Computational Materials Science 62 (2012) 23–34

Contents lists available at SciVerse ScienceDirect

Computational Materials Science

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

Numerical simulation of welding temperature field, residual stress and deformation induced by electro slag welding

Dean Deng [a,*], Shoichi Kiyoshima [b]

[a] College of Materials Science and Engineering, Chongqing University, Shazheng Street 174, Shapingba, Chongqing 400044, China
[b] Research Center of Computational Mechanics Inc., Togoshi NI-Bldg., 1-7-1 Togoshi, Shinagawa-ku, Tokyo 142-0041, Japan

ARTICLE INFO

Article history:
Received 3 March 2012
Received in revised form 5 April 2012
Accepted 15 April 2012
Available online 31 May 2012

Keywords:
Electro slag welding
Numerical simulation
Thermal cycle
Moving heat source

ABSTRACT

To increase productivity, welding process with large heat input such as electro slag welding (ESW) process has been used to connect the joints between the diaphragm and the column plate in high-rise steel building. However, the heat input of ESW is much higher than those of the other welding processes, and the high heat input not only largely alters the properties of steel but also results in large residual stresses. Consequently, the changes of steel properties and residual stresses induced by ESW have significantly effects on the safety of a structure. In this study, a three dimension (3-D) finite element model with considering moving heat source was developed to simulate the welding temperature field, $\Delta t_{8/5}$ time, welding residual stress and distortion in a typical thick plate joint performed by ESW. The thermal cycles computed by finite element model were compared with experimental measurements. Meanwhile, the features of welding residual stress and distortion distributions in the ESW joint were investigated numerically. In addition, the influences of heat input on the size of heat affected-zone (HAZ), $\Delta t_{8/5}$ time welding residual stress and distortion were examined. The thermal cycle curve and simulated by FEM model can be used to deduce the micro-structure as well as toughness of weld zone and HAZ, while the welding residual stress distribution estimated by numerical model can be helpful to assess the structural integrity.

1. Introduction

Beam-to-column joint is an essential structural component used in high-rise steel building. To achieve a high productivity, welding processes with large heat input such as electro slag welding (ESW) process and submerged arc welding (SAW) are widely used to joint beam-to-column connections. However, the heat input of ESW is far larger than those of other welding processes such as gas metal arc welding, shielded metal arc welding and flux-cored arc welding. The large heat input will significantly change the microstructure and mechanical properties especially the grain size and toughness. Generally, the ESW joint is close to the beam-to-column connections where both the stress and strain are large when receiving a dynamic load such as earthquake [1].

Since the Hanshin-Awaji Earthquake occurred in 1995, researchers have been paying attention to the seismic behavior of building steel frame increasing [2–4]. Meanwhile, current design standard requires higher toughness for beam-to-column than before. In Japan, the toughness presently required for this type of connection is no less than 70 J in terms of the Charpy absorbed energy at 0 °C [5–7]. In welding filed, a number of studies have been conducted to investigate the relationships between heat input, inter-pass temperature and Charpy absorbed energy in typical weld joints, and a large database has been establishing [1,8–10]. Recently, Kojima and his co-workers [11] proposed formulae to estimate HAZ toughness according to steel chemical compositions for electro slag welds of building box columns. In their research, they used a thermal simulator to duplicate thermal cycles of HAZ induced by ESW, and the $\Delta t_{8/5}$ time of HAZ was controlled within the range between 300 s and 600 s.

On the other hand, numerical simulation technology based on finite element method has been widely used to calculate thermal cycles, welding residual stress, welding deformation and even microstructure as well as hardness [4,12–14]. Within the scope of our knowledge, very limited models have been developed to simulate electro slag welding process. However, it is possible to combine the numerical simulation method and the database of Charpy absorbed energy in various type joints to directly or indirectly predict the toughness of a joint. The information will be very useful to assess structural integrity and to avoid brittle fracture occurring in weld joints.

In this study, we developed a computational approach based on Quick Welder [15] to compute welding temperature, $\Delta t_{8/5}$ time, residual stress and distortion in a typical ESW joint. Meanwhile, we compared the temperature cycles estimated by finite element

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

0927-0256/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.commatsci.2012.04.037

model and the measurements. In addition, we numerically studied the feature of welding residual stress distribution and welding deformation.

2. Temperature cycle measurement of ESW joint

Fig. 1 schematically shows ESW joints involved in a box column, and Fig. 2 shows the images of vertical section and cross-section in an ESW weldment [1]. From Fig. 1, we can know that a typical ESW joint consists of a diaphragm, a column plate (skin plate) and two backings. In this study, an experimental mock-up whose shape and dimensions are shown in Fig. 3 was manufactured to investigate temperature cycles during welding. The material of the mock-up is high tensile steel named SM490A, and the weld wire is YGW18. The chemical components of SM490A and its mechanical properties at room temperature are shown in Table 1.

