Advances in Engineering Software 99 (2016) 89–99

Contents lists available at ScienceDirect

Advances in Engineering Software

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

Research paper

Estimating inherent deformation in thin-plate Al-alloy joint by means of inverse analysis with the help of cutting technique

Wei Liang[a], Hidekazu Murakawa[b], Dean Deng[c,*]

[a] College of Mechatronics & Automotive Engineering, Chongqing Jiaotong University, Chongqing, 400074, China
[b] Joining and Welding Research Institute, Osaka University, 11-1, Mihogaoka, Ibaraki, Osaka 567-0047, Japan
[c] College of Materials Science and Engineering, Chongqing University, No. 174, Shazhengjie, Shapingba, Chongqing 400044, China

ARTICLE INFO

Article history:
Received 12 March 2016
Revised 12 April 2016
Accepted 8 May 2016
Available online 31 May 2016

Keywords:
Inverse analysis
Inherent deformation
Welding deformation
Buckling

ABSTRACT

The elastic finite element method based on inherent strain theory has been recognized as an effective tool to estimate the total welding deformation for large and complex welded structures. When this computational approach is employed to predict welding deformation in a weldment, one prerequisite is that the inherent deformations of each welded joint included in the welded structure should be known beforehand. The inverse analysis method based on the combination of measuring technology and finite element method can be used to obtain the inherent deformations for various welded joints. However, if buckling distortion occurs in a welded joint, it will be difficult for this method to accurately obtain the inherent deformations especially in thin-plate joints. To overcome this difficulty, an improved inverse analysis method with the help of cutting technique was developed in the current study. The effectiveness of the proposed method was demonstrated through obtaining the inherent deformations in an Al-alloy thin-plate joint with buckling distortion.

1. Introduction

Aluminum alloy, as lightweight structural material, has been widely used in the manufacturing of automobile, ship and passenger train. Welding technology is usually employed to assemble thin-plate elements because of its high productivity and design flexibility. Compared with the structural steels, the young's modulus of the aluminum alloy is only one third of them, and the thermal expansion coefficient is almost twice as large as that of the mild steel [1]. In addition, the thermal conductivity of aluminum alloy is significantly larger than that of steels. Therefore, it can be foreseen that welding deformation in an aluminum alloy welded structure is more serious than that generated in a steel-made weldment. Welding-induced distortion not only negatively affects the performance of product but also hinders the assembling process especially when automatic welding technology is used. Therefore, it is very vital to control and reduce welding distortion in practical engineering application.

To effectively control welding deformation, we must deeply understand the influences of various factors on the final deformation based on the experimental observations or through the predictions obtained by numerical method [2]. With the development of computer hardware and software, numerical simulation technology has been becoming a powerful tool to solve the complicated thermo-mechanical problems during welding process.

At present, there are two methods based on finite element method (FEM) which can be used to predict welding deformation. One is the thermal elastic plastic FEM [3], and the other is the elastic FEM based on the inherent strain theory [4]. When the former method is used, the temperature field, stress and strain fields and even the thermal-metallurgical-mechanical coupling behaviors can be calculated. However, there are some obvious disadvantages in this method. For example, a very long computing time is needed because welding process is coupled nonlinear problem. In addition, the temperature-dependent material properties even phase-dependent material properties should be measured by experiment. Also, heat source model and its parameters should be carefully selected. At present, a number of fast computational approaches such as ISM [5,6] and ideal explicit method [7] based on thermal elastic plastic FEM have been developed, and these methods perhaps are promising ways to solve the thermo-mechanical problems in large-scale welded structures. However, it is inevitable to spend a relatively long time for preparing the huge mesh data and defining the required information related to welding procedure.

The elastic FEM based on inherent strain method has been recognized as an effective approach to estimate welding deformation in large scale welded structure because of the short

* Corresponding author.
E-mail address: deandeng@hotmail.com (D. Deng).

http://dx.doi.org/10.1016/j.advengsoft.2016.05.003
0965-9978/

computing time and the relatively simple preparation of input data [8]. When this method is used to predict welding distortion in a welded structure, one prerequisite is that the inherent deformations of each joint in the whole structure should be known beforehand.

The basic components of inherent deformation in a weld include longitudinal shrinkage, transverse shrinkage, longitudinal bending, and transverse bending (angular distortion) [9]. The first two components are in-plane deformation, and the last two belong to out-of-plane deformation.

At present, three methods can be employed to obtain inherent deformation for a welded joint. The first one is experimental method [9]. Using this method, each component of inherent deformation can be directly obtained by measuring technology. The measuring accuracy strongly depends on the measuring tools and the magnitude of deformation occurred in the joint. Generally, transverse shrinkage and angular distortion can be easily measured because of their relatively large magnitude. However, it is difficult to accurately measure longitudinal shrinkage and longitudinal bending because their values are often small. The inherent deformation in a weld can also be calculated by integration method based on thermal elastic plastic FEM [10]. However, the calculation accuracy strongly depends on the analyst's knowledge and experiences because too many influential factors of welding deformation should be carefully taken into account in thermal elastic plastic finite element model. The third method is the inverse analysis method [11]. This method was proposed by Liang and Murakawa [12] recently, and it has been used to successfully establish the database of inherent deformation in several typical thin-plate mild steel joints.

