Computational Materials Science 79 (2013) 540–546

Contents lists available at SciVerse ScienceDirect

Computational Materials Science

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

Computational fluid dynamics studies on heat generation during friction stir welding of aluminum alloy

Gao-qiang Chen, Qing-yu Shi *, Yu-jia Li, Yan-jun Sun, Qi-lei Dai, Jin-yao Jia, Yu-can Zhu, Jian-jun Wu

Department of Mechanical Engineering, Tsinghua University, Beijing 100084, PR China

ARTICLE INFO

Article history:
Received 3 February 2013
Received in revised form 14 June 2013
Accepted 1 July 2013
Available online 3 August 2013

Keywords:
Friction stir welding
Heat generation
Numerical simulation
Computational fluid dynamics
Heat flux distribution

ABSTRACT

Friction stir welding (FSW) has proved to be a successful joining technology for aluminum alloys and many other metallic materials. The severe plastic deformation of solid-state metal during FSW made it a fully coupled thermo-mechanical process. In order to quantitatively study both the total heat generation and the spatial distribution of the heat flux, a thermo-mechanical coupled model based on computational fluid dynamics was presented in this study. The heat generation, the temperature field and the material flow pattern were simulated in a fully coupled way. The simulated temperature distribution agreed well with the experimental results. The total heat generation was found to be proportional to the 0.75 power of the tool rotating speed. The spatial distribution of the heat flux around the FSW tool was almost axisymmetric about the tool axis. A radial distribution function was defined to describe the heat flux in different rotating rates. The radial distribution function in the shoulder region was fitted to a parabolic function.

1. Introduction

Friction stir welding (FSW) is an advanced solid-state joining technology invented at The Welding Institute (TWI) in 1991. It is a very successful joining technology for aluminum alloys and many other metallic materials [1,2]. Specifically, the process was carried out by plunging a rotating FSW tool into the surface of rigidly clamped sheets and traversed along the weld lines. Large amount of heat was generated during the process, and significant plastic flow was caused by the rotating tool. In result, dynamic recrystallization was brought to occur. FSW joints were formed as a result of the severe plastic deformation and dynamic recrystallization. Key issues about FSW in the published researches included the heat generation [2–6], material flow [1,2,7–10], microstructure evolution [1,2,11–13] and defects formation [2,14–16] during the process, the resulted properties[1,2,17–19] of the joint and the design of the FSW tool [1,2,20].

Heat generation during FSW was a fully-coupled thermo-mechanical process [3]. The transient temperature distribution during FSW was related to both the total amount and the distribution of the heat generation. Meanwhile, the heat generation during plastic deformation was also related to the mechanical property, i.e. the flow stress of the metal, and the transient deformation rate [3,21,22]. Note that the flow stress of aluminum alloy was significantly temperature and strain rate dependent, the total heat generation, the temperature distribution and the material flow pattern were affected by each other in FSW. Since thermal modeling was the basis of other models, evaluation of both total amount and distribution of heat flux during FSW was required [4]. In order to evaluate the heat generation in FSW, the underlying physics, such as the material flow, should be included. This paper provided a first quantitative investigation of the total amount and the distribution of the heat flux by a fully coupled thermal–mechanical model.

In the published works, experimental methods and theoretical analysis had been employed to estimate the heat input during FSW. By using the experimental methods, Pew et al. [23] measured the torque carried by the tool to estimate the heat input during FSW. Based on the theoretical analysis, different theoretical heat source models had been proposed to estimate the heat flux. In these models, the heat flux was calculated based on the contact shear stress on the tool/material boundary. Chao et al. [24] defined the contact stress based on the Coulomb's law to simulate the temperature distribution. Shercliff et al. [25] set the contact shear stress as 5% of the yield stress at room temperature. Schmidt et al. [5,26] proposed to calculate the total heat generation and its distribution analytically from the uniformly distributed contact shear stress on the tool/material boundary. In Schmidt et al.'s studies [5,26], the contact shear stress was determined by the contact state. Li et al. [27] provided an auto-adaptive heat source model for FSW by assuming a temperature dependent contact state. However, the material flow pattern was difficult to be taken into consideration by using the experimental methods and theoretical analysis.

* Corresponding author. Tel.: +86 10 62784578; fax: +86 10 62773637.
E-mail address: shqy@tsinghua.edu.cn (Q.-y. Shi).

0927-0256/$ - see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.commatsci.2013.07.004

In recent years, numerical models based on Computational Solid Mechanics (CSM) and Computational Fluid Dynamics (CFD) had been established to simulate the material flow during FSW.

Based on CSM method, Schmidt et al. [28] established a material flow model for FSW. The Arbitrary Lagrangian–Eulerian (ALE) formulation was adopted to avoid unacceptable element distortion. The frictional force given by the Coulomb's law of friction was applied on the tool/material boundary. Heat was generated from the sliding friction on the boundary and plastic deformation in the model. Both the velocity and temperature could be simulated. Similar models based on CSM were also provided by Zhang et al. [29] and Grujicic et al. [30]. In these models, however, the material flow pattern was obtained only when the tool penetrated through the work pieces [6–8]. Because of the limitation in the above geometric models, the physical processes represented by models based on CSM were significantly different from real FSW.

