Finite Element Modeling of Friction Stir Welding-Thermal Analysis

Amay Deorao Meshram

Student of Master of Technology in Thermal Engineering Government College of Engineering Amravati – 444604,Maharashtra,India

ABSTRACT

The Friction Stir Welding (FSW) is a new type of manufacturing process in which the relative motion between the tool & work-piece produces heat in the tool workpiece interface makes two metal sheets joined by plastic diffusion by virtue of frictional heat. Various researches have been done over the friction stir welding of metal Sheet to investigate the effect of welding parameters as metal sheets are influenced by frictional stir welding parameters.

The aim of the study is to develop a three dimensional model of metal sheet by friction stir welding using finite element method for specific experimental cases & validate the data & results obtained by experimental techniques. The friction stir welding is to be simulated using FEM model with software tool ANSYS.

Keywords-Friction welding, FEM method, ANSYS

1.INTRODUCTION

Friction stir welding (FSW) is a recently emerged solid-state joining technology patented by The Welding Institute (TWI) in 1991[1].The use of the Finite Element Method (FEM) in product development is now well established. Its use in manufacturing processes is increasing and is part of the field of new applications in computational mechanics. The most important reason for this development is the industrial need to improve productivity and quality.

Welding is defined by the American Welding Society (AWS) as a localized coalescence of metals or non-metals produced by either heating of the materials to a suitable temperature with or without the application of pressure, or by the application of pressure alone, with or without the use of filler metal. Welding techniques are one of the most important and most often used methods for joining pieces in industry. Any information about the shape, size and residual stress of a welded piece is of particular interest to improve quality [6].

FSW is based on strong couplings of thermo-mechanical phenomena. It induces very complex material motions and large shear forces. The material temperature is raised to about 80% of the melting temperature .Never the less the simulation of the process will be a further aim, as it is difficult to be numerically modelled due to the complex thermal and material flux occurring during the process, similar to Friction Stir Welding .

The conventional processes, working with molten phases are characterized by large heat input, which can change the microstructure of the diverse materials. This can provide mixed phases, which are very brittle and hardly formable, as well as hot cracks due to shrinkage during cooling or shape deviation. Contrary to melting joining techniques Friction Stir

Welding is characterized as a solid phase welding technology, which was patented in 1991 [6].The probe primary function is to mix the material under the tool shoulder, which can be enhanced by threads. FSW is actually performed in three steps. First, the probe is plunged into the joint formed by the two sheets to be welded, until the shoulder gets in contact. As the tool rotates at a high velocity, the sheets are heated up by plastic deformation and friction. Second, the tool keeps rotating without any translational motion, so the material heating due to friction increases. Finally, the tool moves along the joint line, heats the material further, moves it from the front of the tool, and deposits it behind its trailing edge, producing the weld. This process is illustrated in Fig. 1. [4].

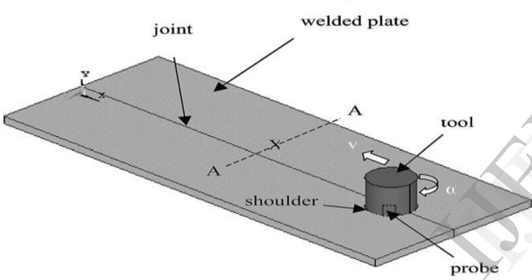

Fig. 1.A schematic illustration of FSW process.

2. MODEL DESCRIPTION

2.1. Heat transfer model

The temperature calculation is based on Fourlier's equation:

$$\rho c\frac{dT}{dt} = div(k.gradT) + q \; In \; \Omega$$
(1)

where q is the power generated by friction between the tool and the top of the workpiece and by the plastic deformation work of the central weld zone, T is the temperature ,k is the conductivity, ρ is the material density, and c is the heat capacity.

