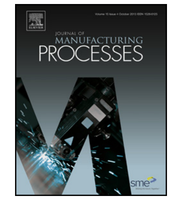
Journal of Manufacturing Processes 23 (2016) 278–286

Contents lists available at ScienceDirect

Journal of Manufacturing Processes

journal homepage: www.elsevier.com/locate/manpro



Technical Paper

A study on the variation of forces and temperature in a friction stir welding process: A finite element approach

Rahul Jain [a], Surjya K. Pal [a,*], Shiv B. Singh [b]

[a] Department of Mechanical Engineering, Indian Institute of Technology Kharagpur, 721302, India
[b] Department of Metallurgical and Materials Engineering, Indian Institute of Technology Kharagpur, 721302, India

ARTICLE INFO

Article history:
Received 23 November 2015
Received in revised form 7 March 2016
Accepted 15 April 2016
Available online 27 May 2016

Keywords:
Friction stir welding
Finite element simulation
Temperature distribution
Pin shape

ABSTRACT

A three dimensional coupled thermo-mechanical finite element model (FEM) is proposed to simulate a friction stir welding (FSW) process based on Lagrangian incremental technique. Since FSW is a large deformation process, workpiece is considered as a rigid visco-plastic material. The model has been developed for predicting forces, spindle torque, temperature and plastic strain for a butt welding between two AA2024-T4 metals having thickness of 5.9 mm each. The developed model has been validated with experimental results (forces, spindle torque) obtained from literature. Maximum force is obtained during the plunging phase of the tool and this makes tool susceptible to failure. Forces and spindle torque reduce with the increase in rotational speed due to increase in heat generation rate which is also reflected in temperature distribution. Effect of welding speed and frictional boundary condition are studied. Conical pin shape produces higher material velocity as compared to cylindrical with reduced plunge force.

1. Introduction

FSW is a joining process which welds two materials in solid state. This technique was successfully executed in the last two decades in TWI Cambridge [1]. Since FSW welds materials in solid state, it has got many advantages over conventional welding techniques, such as no consumable materials, no defects related to solidification of the material, higher weld strength, reduced power consumption, etc. [2]. A rotating, non-consumable, specifically designed tool plunges on the faying surfaces of the clamped workpiece to generate sufficient heat and to stir the material beneath the shoulder, and subsequently, a relative motion is given to the tool in weld-line direction to perform the weld. A schematic of different stages of friction stir welding process is shown in Fig. 1. Heat generation due to friction between the tool and the work-surface, and plastic deformation inside the workpiece are responsible to bring the material into a viscous state [3].

Experimental investigation reveals the generation of higher forces in the plunging stage as compared to the welding stage [4]. Different methodologies have been implemented to experimentally measure the forces such as traditional load cell to measure the axial force[5], three component rotating type piezoelectric dynamometer to measure forces and torque generated during the process [6], responses of electrical motor (like power, current, etc.) to measure the forces and torque [7,8].

FEM of a FSW process is complex due to highly nonlinear contact interaction between workpiece and tool, large deformation of workpiece leading to higher strains and strain rates. Some of the notable FE models are discussed below.

Chao et al. [9] have simulated heat transfer during FSW as two boundary value problems: firstly, steady state heat transfer for the tool, and second a transient heat transfer problem for the workpiece. They have considered heat generation due to frictional force only neglecting the heat generation due to plastic deformation. Based on temperature, residual stress over the welding domain was also calculated. Deng and Xu [10] carried out a two dimensional analysis in ABAQUS to predict material flow and material spatial velocity. They reported that predicted material flow pattern is comparable to experimental pattern. Chen and Kovacevic [11,12] have developed three dimensional symmetric thermo-mechanically coupled model to simulate a FSW process by using ANSYS to predict temperature history and stress distribution. They have also predicted forces by integrating the stress over the area. Buffa et al. [13] have simulated three dimensional coupled thermo-mechanical analysis using Lagrangian implicit code in DEFORM-3D. They have simulated plunging and welding stages by defining workpiece as a rigid visco-plastic material. Trimble et al. [6] have simulated force generation during FSW process using Johnson-Cook material model and compared force, yield strength, % elongation between a cylindrical pin and a threaded cylindrical

* Corresponding author. Tel.: +91 3222 282996.
E-mail address: skpal@mech.iitkgp.ernet.in (S.K. Pal).

http://dx.doi.org/10.1016/j.jmapro.2016.04.008

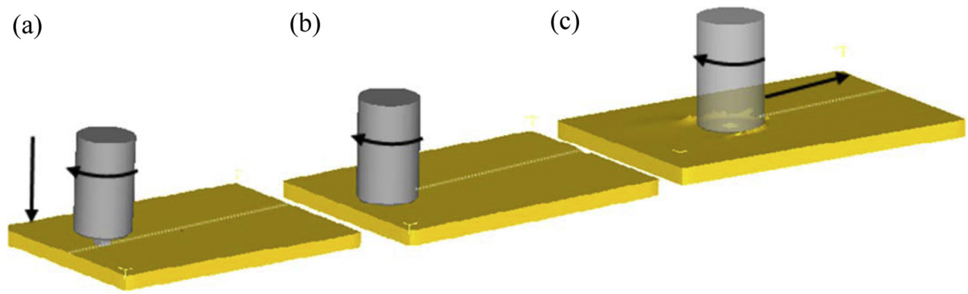


Fig. 1. Schematic of various stages of FSW process (a) plunging (b) Dwelling (c) Welding [3]. **Note:** Dashed line indicates abutting edges, while arrows indicates the tool movement direction.