The length of ESW joint is 600 mm, and the breadth of skin plate is 500 mm. The height of diaphragm is 300 mm, and that of backing plates is 50 mm. The thickness of the both skin plate and the diaphragm is 50 mm, and that of the backing plates is 30 mm. The gap between the diaphragm and the skin plate is 23 mm. Before welding, all parts were fixed together by tack welded. The locations of tack welds are shown in Fig. 4. The welding conditions are shown in Table 2. In this study, K-type thermocouples were used to measure the thermal cycles. To obtain the thermal cycles at the representative locations, the thermocouples were arranged in the skin plate as shown in Fig. 5a. In addition, two more thermocouples were arranged at the mid-section as shown in Fig. 5b.

After welding, we examined the macrostructure of the mid-section in the ESW joint. The shape of nugget is shown in Fig. 4.

3. Computational approach

Measuring temperature cycles of ESW welding by experiment is time-consuming and expensive. Moreover, experiment can only provide the thermal cycle and distribution near the surface, but it is difficult to obtain the temperature history and distribution inside the plate. However, numerical simulation method is not only economical but also can provide the temperature distribution for the whole welded joint. When a numerical method based on FEM is used to simulate temperature distribution and stress field, it should be verified by experiment. In this study, based on Quick Welder software [15], a computational approach was developed to calculate temperature field, residual stress distribution and distortion induced by ESW process. The temperature cycles obtained by the computational approach were verified by the experimental measurements.

3.1. Finite element model

The finite element model is shown in Fig. 6. Considering the symmetry of geometry, location of welding line and boundary

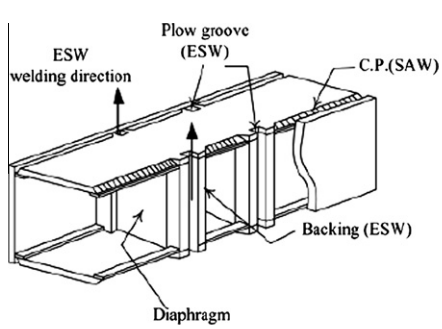

Fig. 1. Schematic representation of ESW in box column.

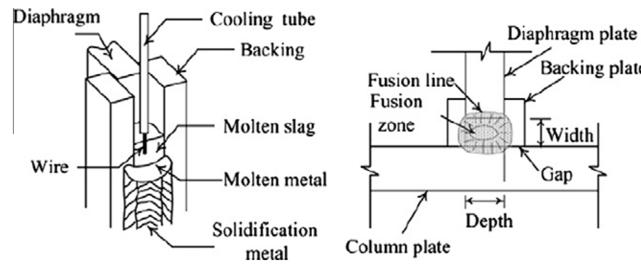

Fig. 2. Image of ESW process and cross-section of weldment.

conditions, only half of the ESW joint model was used in the present study. To reasonably compute the temperature field, the residual stress distribution and distortion induced by ESW process, a finer mesh was designed at the weld zone and its vicinity, while the mesh size gradually increased with the distance from the weld center. The number of division in the longitudinal direction is 30 as shown in Fig. 6. The number of nodes is 20,367; and that of elements is 22,200.

In this work, besides welding temperature field, the thermo-mechanical behavior was also analyzed using a sequentially coupled formulation. The heat conduction problem was solved independently from the stress–strain problem to obtain temperature histories. However, the formulation took into account the contributions of the transient temperature field to the mechanical analysis through thermal expansion coefficient, as well as temperature-dependent mechanical properties such as Young's modulus, yield strength and strain hardening coefficient. To accurately predict welding residual stress and distortion, an advanced material model with considering stain-hardening and annealing effect has been developed to simulate the transient thermo-mechanical behavior during ESW process.

3.2. Thermal analysis and heat source model

Transient nonlinear heat transfer analysis with given welding conditions was performed using a 3-D finite element model with moving heat source. In the thermal analysis, the governing equation for transient non-linear heat transfer analyses is:

$$\lambda_x \left(\frac{\partial^2 T}{\partial x^2}\right) + \lambda_y \left(\frac{\partial^2 T}{\partial y^2}\right) + \lambda_z \left(\frac{\partial^2 T}{\partial z^2}\right) + q_E = \rho c \frac{\partial T}{\partial t} \tag{1}$$

where $\lambda_i (i = x, y, z)$, T, ρ and c are thermal conductivity, temperature, density and specific heat capacity, respectively. It is assumed that both the base metal and the weld metal are isotropic material, so the conductivities in x, y and z directions have the same value, namely $\lambda_x = \lambda_y = \lambda_z$. q_E is the rate of internal heat generation.

As shown in Fig. 2, a consumable wire electrode is fed continuously into a molten slag pool, which is resistively heated by current passing from the electrode through the molten slag pool, the weld pool to the base metals. In the calculation of heat generation pattern, it can be assumed that all the voltage drop occurs across the slag layer, which is reasonable, because the electric resistance of slag phase is of the order of a few thousand times that of the metal [16]. According to previous research [17], in the case of low carbon steel, the temperature of the bath is reported to be 1925 °C, while the surface temperature is approximately 1650 °C. This information suggests that the temperature distribution within the bath of molten flux is relatively even. In addition, the temperature gradient in weld pool is not large. Therefore, according to the feature of temperature distribution in the bath of molten slag and weld pool, the moving heat source of ESW seems to be represented