For mild steel or stainless steel joints, a relatively small size can be used to obtain the inherent deformation because the thermal conductivity is relatively small and the moving temperature field can reach a quasi-steady-state in a short time. For Al-alloy joints, the thermal conductivity is much larger than that of steel, so the moving weld pool needs a longer time to reach the quasi-steady-state. In such situation, a relatively large size of welded joint is required to obtain accurate inherent deformation. Because the stiffness of thin-plate Al-alloy joint is relatively small, it can be foreseen that buckling distortion will be apt to occur especially when a large heat input is applied or the thickness of plate is small. If buckling distortion occurs in a joint, it will be difficult to accurately obtain the inherent deformation by inverse analysis method. To avoid this problem, an improved inverse analysis method with the help of cutting technique was proposed to obtain inherent deformation of thin-plate Al-alloy joint in the current study.

In this study, the distribution shape and feature of each inherent deformation component along welding line in a thin-plate Al-alloy joint were clarified based on the results of residual plastic strain (inherent strain) distribution simulated by thermal elastic plastic FEM. Meanwhile, the welding deformation of the Al-alloy thin-plate joint predicted by thermal elastic plastic FEM was verified by experiment. Then, according to the distribution features of inherent deformation, an improved method based on inverse analysis with the help of cutting technique was proposed to estimate inherent deformations in the Al-alloy thin-plate joint. Finally, the welding deformation of Al-alloy thin-plate joint was computed using the inherent deformations obtained by the new method, and the computed result was verified by the corresponding thermal elastic plastic FE model.

2. Welding deformation in Al-alloy thin-plate joint

In the current research, an Al-alloy thin-plate joint was welded by TIG welding process to investigate welding deformation. Meanwhile, welding deformation in the Al-alloy joint was simulated by thermal elastic plastic FEM.

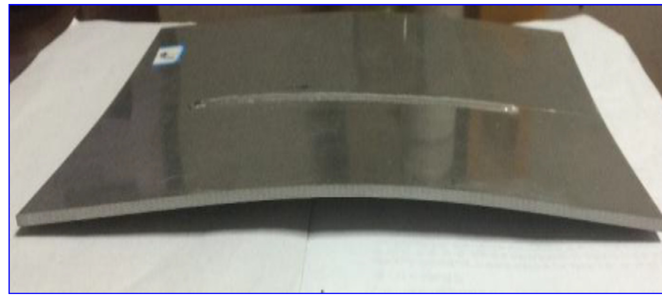

Fig. 1. Welding deformation of 5083 Al-alloy thin-plate joint.

2.1. Experimental procedure

Fig. 1 shows the Al-alloy thin-plate joint. The base material is 5083 Al-alloy. The length, width and thickness of this joint are 300 mm, 200 mm, and 3 mm, respectively. As shown in Fig. 1, a partial welding was performed in the plate, and the length of welding line is 200 mm. TIG welding process without filler was used to perform the welding. This means the TIG torch was used to melt the based metal and then formed a welding seam. The welding current, arc voltage, and welding speed are 200 A, 20 V and 15 mm/s respectively. The joint was welded without any external restraint during the entire process in experiment. After welding, Vernier caliper was used to measure the out-of-plane deformation.

2.2. Predicting welding deformation by thermal elastic plastic FEM

In this study, a thermal elastic plastic finite element method based on ABAQUS software was developed to simulate welding deformation in Al-alloy thin-plate joint. In the developed computational approach, both material and geometrical nonlinearities are taken into account.

Fig. 2 shows the finite element (FE) model and its boundary conditions. A finer mesh was used in fusion zone and its vicinity, and a coarser mesh was employed in the rest region. Using such mesh, the balance between computing time and prediction accuracy can be achieved. The element type in the thermal analysis is DC3D8, while that in the mechanical analysis is C3D8I [13].

Before mechanical analysis, transient welding heat transfer analysis with given welding conditions was performed on the FE model. The thermal cycle of each node during the entire welding process and the transient temperature distribution in the whole region of model are acquired by solving the non-linear heat transfer governing equation. In the mechanical analysis, the thermal cycle of each node was introduced into the finite element model as thermal loading to obtain displacement, strain and stress.

The heat source model is considered to be an important aspect of the welding thermal analysis. In the thermal analysis, the moving heat source proposed by Goldak [14] was employed to model the heat input. The heat input (Q) can be calculated according to the welding conditions and the assumed arc efficiency. The arc efficiency is assumed to be 0.7 [15]. The temperature dependent thermal physical properties shown in Fig. 3 [16] were used in the thermal analysis.

Besides considering the moving heat source, heat losses due to convention and radiation are also taken into account in the FE model.

The elastic strain-stress relationship was modeled using the isotropic Hooke's law, and plastic behavior was considered through Von Misses criterion. The effect of work hardening was neglected