The CFD method was another important approach employed to simulate the material flow pattern in FSW. The work presented by Colegrave et al. [9] was one of the earliest 3D thermal–mechanical-coupled models of FSW based on CFD. In the study, the material viscosity was considered as a function of strain rate and temperature. The material close to the tool was assumed to stick on the tool. Heat was generated fully by viscous dissipation. Based on this model, both the temperature and material flow pattern were simulated, but both the temperature and the size of the deformation zone were over-predicted. Nandan et al. [10] calculated the heat input from the contact stress in their material flow model based on CFD. In their model, the 3D distributed heat generation from viscous dissipation was taken as a small fraction of the total heat generation. They reported that the asymmetric feature of the temperature field would not be simulated if the small part of heat from the viscous dissipation was ignored. Colegrave et al. [6] established a 2D thermo-fluid model, coupling with a 3D thermal model to calculate the heat generation from the axisymmetric flow pattern and the material properties. In their model, the heat generation was coupled with the material flow pattern.

Though lots of researches have been carried out to simulate the material flow during FSW, quantitative analysis of the total amount and the spatial distribution of the heat flux during FSW has not been reported yet. This paper provides a first attempt to investigate both the total amount and the distribution of the heat flux by a fully coupled thermal–mechanical model based on CFD. Different from the published researches, this paper improves the constitutive equations in order to predict the flow stresses at high temperatures more accurately. Based on the improved constitutive model, the heat generation, material flow pattern and the temperature distribution are simulated. The computed temperature distribution is compared with a set of published measured data. Special attention is paid to quantitative evaluation of the total heat generation and its spatial distribution. Simulations using different welding parameters are carried out to investigate the effect of welding parameter on both the total heat generation and spatial distribution of the heat flux.

2. Simulation model

2.1. Governing equations

The conservation equations of mass, momentum and energy were solved, using the commercial package, ANSYS Fluent [31]. The continuity equation and steady-state momentum conservation equation for incompressible single-phase flow were given by:

$$\nabla \cdot (\rho \vec{v}) = 0 \tag{1}$$

$$\nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\mu(\nabla \vec{v} + \nabla \vec{v}^{\mathrm{T}})) \tag{2}$$

where ρ was the density, μ was the non-Newtonian viscosity, p was the pressure and $\vec{v}$ was the velocity of material flow. The energy conservation equation was given by:

$$\nabla \cdot (\vec{v}H) = \nabla \cdot (k\nabla T) + S_V \tag{3}$$

where H was the enthalpy, T was the temperature, k was the thermal conductivity and S_V was a spatial source term.

2.2. Geometric model

The geometric model was shown in Fig. 1. The geometric model was obtained by subtracting the FSW tool from the work piece. The size of the work piece was 60 mm × 40 mm × 3.1 mm. The tilt angle and the concave shoulder were not employed. The FSW tool was taken as a rigid body, and was not included in the geometric model. The geometric model was meshed by hexagonal grids before simulation. The diameter of the tool shoulder was 12 mm, and the pin diameter was 3 mm at the root. The pin length was 2.8 mm. The rotating direction and welding direction were shown in the figure. The welding parameters were taken to be 920 rpm, 20 mm min^{-1}, the same as Ref. [32].

2.3. Material properties

Aluminum alloy 6061 was selected as the target material. The density of AA6061 was assumed to be constant and was taken as 2700 kg m^{-3}. Temperature dependent thermal physical properties were taken from Ref. [33]. The viscosity of the material was considered to be both temperature and strain rate dependent. The viscosity was calculated based on the following formulation [9,34]:

$$\mu = \frac{\sigma}{3\dot{\varepsilon}} \tag{4}$$

where σ was the flow stress and $\dot{\varepsilon}$ was the strain rate. The expression of flow stress was given by [35]:

$$\sigma_e = \frac{1}{\alpha}\sinh^{-1}\left[\left(\frac{\dot{\varepsilon}}{A}\exp\left(\frac{Q}{RT}\right)\right)^{1/n}\right] \tag{5}$$

where A (=8.86 × 10^6 s^{-1}), α (=0.045 MPa^{-1}) and n (=3.55) were material constants, Q (=145 kJ mol^{-1}) was the temperature-independent activation energy, and R was the gas constant. The parameters in the above Eq. (5) were taken from Ref. [36].

It could be found from Fig. 2 that large discrepancy existed between the predicted flow stress by Eq. (5) and the measured yield stress. In this paper, the expression of flow stress in Eq. (5) was modified by multiplying a temperature dependent factor. Fig. 2 showed that the modified flow stress agreed well with the yield stress especially at high temperatures. Similar modifications of

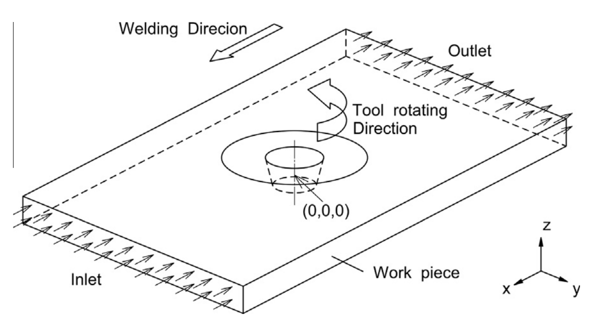

Fig. 1. Geometric model in the simulation. (0,0,0) was the origin of coordinates.