, and indicate the distribution of stresses and displacements. For solving the complexity of the FSW process, finite elements methods can provide a various range of simulation options. Numerical methods of the FSW help to have a validate prediction of main process parameters and also can reduce the number of experiments. Thus, accelerating the design processes while not only optimizing the involved technological variables but also reducing the costs. In this study different aspects of the FSW process such as thermomechanical behaviour, fatigue behaviour, plastic deformation, stress and friction modelling were compared. Results show that, the software of ABAQUS and ANSYS are the prominent software in terms of simulating mechanical properties specifically in this procedure. Also it is reported that in material flow and fluid dynamic modelling, FLUENT shows more accurate achievements for solving the wide-ranging fluid flow.

REFERENCES

[1] Thomas, W., *et al*., Friction stir welding. International patent application no. PCT/GB92102203 and Great Britain patent application, 1991(9125978.8).

[2] Kishta, E.E., A. Farid, and B. DARRAS, Nonlinear Finite Element Simulation of Friction Stir Processing of Marine Grade 5083 Aluminum Alloy. Engineering Transactions, 2014. 62(4): p. 313–328.

[3] Khosa, M.S.S.U., Thermo-mechanical investigations during friction stir spot welding (FSSW) of AA6082-T6. Welding in the World, 2010. 54(5-6): p. R134-R146.

[4] Hamilton, R., D. MacKenzie, and H. Li, Multi-physics simulation of friction stir welding process. Engineering Computations, 2010. 27(8): p. 967-985.

[5] Jang, C.D. and Y.J. Jang. Welding deformation analysis of friction stir welded aluminum alloy structures using equivalent load method based on inherent strain. in The Twentieth International Offshore and Polar Engineering Conference. 2010. International Society of Offshore and Polar Engineers.

[6] He, X., F. Gu, and A. Ball, A review of numerical analysis of friction stir welding. Progress in Materials Science, 2014. 65: p. 1-66.

[7] Schmidt, H. and J. Hattel, A local model for the thermomechanical conditions in friction stir welding. Modelling and Simulation in Materials Science and Engineering, 2005. 13(1): p. 77.

[8] Grujicic, M., *et al.,* Computational analysis of material flow during friction stir welding of AA5059 aluminum alloys. Journal of materials engineering and performance, 2012. 21(9): p. 1824-1840.

[9] Aval, H.J., S. Serajzadeh, and A. Kokabi, Experimental and theoretical evaluations of thermal histories and residual stresses in dissimilar friction stir welding of AA5086-AA6061. The International Journal of Advanced Manufacturing Technology, 2012. 61(1-4): p. 149-160.

[10] Chiumenti, M., *et al.,* Numerical modeling of friction stir welding processes. Computer Methods in Applied Mechanics and Engineering, 2013. 254: p. 353-369.

[11] Zhang, Z. and H. Zhang, A fully coupled thermo-mechanical model of friction stir welding. The International Journal of Advanced Manufacturing Technology, 2008. 37(3-4): p. 279-293.

[12] Zhang, Z. and Y. Liu, Effect of preheating time on friction stir welding. Chin J Mech Eng, 2009. 45: p. 13-8.

[13] Guo, Z.H., *et al.* Finite element analysis of materials flow behavior in friction stir welding of 7075 aluminum alloy plate. in Applied Mechanics and Materials. 2012. Trans Tech Publ.

[14] Gao, Z., *et al*. Analysis of plastic flow during friction stir spot welding using finite element modelling. in Key Engineering Materials. 2012. Trans Tech Publ.

[15] Hamilton, C., *et al.,* A coupled thermal/material flow model of friction stir welding applied to Sc-modified aluminum alloys. Metallurgical and Materials Transactions A, 2013. 44(4): p. 1730-1740.

[16] Kim, D., K. Chung, and H. Badarinarayan. An experimental and numerical approach to evaluate the effect of welding conditions on temperature during friction stir spot welding. in ASM proceedings of the international conference: trends in welding research. 2009.

[17] Ji, S.D., *et al.,* Numerical simulation of material flow behavior of friction stir welding influenced by rotational tool geometry. Computational Materials Science, 2012. 63: p. 218-226.

[18] Colegrove, P., H. Shercliff, and P. Threadgill. Modelling and development of the Trivex friction stir welding tool in Friction Stir Welding: Proceedings of the 4th International Symposium on Friction Stir Welding. 2003. TWI Limited.

[19] Hyoe, T., H. Shercliff, and P. Colegrove. Thermal and microstructure modelling in thick plate aluminium alloy 7075 friction stir welds. in Friction Stir Welding and Processing II: Proceedings of the 2nd International Conference on Friction Stir Welding and Processing. 2003. TMS: The Minerals, Metals and Materials Society.

[20] Colegrove, P. and H. Shercliff, Two-dimensional CFD modelling of flow round profiled FSW tooling. Science and Technology of Welding & Joining, 2004. 9(6): p. 483-492.

[21] Colegrove, P.A. and H.R. Shercliff, 3-Dimensional CFD modelling of flow round a threaded friction stir welding tool profile. Journal of Materials Processing Technology, 2005. 169(2): p. 320-327.

[22] Colegrove, P. and H. Shercliff, Experimental and numerical analysis of aluminium alloy 7075-T7351 friction stir welds. Science and Technology of Welding & Joining, 2003. 8(5): p. 360-368.

[23] Peng-Fei, Y., *et al.,* Numerical simulation of coupled thermo-mechanical process of friction stir welding in quasi-steady-state. 2013.

[24] Muci-Küchler, K.H., S. Kalagara, and W.J. Arbegast, Simulation of a refill friction stir spot welding process using a fully coupled thermo-mechanical FEM model. Journal of manufacturing science and engineering, 2010. 132(1): p. 014503.

[25] Awang, M. and V.H. Mucino, Energy generation during friction stir spot welding (FSSW) of Al 6061-T6 plates. Materials and Manufacturing Processes, 2010. 25(1-3): p. 167-174.

[26] Saluja, R.S., R.G. Narayanan, and S. Das, Cellular automata finite element (CAFE) model to predict the forming of friction stir welded blanks. Computational Materials Science, 2012. 58: p. 87-100.

[27] Aval, H.J., S. Serajzadeh, and A. Kokabi, Thermo-mechanical and microstructural issues in dissimilar friction stir welding of AA5086–AA6061. Journal of materials science, 2011. 46(10): p. 3258-3268.

[28] Chuan-song, W., *et al.,* Visualization and simulation of plastic material flow in friction stir welding of 2024 aluminium alloy plates-TNMSC. The Chinese Journal of Nonferrous Metals, 2015. 22(6).

[29] Cho, H.-H., *et al.,* Three-dimensional numerical and experimental investigation on friction stir welding processes of ferritic stainless steel. Acta Materialia, 2013. 61(7): p. 2649-2661.