The main heat source in FSW is generally considered to be the friction between the rotating tool and the welded plates, and the "cold work" in the plastic deformation of material in the vicinity of the tool. Considering an element at the contact surface between the tool shoulder and the top surface of workpiece, the rate of heat generation derived from the friction in the element at radius r is:

$$dq = 2pw \cdot r2m(T)p(T)dr \quad (2)$$

The rate of heat generation (caused by the friction) over the entire interface of the contact will be:

$$q = =pwm(T)p(T)(R - r) \quad (3)$$

The rate of heat generation at the interface between the shoulder and the top of the workpiece surface is a function of the coefficient of friction m, angular velocity w, and radius r. As the m(T) and p(T) are dependent on the local temperature and the radius r, Equation is difficult to evaluate. As the temperature increases, the friction coefficient is expected to decrease, and the work of the plastic formation increases. In this model, the p is from the experimental measurement. A constant value of the friction coefficient is used to approximate the comprehensive effect of both factors of thermal and plastic effects during FSW, and the predicted temperature history is verified by the measured ones. The heat generation in the vicinity of the probe is considered the same as that at the shoulder periphery with a radius equal to that of the probe.

2.2. Finite element model

The ANSYS as a commercial FEM software is used to carry out the numerical simulation. In the simulation, the thermomechanically coupled three-dimensional Lagranrian finite element model incorporating

ISSN: 2278-0181
Vol. 3 Issue 1, January - 2014

temperature and multilinear strain hardening effects is used for the three-dimensional modeling of the solid structures. The element topology used is eight nodes and has plasticity, stress stiffening, large deflection, and large strain capabilities[11].

Symmetry along the weld line is assumed in the calculation, so one half of the welded plate is meshed with a total of 3537 nodes. In modeling the temperature history, the moving heat sources of the shoulder and the probe are represented as moving the heat generation of the nodes in each computational time step. The mechanical effect by the shoulder is involved in the mechanical model, as the relatively larger contact region of the shoulder and the workpiece is expected to contribute a large part of the mechanical stress, especially in the uphalf part of the weld. The temperature gradient is large around the welding zone and seriously changes the materials properties. In order to increase the accuracy of the mechanical solution, the thermal and mechanical solutions are coupled the temperature data at each increment time is used to evaluate the mechanical properties and the thermal parameters.

3.LITERATURE REVIEW

i)Chao Yuh J. & Qi Xinhi [1989] produced numerical FSW studies. In their research, temperature fields during welding, the residual stress distribution and distortion of workpiece after FSW were studied. Decoupled heat transfer and Thermo-mechanical modeling and analysis and 3 dimensional finite element formulation was used in their study. Also a moving heat source with heat distribution simulating the heat generated from friction between tool shoulder and workpiece was implemented as the heat input.

The main sources of heat were from friction between tool shoulder and workpiece and from the plastic deformation of the weld material in the vicinity of rotating pin. The empirical equation for calculating the heat input in their analysis is given by Equation (11.1)

$$q(r) = 3Q_3 r \quad 2\pi r_o^3 \quad \text{where } r \le r_o \qquad (11.1)$$

Where q(r) is the rate of heat flux, Q_3 is the heat flux to shoulder.

A trial and error approach was used where in the total heat input to workpiece and heat convection coefficient of bottom surface are calculated by fitting the major temperature data with the analytical model. This was also known as the inverse approach.

These temperatures were then used to determine the residual stress after FSW process. The results matched to a greater extent and the process was comparatively simpler than the direct approach used by various other authors. According to the authors, approximately 95% of the heat generated from friction was transfer to workpiece and the remaining 5% was transferred to the tool. The fraction of rate of plastic work dissipated as heat was approximately equal to 80%.

ii) Andres Anca, Alberto Cardona , Jose Risso, Victor D. Fachinotti [2011] This work deals with the simulation of fusion welding by the Finite Element Method. The implemented models include a moving heat source, temperature dependence of thermophysical properties, elasto-plasticity, non-steady state heat transfer, and mechanical analysis. finite element models have been used to analyze the thermal and mechanical phenomena observed in welding processes. Thermal histories and residual stresses have been predicted.

The described methods provide a powerful means to optimize process parameters off-line. The welding model can be used to predict and minimize deformations due to changes in the weld sequence during the product development stage.

A mathematical model was implemented to represent the power density distribution of the external heat source.

iii)Selvamani S.T, Umanath K and Palanikumar K [2011]In this work, a moving co-ordinate has been introduced to model the three-dimensional heat transfer process because it reduces the difficulty of modeling the moving tool. A three dimensional finite element model is developed to study the thermal history in the butt welding of 6061 aluminum alloy using ANSYS package. Solid 70 elements are used to develop the model.

