Thermo-Mechanical Modeling of Friction-Stir Welding Tool Used in Aluminum Alloys Joints

Jonathan Fabregas Villegas¹, Jennifer Villa Dominguez², Guillermo Valencia Ochoa³ and Jimy Unfried-Silgado⁴

¹ Grupo de Investigación IMTEF, Programa de Ingeniería Mecánica Universidad Autónoma del Caribe, Colombia
² Grupo de Investigación GI Fourier, Facultad de Ingeniería Mecánica Universidad Antonio Nariño, Colombia
³ Grupo de Investigación KAI, Programa de Ingeniería Mecánica Universidad del Atlántico, Colombia
⁴ Grupo de Investigación, Ciencia y Tecnología ICT, Departamento de Ingeniería Mecánica Universidad de Córdoba, Colombia

Copyright © 2017 Jonathan Fabregas Villegas et al. This article is distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Abstract

In this paper we study the thermomechanical effects generated in the tool used in the friction-stir welding (FSW) process by finite element analysis (FE) in welded joints of structural aluminum AA1100. A modeling was carried out using Structural Multiphysics and Computational Fluid Dynamics (CFD) couplings using tools with three different shoulder profiles (flat, spiral and concentric), in order to evaluate the influence of its geometry on the heat generated during the process and heat transferred and mechanical stress on the tool. As a result, thermal profiles, stress analysis, heat generated in the process and torque transmitted in the joint and tool interaction zone were obtained. In addition, the values of heat loss through the tools were estimated, which are less than 10%, with 87% of the melting temperature reached of the joints, corroborating the established in the literature.
Keywords: Thermo-mechanical modeling, CFD, tool, friction stir welding and heat

1 Introduction

The friction and stir welding (SFA) process was developed and patented by W. Thomas at the Welding Institute (TWI) in 1991 [1] and consists of a solid state joining process that is executed using a tool that has a Non-consumable specific geometry as shown in Fig. 1. The process method is developed using an appropriate geometry tool to penetrate the material while rotating at a certain speed until the tool shoulder makes contact with the surface of the materials to be joined, in which due to the pressure and the friction generates heat increasing the temperature, causing plastic deformation in a stirred area. Maintaining this condition for a certain time, then initiate a longitudinal movement of the work table until achieving the bonding of the joints. In the SFA process heat losses occur due to the convection of the plates with the surroundings, loss of heat of the plate by conduction towards the work table, and losses of heat by conduction by the contact of the tool with the plates. In finite element modeling processes used for SFA in AA1100 aluminum plates, maximum temperatures of 89.4% of the melting temperature of the material [2] have been obtained. For the aforementioned in this work, a finite element modeling is performed with the purpose of obtaining the thermal distribution product of the losses by conduction through the tool used in the SFA process for welding joints of aluminum alloy AA1100 H14 when tool material H13 steel is used.

Fig 1. Friction Stir Welding Process

The finite element simulation methods are used more frequently in the study of the friction stir welding process, due they allow to determine the behavior, the distribution of stresses and temperatures in the agitated zone and its surroundings [3, 4]. Additionally, these data have allowed to feed thermomechanical models in order to establish the efficiency of the process, which is at 80% and 97%. Among
these, the study by C. Chen and R. Kovacevic [5], which carried out the modeling of the thermomechanical effect of an SFA process on 6061-T6 aluminum plates, is considered. This study considers the mechanical reaction of the working tool in the heat generation, obtaining the contours of process temperatures. These validate their study by analytically obtaining the residual stresses of the process, which later is compared with those obtained experimentally by X-rays. Schmidt and Hattel [6] employ a pseudomechanical thermal model where they relate the stresses generated by the thermoplastic work to determine the heat input during the process and the effective thermal energy to generate the welding joint. These relate the total heat to the heat of the welded zone, obtaining thermal efficiencies between 83% and 88%.

This work seeks to simulate the friction stir welding process for different tool profiles in order to establish the temperature, stress, torque and heat at which it is subjected, to compare and establish the effect of the geometry on these variables.

