Construction and Building Materials 154 (2017) 557–566

Contents lists available at ScienceDirect

Construction and Building Materials

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

Analysis and fracture behavior of welded box beam-to-column connections considering residual stresses

Jianrong Pan [a,b], Shizhe Chen [a,b,*], Zhichao Lai [c,b], Zhan Wang [a,b], Junqi Wang [d], Huashen Xie [a]

[a] *School of Civil Engineering and Transportation, South China University of Technology, Guangzhou 510640, China*
[b] *State Key Laboratory of Subtropical Building Science, South China University of Technology, Guangzhou 510640, China*
[c] *Lyles School of Civil Engineering, Purdue University, West Lafayette, IN 47906, USA*
[d] *Architecture Design and Research Institute, Guangzhou 510641, China*

HIGHLIGHTS

- Develops finite element models for simulating welding.
- Evaluates three heat source models.
- Conducts cyclic tests on welded box beam-to-column connections.
- Evaluates the effect of residual stresses on the fracture behavior of connections.

ARTICLE INFO

Article history:
Received 24 April 2016
Received in revised form 19 June 2017
Accepted 31 July 2017
Available online 8 August 2017

Keywords:
Welding
Residual stresses
Heat source models
Fracture
Finite element analysis
Box connections

ABSTRACT

This paper investigates the effects of residual stresses on the fracture behavior of welded box beam-to-column connections. Finite element analyses are first conducted to evaluate three commonly used heat source models for simulating welding. The selected models are the volumetric heat generation rate model, Gaussian model, and double ellipsoidal model. The evaluations indicate that the volumetric heat generation rate model has the best performance. Experimental tests on two welded box beam-to-column connections are then conducted to evaluate their fracture behavior when subjected to cyclic loading. The test results show that the governing failure mode is the fracture of the weld at the beam flange. Finally, additional finite element analyses are conducted to evaluate the effects of residual stresses on the fracture behavior of the two tested connections. The analyses indicate that welding residual stresses result in the increase of equivalent plastic strain at the welds, which will lead to earlier fracture of the connection.

1. Introduction

Steel structures are typically designed to have good ductility. However, brittle fracture of welded connections can occur in extreme events such as the Northridge earthquake in 1994 [1]. There are several reasons that result in the brittle fracture of welded connections, which include: (i) inadequate welding quality control [2,3], (ii) poor construction quality control [4], and (iii) significant welding residual stresses [5–7]. To address this, several methods such as reinforcing with cover plates have been proposed [8–10].

During welding, welds and adjacent base metals deform significantly. Their deformations are restrained by surrounding metals subjected to lower temperature. This results in the formation of residual stresses. Several researchers have analytically investigated the behavior of welded connections using different heat source models. For example, Pavelic et al. [11] proposed the Gaussian heat source model and implemented it in finite element (FEM) models to simulate welding. Zhang and Dong [12] developed detailed FEM models that can be used to simulate welded wide flange beam-to-column connections. In their study, the volumetric heat generation rate heat source model was used. Deng et al. [13] developed an elastic FEM model to predict welding distortion during the assembly of stiffeners and skin plate. In thermal analysis, they used combined Gaussian and double ellipsoidal heat source model. Liu et al. [14] used the Gaussian heat source model for simulating the welding of pipes. The pipe wall thickness was 70 mm, and 73 weld passes were used. Jia [15] developed FEM models to simulate the welding between wide flange beams and end plates, using the volumetric heat generation rate heat source model. Li and Wang

---

[*] Corresponding author at: School of Civil Engineering and Transportation, South China University of Technology, Guangzhou 510640, China.

E-mail address: ctszchen@scut.edu.cn (S. Chen).

http://dx.doi.org/10.1016/j.conbuildmat.2017.07.228
0950-0618/© 2017 Elsevier Ltd. All rights reserved.

[16] used double ellipsoidal heat source model in the simulation of single bead-on-plate specimens.

These prior studies provide valuable insights into the modeling of welding and obtaining resulting residual stresses. However, there is still lack of knowledge regarding (i) the performance of different heat source models, and (ii) the effect of residual stresses on the fracture behavior of welded connections. This paper contributes towards addressing the knowledge gap by conducting both finite element analyses and experimental tests. Finite element analyses were first conducted to evaluate three commonly used heat source models for simulating welding. The selected models are the volumetric heat generation rate model, Gaussian model [11], and double ellipsoidal model [17]. Experimental tests on two welded box beam-to-column connections were then conducted to evaluate their fracture behavior when subjected to cyclic loading. Finally, additional finite element analyses were conducted to evaluate the effects of residual stresses on the fracture behavior of the two tested connections.