A three-dimensional thermo mechanical model and the thermo mechanical effect of the welded material is developed for the FSW of an Al-alloy, in order to build qualitative framework to understand the thermo mechanical process in FSW.

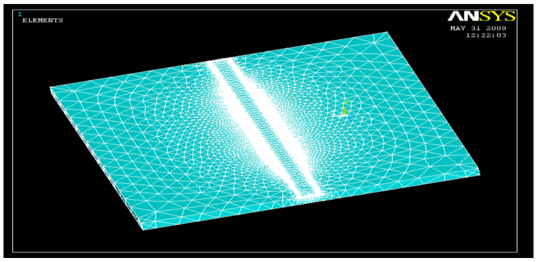

Fig. 2.A view of Finite Element mesh of Friction stir welding plate

iv) C.M. Chen, R. Kovacevic[2003]A three-dimensional model based on finite element analysis is used to study the thermal history and thermomechanical process in the butt-welding of aluminum alloy 6061- T6. The model incorporates the mechanical reaction of the tool and thermomechanical process of the welded material. The heat source incorporated in the model involves the friction between the material and the probe and the shoulder. In order to provide a quantitative framework for understanding the dynamics of the FSW thermomechanical process, the thermal history and the evolution of longitudinal, lateral, and through-thickness stress in the friction stirred weld are simulated numerically.

A three-dimensional thermomechanical model including the mechanical action of the shoulder and the thermomechanicaleffect of the welded material is developed for the FSW of an Al-alloy, in order to build qualitative frame work to understand the thermomechanical process in FSW. Modeling and measurement of the temperature and stress evolution in the FSW of 6061-T6 Al alloy is conducted.

v) P. Heurtier , M.J. Jones , C. Desrayaud , J.H. Driver, F. Montheillet , D. Allehaux[2010]A three-dimensional thermomechanical model for Friction Stir Welding (FSW) is presented. Based on the velocity fields classically used in fluid mechanics and incorporating heat input from the tool shoulder and the plastic strain of the bulk material, the semi-analytical model can be used to obtain the strains, strain rates, and estimations of the temperatures and micro-hardness in the various weld zones. The semi-analytical simulation of the FSW process proposed

In this paper highlights the thermomechanical history of the various material elements of the weld. This original modeling provides the trajectory of each material element of the weld, the strain, strain rates and estimations of the temperatures and micro-hardness in the various weld zones.

vi) Hazman Seli, Ahmad Izani Md. Ismail, Endri Rachmanc, Zainal Arifin Ahmadd[2010] In friction welding of two dissimilar materials, two rods are welded together by holding one of them still while rotating the other under the influence of an axial load which creates frictional heat in the interface. In this study, mechanical properties of mild steel and aluminium welded rods were evaluated

to understand the thermal effects, and an explicit one-dimensional finite difference method was used to approximate the heating and cooling temperature distribution of the joint. The thermal effects of the friction welding were observed to have lowered the welded materials hardness compared to the parent materials. The tensile strength of the welded rods is lower than the parent rods due to incomplete welding.

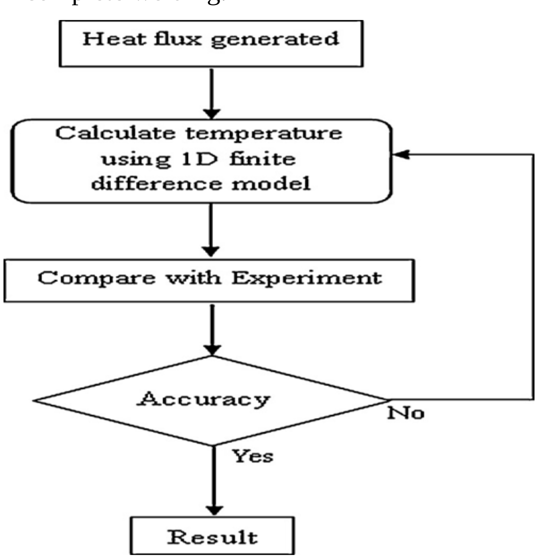

Fig. 3.Flowchart of the temperature calculation.

4.RESULT AND DISCUSSIONS

The study would consists of three parts which are :

1}The theoretical basis : The theoretical basis explores the information related to friction stir welding and the factors related to utilize these theories. Friction stir welding is relatively new solid state joining process. Frictional heat is generated at the tool-work material interface by the tool under the action of a vertical load, thereby reducing the material flow stress.