2 Materials and Methods

A finite element modeling of the SFA process using multiphysics couplings to perform the interaction of the thermal properties of the process is done, generating volumetric mesh and border identification for the definition of the relevant models for the solution through numerical analysis.

The physical properties of the materials are defined as well as the process conditions for starting up the numerical solution, by entering as parameters a tool rotation speed of 1000 RPM, a holding time of 10 s and a room temperature of 25 °C. The physical properties of the material for the joints and tool are shown in Table 1 and 2.

<table>
<thead>
<tr>
<th>Fe</th>
<th>Si</th>
<th>Cp (J/kg K)</th>
<th>σ_y (MPa)</th>
<th>K (W/m K)</th>
<th>T_f (K)</th>
<th>Espesor (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.95</td>
<td>0.95</td>
<td>904</td>
<td>120</td>
<td>220</td>
<td>916</td>
<td>6</td>
</tr>
</tbody>
</table>

Table 1. Physical Properties of Aluminum AA1100

<table>
<thead>
<tr>
<th>P (kg/m^3)</th>
<th>Cp (BTU/hr ft °F)</th>
<th>E (GPa)</th>
<th>K (W/m K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>7750</td>
<td>17</td>
<td>207</td>
<td>27</td>
</tr>
</tbody>
</table>

Table 2. Physical Properties of H13 Steel

2.1 Thermomechanics in FSW

In the FSW process it starts with a preheating of the contact zones when a speed of rotation of the tool is applied on the joints, which generates a heat by friction that is estimated by the following equation:

$$Q_{total} = \frac{2}{3} \pi \cdot \omega \cdot \tau_{contact} \left( R_h^3 - R_{bp}^3 + \frac{3}{4} \cdot \frac{H}{\cos B} \left( 2 \cdot R_{bp} - H \cdot \tan B \right)^2 + \left( R_{bp} - H \cdot \tan B \right)^3 \right)$$  (1)
Where $Q_{\text{total}}$ is the heat generated at the shoulder, $\omega$ is the speed of rotation of the tool, $H$ is the height of the pin, $R_h$ is the radius of the shoulder, $R_{bp}$ is the radius of the base of the pin, $R_p$ is the radius of the pin, $B$ is the angle of inclination of the cone of the pin, $\tau_{\text{contact}}$ is the shear shear force [7, 8].

The heat generated in the contact zone is distributed in a heat towards the joints, a heat towards the tool, a heat towards the surroundings. The heat directed towards the tool is represented by the following equations:

**Heat by conduction towards the tool:**

$$1 \frac{\partial}{\partial r} \left( kr \frac{\partial T}{\partial r} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right) + \dot{\epsilon}_{\text{gen}} = \rho c \frac{\partial T}{\partial t}$$  \hspace{1cm} (2)

Where $r$ is the radius of the cylindrical object, $\dot{\epsilon}_{\text{gen}}$ is the heat generated, $\rho$ is the density of the material, $z$ and $r$ represent the radial and axial coordinates, and $t$ is the elapsed time.

**Heat of the tool towards the surroundings:**

$$Re_D = \frac{v_D}{v}$$  \hspace{1cm} (3)

$$\bar{N}u_D = C Re_D^n Pr^{1/3}$$  \hspace{1cm} (4)

$$\bar{h} = \frac{\bar{N}u_D k}{D}$$  \hspace{1cm} (5)

$$\dot{Q} = \bar{h} A_s (T_a - T_s)$$  \hspace{1cm} (6)

Where $Re_D$ is the dimensionless number of Reynolds, $\bar{V}$ is the ambient air velocity, $D$ is the diameter of the tool, $v$ is the kinematic viscosity, $\bar{N}u_D$ is the dimensionless number of Nusselt, $Pr$ is the dimensionless number of Prandtl, $k$ is the thermal conductivity coefficient of ambient air, $\dot{Q}$ is the heat transfer rate by convection, $A_s$ is the contact surface area between ambient air and the tool, $T_a$ is the ambient air temperature and $T_s$ is the surface temperature of the tool [8].