pin. Instead of constraining the mesh nodes, they have enclosed the workpiece inside the box to restrict the rigid motion. Trimble et al. [14] have studied the effect of different pin shape and rotational speed for increased welding speed. They achieved a good weld with triflute pin at a speed of 355 mm/min. Assidi et al. [15] have adopted Arbitrary Eulerian Lagrangian (ALE) method to predict and validated the forces and power with the experimental values. They have compared coulomb's law of friction and Norton's frictional model. Some other researchers have simulated the material flow and force generation for magnesium alloy and have also studied the effect of shoulder diameter and pin diameter on strain, temperature and heat affected zone [16,17]. A few researchers [18,19] have also studied the different constitutive material models and their influence on the output responses of the FSW process.

Measurement of forces and torque is quite difficult as it involves costly equipment and extra fixture. One of the alternative ways of estimating forces is through the numerical modeling of the process. Though modeling of FSW has been done by a lot of researchers but only a few [6,20,15] have predicted the forces generated during the process. FSW process is performed for a range of input process parameters (rotational speed, welding velocity, etc.) and have various output responses viz. temperature, forces, torque, etc. Mostly, validation of FEM model is carried out with numerically predicted temperature-time plot for a single input process parameter. A complete validation of model can only be achieved if it is validated with different output responses and under different input conditions. Also, modeling is only carried out for a steady state welding condition [21–23] or for plunging and welding stages [20,24]. A complete realistic modeling of a FSW process, including all the three stages having proper duration i.e. plunging, dwelling and welding has not yet done.

In the current research work, a fully coupled 3-dimensional thermo-mechanical model is developed based on Lagrangian implicit technique to simulate the three stages of a FSW process. Workpiece is considered as a rigid visco-plastic material and sticking friction boundary condition is defined at the interface. The model is capable of predicting forces and spindle torque generated during the process as these parameters influence the tool design. Distribution of temperature and plastic strain are also predicted, which influences the microstructure and grain distribution in the nugget zone. Axial force, welding force and spindle torque are validated with the experimental results published in the literature [8] for different input parameters. Effects of tool rotational speed, welding speed and pin shape (cylindrical and tapered cylindrical) on the output responses (forces, torque, and temperature) are also studied.

2. Simulation details

FSW is a large deformation process and for the simulation, it is very crucial to control the mesh distortion, with the use of effective re-meshing technique. Contact interaction between tool and workpiece is nonlinear and precise definition of frictional boundary condition is also required. Keeping the above criteria in mind, DEFORM-3D software is selected to simulate the process. Apart from efficient re-meshing capability the software also has a unique point tracking method, which is helpful in simulating material flow and to visualize the deformation pattern. The FEM model has been validated with forces and spindle torque values obtained from the research work of Su et al. [8]. They have monitored and recorded the electrical signals of three phase AC induction motor in FSW machine. Based on the electrical signals forces and spindle torque are calculated. The experiments were performed for different input process parameters.

FSW modeling is divided into three stages viz. plunging, dwelling and welding stages. During assembly of workpiece and tool; tool is titled by 2.5° towards its trailing edge. During the plunging stage, tool moves in negative *Z* direction at a defined rotational speed (600, 800 and 1000 rpm are used in this study) and a feed velocity of 8 mm/min, afterwards dwell time of 10 s is defined, in which tool rotates at the same rpm in its own position to further increase the temperature and make material beneath the material softer, and finally welding speed is defined to the tool to simulate the welding stage till steady stage is achieved. Time required for plunging is 43 s to achieve a plunge depth of 0.1 mm.

All simulations are performed with the similar procedure explained above with a tilt angle of 2.5 degrees, plunge velocity of 8 mm/min, plunge depth of 0.1 mm, and welding velocity of 80 mm/min. Three rotational speeds (600 rpm, 800 rpm and 1000 rpm) are chosen to investigate their effect on output responses. All input parameters and dimensions are same as the experimental data of Su et al. [8].

2.1. Geometric modeling and boundary conditions

Workpiece and tool dimension are taken as per the experimental data available in literature [8]. Workpiece material is AA2024-T4 aluminum alloy with 5.9 mm thickness and has a length and width of 120 mm and 60 mm, respectively each. Workpiece is modeled as rigid visco-plastic material and meshed with tetrahedral elements. To improve the computational efficiency of the model, biased meshing is defined using mesh window option as shown in Fig. 2. Here, a mesh window at the interface is created and finer mesh with a uniform element size of 0.8 mm is defined, such that it has approximately 7 nodes along the thickness direction and remaining workpiece is meshed with the minimum element size of 2 mm, with a size ratio of 3 to have 40,000 tetrahedral elements on workpiece. Tool is modeled as a rigid body with defined thermal degree of freedom to calculate the heat transfer between workpiece and tool. Since yield strength of tool material is much higher as compared to workpiece material, assumption of rigid tool is valid. Tapered cylindrical pin tool has following dimensions: shoulder