Factors related friction stir welding,

Time, speed, strength, depth of cut etc.

2}Finite Element Analysis: The second section is the development of a three-dimensional FEA model to analyze the welding process.

The ANSYS as a commercial FEM software is used to carry out the numerical simulation. In the thermomechanically coupled three-dimensional modeling of the solid structures.

3}The Thermal Effects. The third part concentrates on study of the various thermal effects,

In the FSW process, the heat increase comes from various sources:

1] The plastic strain;

2] The friction of the pin on the material;

3] The friction of the shoulder on the surface of the work pieces

5. CONCLUSION

It is concluded that the computation finite element results are very close to the experimental results. The ANSYS software is used to take part in this analysis it also show the validated result.

REFERENCES

[1]C.M. Chen, R. Kovacevic" Finite element modeling of friction stir welding—thermal and thermomechanical analysis" International Journal of Machine Tools & Manufacture 43 (2003) 1319–1326

[2]P. Heurtier , M.J. Jones , C. Desrayaud , J.H. Driver, F. Montheillet , D. Allehaux, "Mechanical and thermal modelling of Friction Stir Welding"Journal of Materials Processing Technology 171 (2006) 348–357.

[3]Hazman Seli, Ahmad Izani Md. Ismail, Endri Rachmanc, Zainal Arifin Ahmadd," Mechanical evaluation and thermal modelling of friction welding of mild steel and aluminium" Journal of Materials Processing Technology 210 (2010) 1209–1216.

[4]Selvamani S.T , Umanath K and Palanikumar K / International Journal of Engineering Research and Applications (IJERA)''Heat Transfer Analysis during Friction Stir Welding of Al6061-T6 Alloy''(2011) ISSN: 2248-9622 www.ijera.com Vol. 1, Issue 4, pp. 1453-1460

[5]Mustafa Kemal Bilici , Ahmet _Irfan Yukler, Memduh Kurtulmus"The optimization of welding parameters for friction stir spot weldingof high density polyethylene sheets"Materials and Design 32 (2011) 4074–4079.

[6]Andres Anca, Alberto Cardona , Jose Risso, Victor D. Fachinotti" Finite element modeling of welding processes" Applied Mathematical Modelling 35 (2011) 688–707

[7] Chao Yuh J. & Qi Xinhi. Thermal and Thermo-mechanical Modeling of Friction Stir Welding of Aluminium Alloy 6061-T6. Journal of Materials Processing & Manufacturing Science. 1998, Page No. 215-233.

[8] M. Sunar , B.S. Yilbas , K. Boran"Thermal and stress analysis of a sheet metal in welding" Journal of Materials Processing Technology 172 (2006) 123–129

[9]Xin Yang, Yong Xia, Qing Zhou "A simplified FE model for pull-out failure of spot welds", Engineering Fracture Mechanics 77 (2010) 1224–1239

[10] Hyoe P., Colegrove P. & Shercliff H. "Thermal And Microstructure Modeling In Thick Plate Aluminium Alloy 7075 Friction Stir Welds". 2003.

[11] AYSYS User's Manual: Elements, vol III, Swanson Analysis Systems.

[12] G. Oertelt, S. S. Babu, S. A. David & E. A. Kenik. "Effect of Thermal Cycling On Friction Stir Welding of 2195 Aluminium Alloy".

[13] Dragi Stamenkovic, Ivana Vasovic. "Finite Element Analysis OF Residual Stress in Butt Welding Two Similar Plates", Scientific Technical Review, Vol. LIX, No. 1, 2009, Pages No. 57-60.

[14]Ahmed Khalid Hussain, Syed Azam Pasha Quadri. "Evaluation of parameters of friction stir welding for aluminium AA6357 alloy",International Journal of Engineering Science and Technology Vol. 2(10), 2010, 5977-5984.

[15]Adelin Tuta, Dorel Spiru Dumitriu. "Finite Element Analysis of Polyethylene Pipe Heating During Welding with A Heating Plate", AXUL. XV, NR.1, 2008, ISSN 1453-7397.

AUTHOR PROFILE

AMAY DEORAO MESHRAM
Student of M.Tech. in Thermal Engineering
Government College of Engineering
Amravati – 444604,
Maharashtra, India