2. FEM model

2.1. Prototype connection details

Fig. 1 shows the prototype T-connection used in the finite element analysis. This type of connection represents the welding between beam elements (i.e., flanges and webs) and column elements (i.e., flanges and webs) in a box beam-to-column connection. As shown, the T-connection consists of 450 × 400 × 14 mm base plate, 250 × 250 × 14 mm welded plate, and full penetration welds (see Fig. 2). These dimensions were determined based on test specimens reported later in Section 4.

Gas metal arc welding (GMAW) was used along with ER50-6 welding wire (diameter = 12 mm). The welding voltage varied from 21 to 24 V, the welding current varied from 210 to 270 A, and the welding speed varied from 23 to 28 cm/min. In the FEM models used in this study, the weld bead was simplified into three layers (as shown in Fig. 3), i.e., backing bead (4 mm thick), filling bead (6 mm thick), and cosmetic bead (4 mm thick). The effect of welding sequences was not evaluated in this study but is recommended for future research.

Q345 steel as per GB/T 1591-2008 [18] was used for both the base plate and welded plate. Table 1 summarizes the steel mechanical and thermal properties. The steel mechanical properties at ambient temperature were measured as per GB50017-2014 [19], which include the elastic modulus (E_s), tangent modulus (E_t), and yield stress (σ_s). The steel mechanical properties at elevated temperature and other thermal properties of the steel were determined based on [20–22]. The latent heat during phase change was not considered because including it in the analysis resulted in convergence issue. Further research regarding this is recommended.

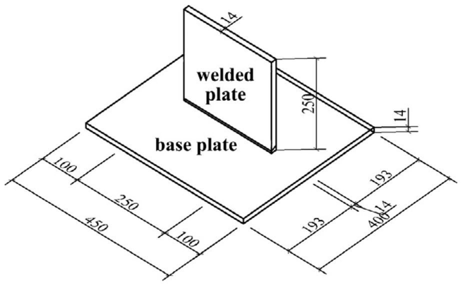

Fig. 1. Prototype T-connection (unit: mm).

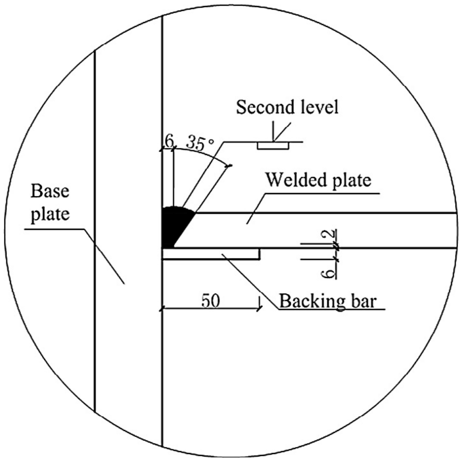

Fig. 2. Weld size (unit: mm).

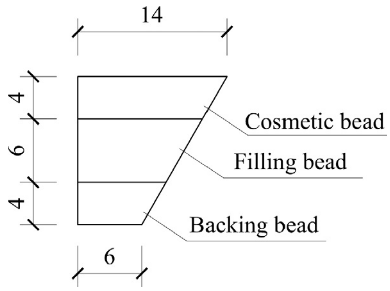

Fig. 3. Simplified weld model (unit: mm).

2.2. FEM model details

Detailed three-dimensional finite element (FEM) models were developed for the T-connection using ANSYS [23]. As shown in Fig. 4, the models consisted of five regions, i.e., base plate, welded plate, weld, and two transition regions between: (i) weld and base plate and (ii) weld and welded plate. The backing bar was not modeled for simplicity. In the thermal analysis, the first three regions were modeled using eight-node solid elements Solid70, and the transition regions were modeled using higher order 20-node solid elements Solid90. While in the stress analysis, the first three regions were modeled using eight-node solid elements Solid185, and the transition regions were modeled using higher order 20-node solid elements Solid186. Elements Solid90 and Solid186 are suitable for modeling irregular meshes in the transition region. Mesh sensitivity and convergence were conducted to finalize the size and distribution of finite elements. The selected mesh size (as shown in Fig. 4) produced reasonable results without excessive computational effort. The finalized model has 31,077 elements and 41,247 nodes in total.

The steel material multiaxial stress-strain behavior was defined using Von Mises yield surface, associated flow rule, and kinematic hardening. An idealized trilinear curve as shown in Fig. 5 was used to specify the temperature-dependent uniaxial stress-strain behavior of the steel. This curve can be completely defined by specifying the following four parameters: (i) elastic modulus (E_s), (ii) yield stress (F_y), (iii) tangent modulus (E_t), and (iv) yield strength (F_u). These four parameters were defined using the values listed in Table 1.

The finite element analysis consisted of two steps. In the first step, thermal analysis was conducted to simulate the welding process and obtain corresponding temperature distributions. In the