Numerical Simulation of Temperature Field and Residual Stress of T Welded Joint of Aluminum Alloy

Shao-hong YANG, Zhao-yi WEI* and Fang HUANG
Department of Basic Courses, Naval Univ. of Engineering, Wuhan 430033, China
*Corresponding author

Keywords: Aluminum alloy, T-joint, Residual stress, Temperature field.

Abstract. The numerical simulation of residual stress of aluminum alloy welded joints can optimize the welding process and increase the joints strength. In this paper, the Gauss distribution heat source model and ANSYS finite element method were used to simulate the MIG welding of T-joint of aluminum alloy. The actual weld molten was obtained by experiment and compared with the numerical results of temperature field. The residual stress of T-joint of aluminum alloy under three kinds of current was calculated by elastic-plastic theory and incremental method. The results show that the finite element calculation results are in good agreement with the actual molten pool, which indicates the accuracy of the finite element simulation calculation. The welding residual stress mainly concentrates on the weld zone, and the longitudinal tensile stress on the weld is larger and the longitudinal compressive stress is smaller. With the increase of welding current, the maximum residual tensile stress of the weld increases gradually.

Introduction

Aluminum alloy welding structure is widely used in automobile, ship, high-speed railway, aviation, aerospace and other fields. Because welding is a process of local rapid heating to high temperature and subsequent rapid cooling, and the characteristics of high thermal expansion coefficient, high thermal conductivity, small elastic modulus and low yield strength at high temperature of aluminum alloy make the aluminum alloy junction. The residual stress of the components is large, which has a great influence on the stiffness and strength of the members. The fatigue crack and stress corrosion crack are easy to be induced by large welding residual stress, which reduces the load-carrying capacity and service life of aluminum alloy welded structures [1]. In order to reduce the residual stress and its hazards, it is necessary to accurately predict the distribution characteristics of welding residual stress.

The residual stress data measured by experimental means are usually only a few local measuring points, which cannot reflect the distribution of stress and deformation of welded parts as a whole. By means of thermo-elastic-plastic numerical simulation of welding process, the magnitude and distribution law of welding residual stress and deformation can be fully reflected. Only a few validation experiments are needed to prove the practicability of the numerical method in dealing with a certain problem. A large number of other project designs, optimum process methods and welding parameters can be screened by the method. The computer is complete. It plays a key role in controlling and reducing welding residual stress and deformation, and saves a lot of manpower, material and financial resources [2].

In welding numerical simulation, the accurate selection and establishment of welding heat source model is very important to the efficiency of numerical calculation and the accuracy of calculation results. Sen Li [3] studied the deformation of TIG welded T-joints of aluminum alloy, Liang Zhu [4] studied the mechanism of the influence of the uneven mechanical properties of materials on the yield strength and tensile strength of butted joints, Liang Zhang [5] et al. studied and simulated the welding process of 2024 aluminum alloy for medium and heavy plate, and Pi-yang Jiang [6] simulated by finite element method and X-ray. Residual stress characterization and stress relief by induction heat treatment for 2219 aluminum alloy with different welding methods were studied by linear diffraction.
Lai-shun Wang\textsuperscript{[7]} used ANSYS finite element software to simulate the residual stress and deformation of 5083 aluminum alloy butt plate welded joint. Si-qun Ma\textsuperscript{[8]} used SYSWELD finite element software to study the effect of welding speed on residual stress of multi-pass welding of aluminum alloy. Wei Liang\textsuperscript{[9]} used thermal-elastic-plastic finite element method to predict the deformation and residual stress of welded joints considering the softening of aluminum alloy joints. There are few studies on residual stresses in T welded joints of aluminum alloy.

Aiming at typical T-type welded joints, the welding process of aluminum alloy is numerically simulated by using Gauss distribution heat source model and ANSYS finite element method. The influence of temperature dependence of material thermal physical properties and convection conditions on temperature field in general welding process is considered, and the weldability of aluminum alloy is weakened. The temperature field and welding residual stress field can be obtained without even dealing with those factors which have little influence on the temperature field, which provides a theoretical basis for optimizing the actual welding process and obtaining higher joint performance.

**Finite Element Modeling**

The T type welded joint is made of 5083 aluminum alloy with a size of 400mm * 100mm * 6mm and a length of 400mm. The profile is made of 6082 aluminum alloy with a size of 50mm * 50mm * 4mm * 4mm and a length of 350mm. The finite element model of the simulated T type double-sided welded joint is shown in Figure 1.

![Figure 1. FEM model.](image)

The finite element is a hexahedral element, and the mesh is divided into two dimensions. Because of the high temperature and stress gradient in the weld and its vicinity, the mesh size at the weld seam is controlled to 5 mm x 1.5 mm. The mesh size of the components far from the weld seam increases gradually, and the mesh becomes sparse gradually. This not only ensures the accuracy of calculation, but also reduces the workload of gauging greatly. The number of nodes in the whole model is 19298 and the number of cells is 11940 (including 7358 surface effect units).

**Welding Heat Source.** For the usual welding method, the heat source model with Gauss function distribution can obtain satisfactory simulation results. The welding arc is transmitted continuously to the weldment through heating spots. However, the heat distribution is uneven and diverges from the center to the edge. Usually, the distribution of heat flux on heating spots can be approximately defined by Gauss's function:\textsuperscript{[2]}

\[ q(r) = q_m \exp(-3 \frac{r^2}{R^2}) . \]  

(1)
Among them: $q_m$ is the maximum heat flux of the heating spot center; $r$ is the effective heating radius of the arc, $R$ is the distance between the node and the arc heating spot center.