The variable $\tau_{\text{contact}}$ of equation 1 is the shear stress produced in the contact zone which represents the shear stress of the joints caused by the speed of rotation and the crushing of the material when sliding in plastic state, and in the tool represents the shear stress due to the reactive torque produced by the strength of the joints in the profiles of the contact zone of the tools. Due to the aforementioned,
this variable is used as the object of study since it allows obtaining the mechanical effects of the FSW process.

2.2 Numerical Modeling and Boundary Conditions

A tool geometry was realized that counts on a body, shoulder and a threaded pin whose dimensions are: total length of 118 mm, pin radius of 1.5 mm, radius of pin base of 3 mm, pin height of 5.33 mm, 15° pin tilt angle, 9 mm shoulder radius and tool body radius of 12 mm. Three geometric profiles were made for the base of the shoulder, which have a spiral shoulder shape, concentric shoulder and flat shoulder. In order to identify the behavior of the heat to the tool, the temperature distributions and mechanical effects the mentioned geometric profiles are used. These geometric representations are shown in Fig 2.

![Fig 2. Tool Profiles in H13 Steel](image)

The CAD files of the geometries are developed by means of finite elements the volumetric meshes of the geometries, in which tetragonal elements were used as shown in Fig 3.

![Fig 3. Volumetric mesh of the tool for FSW](image)
Once the meshes have been developed, the parameterization of variables by bounding conditions is carried out by making a structural analysis for the effects on the tools and analysis by CFD for visco plastic effects for the joints, making an interaction using couplings between interfaces.

Once the solvers of the computational tools are started up, the behavior of the identified variables is obtained while the process is performed, which allows to obtain between the behavior of the variables the shear stress in the zone of contact of the tool node to node, which allows by using of equation (1) to estimate the heat reached in this area of the tool.

3 Results and Discussion

When the numerical solutions are finished the profiles of temperature, the profiles of the shear forces, the torque generated in the process, the heat lost to the tool, and the efficiency of the process are obtained.

In the Fig. 4 the temperature distribution profiles in the three geometries studied are shown, in which it can be observed that for the spiral-shaped tool the temperatures reached are higher due it has a larger area of contact and higher torque in the process, followed by the tool with concentric shoulder shape and finally the tool with a flat shoulder shape.

![Fig 4. Temperature distribution on tool’s profiles](image-url)
In Fig 5 the shear forces obtained in the contact zones of the tools with the joints are shown, these are produced by the reactive torque products of the rotation speed and the visco plastic resistance of the joints when resisting the deformation.

![Shear stress on FSW tools](image)

**Table 3. Variables obtained from the FSW process**

<table>
<thead>
<tr>
<th>Hombro</th>
<th>Q herramienta [W]</th>
<th>Q total [W]</th>
<th>Torque [N m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plano</td>
<td>32,1534855</td>
<td>1365,03123</td>
<td>13,3421</td>
</tr>
<tr>
<td>Espiral</td>
<td>15,0315203</td>
<td>1423,26626</td>
<td>13,7347</td>
</tr>
<tr>
<td>Concentrico</td>
<td>31,3634618</td>
<td>1415,17579</td>
<td>13,8134</td>
</tr>
</tbody>
</table>

**4 Conclusions**

From the results obtained, it can be seen that the geometry of the tool shoulder has a great influence on the thermal and mechanical effects of the FSW process.

It can be validated that the temperatures reached are less than 80% of the fusion temperature of the joints. This result is appreciated by other authors and indicates that the welding is done in a plastic state.
By performing FSW simulations using finite elements, it was possible to obtain estimative values that are very difficult to obtain experimentally if they do not have adequate instrumentation.

The results reflect what has been obtained in practice indicating that finite element modeling is a good practice for further exploration of the FSW process.

References


Received: November 15, 2017; Published: December 21, 2017