**Material Parameters.** The main parameters of materials are thermal conductivity, convective heat transfer coefficient, density, melting point, specific heat capacity and initial temperature of welding parts. Poisson's ratio, modulus of elasticity, coefficient of thermal expansion and yield limit should also be defined after thermal-mechanical conversion. The thermal physical properties of materials are nonlinear when analyzing the transient temperature field. The method of ANSYS to deal with the thermophysical properties of materials is to give the corresponding values of several key temperature points, and the other temperature values are obtained by ANSYS linear interpolation or extrapolation. Among them, the thermal physical parameters of 5083 aluminum alloy refer to [10]. Reference material [11] for thermophysical parameters of profile 6082 aluminum alloy.

**Time Step.** The welding seam with 350 mm length is divided into 70 segments. When the welding speed is 5 mm/s, the action time of each segment thermal load is 1 s. When the next loading begins, the welding heat source load added in the previous step is eliminated, and the temperature field obtained by the next loading is the initial condition for the next loading. In this way, the transient welding temperature field which can simulate the moving heat source is loaded on the finite element model. This process can be realized by defining the cyclic statements through the ANSYS parametric design language (APDL).

**Calculation Results and Analysis**

**Comparison of Temperature Field.** The welding temperature field at 140A, 160A and 180A current $t=102.2s$ is calculated by ANSYS finite element method, as shown in Fig. 2. In order to verify the accuracy of temperature field calculated by simulation, semi-automatic MIG welding of T-type aluminum alloy was carried out. Three current of 140A, 160A and 180A, voltage and speed specifications were used to conduct welding tests, and actual welds under three different welding processes were obtained. The simulation results are compared with the actual molten pool, as shown in Figure 2.

From Figure 2, it can be seen that the simulation results of the finite element method are in good agreement with the actual molten pool, which shows the accuracy of the finite element simulation calculation and the rationality of parameter selection.

![Figure 2. Comparison between finite element simulation and actual weld pool.](image)

**Effect of Current on Welding Residual Stress.** After the temperature field of welded joints is obtained by ANSYS finite element method, the thermal analysis unit is transformed into corresponding structural unit. The boundary conditions are applied to the structural analysis. The node temperatures at each time point obtained by thermal analysis are read by cyclic statements as initial loads. Elastoplastic theory and incremental method [2] are used to calculate the response of welding process. Force field. Figures 3-5 show the distribution of welding residual stress in X-axis direction (parallel to weld direction, longitudinal) and Y-axis direction (perpendicular to weld direction, transverse) at 140A, 160A and 180A, respectively, in Pascal (Pa). For convenience, the hidden floor on both sides is hidden in the display.
It can be seen from Figure 3~5 that the welding residual stress is mainly concentrated in the weld zone. Along the X-axis direction, the tensile stress on the weld is larger, almost uniform distribution; near the weld area in the middle of the floor, the tensile stress appears, while the compressive stress is far from the weld area at both ends. Along the Y-axis direction, there are larger stresses at the starting and ending points of welding, and the numerical value is smaller than the residual stresses along the X-axis direction.

The maximum welding residual stress varies with the current as shown in Table 1. It can be seen from Table 1 that with the increase of welding current, the energy of heat source increases gradually, the maximum tensile stress along X axis increases gradually, while the maximum tensile stress along
Y axis is relatively small, and the maximum value increases with the increase of welding current. Because the tensile stress of the weld along the X-axis direction is relatively large, when T-joint is deformed by bending, it is easy to cause weld damage. The stress at both ends of the joint is relatively large, coupled with the stress concentration caused by the size change, it is easy to cause cracks at both ends of the joint, leading to fatigue failure.

Table 1. Variation of maximum residual stress of weld with current.

<table>
<thead>
<tr>
<th>Current</th>
<th>Maximum tensile stress along X axis</th>
<th>Maximum compressive stress along X axis</th>
<th>Maximum tensile stress along Y axis</th>
<th>Maximum compressive stress along Y axis</th>
</tr>
</thead>
<tbody>
<tr>
<td>140A</td>
<td>105</td>
<td>27.6</td>
<td>49.3</td>
<td>38.3</td>
</tr>
<tr>
<td>160A</td>
<td>107</td>
<td>25.2</td>
<td>61.5</td>
<td>43.8</td>
</tr>
<tr>
<td>180A</td>
<td>117</td>
<td>22.8</td>
<td>71</td>
<td>43.6</td>
</tr>
</tbody>
</table>

Summary

Aiming at MIG welding process, the temperature field distribution of aluminum alloy welding process is simulated by finite element method. The temperature field of T-joint of aluminum alloy is obtained by Gauss distribution heat source model. The distribution of welding residual stress is obtained by elastic-plastic model and incremental method. The influence of different current on residual stress is analyzed. The results show that the calculated temperature field agrees well with the actual molten pool, which indicates the accuracy of the finite element simulation calculation and the rationality of parameter selection. The welding residual stress mainly concentrates on the weld zone. On the weld, the tensile stress along the longitudinal (X-axis) direction is larger, and the compressive stress is relatively smaller. With the increase of welding current, the maximum tensile stress along the longitudinal direction of the weld zone increases gradually.

Acknowledgement

This research was financially supported by the National Natural Science Foundation of China (51479226).

References


