Composite Structures 143 (2016) 180–188

Contents lists available at ScienceDirect

Composite Structures

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

Modeling of composite beam–column flexible endplate joints at elevated temperature

Khalifa S. Al-Jabri [a], Prashob Pillay [a], Muhammad B. Waris [a,*], Tasneem Pervez [b]

[a] Department of Civil and Architectural Engineering, Sultan Qaboos University, Oman
[b] Department of Mechanical and Industrial Engineering, Sultan Qaboos University, Oman

ARTICLE INFO

Article history:
Available online 13 February 2016

Keywords:
FEM
Steel–concrete composites
Joints
Fire

ABSTRACT

The paper presents a numerical study on two composite cruciform beam–column joints with flexible endplates subjected to elevated temperature. A two 356 × 171 × 51 UB connected to a 254 × 254 × 89 UC labeled Joint-A and two 610 × 229 × 101 UB connected to 305 × 305 × 137 UC labeled Joint-B. Numerical experiments were performed to estimate moment capacity of the joints at ambient temperature. Four fire tests for Joint-A and two for Joint-B were investigated with applied moment varying between 30%–70% of the joint capacity. Heating was applied linearly at a rate of 10 °C/min. Utilizing biaxial symmetry, only one fourth of joint configuration was modeled in ABAQUS. The bare steel and concrete slab components of the joints were modeled using a four-noded tetrahedral element capable of coupled thermal displacement analysis. EC3 recommendations for degradation in strength and stiffness were employed to model steel behavior. The concrete slab was modeled using damaged plasticity. Based on temperature–rotation relationship, the predicted results showed good agreement with experiment data in the elastic range while slight overestimates were observed in the plastic region. The results indicated high stress concentration in the top end of the endplate, top bolts and lower portion of the beam web close to bottom of the endplate. It was also observed that base of the embedded shear studs were heavily stressed. The failure modes of both joints in all loading cases were well predicted in simulation.

© 2016 Elsevier Ltd. All rights reserved.

1. Introduction

Joints are essential part of any structure and their moment capacity, flexural rigidity and deformation characteristics has significant effect on the overall behavior of such systems. In a fire event, the behavior of joints is therefore of great importance as it may govern the behavior of entire structure. With an increasing trend of composite construction, there is a need to investigate the behavior of joints in such systems. Experiments can provide a reliable insight into the behavior of composite beam-to-column joints. However, difficulty of execution and the expensive nature of such endeavors limit their number in terms of geometrical and mechanical parameters. Experiments on joints under fire have revealed that joints fail in the tension zone, through bolts or endplates due to high cooling strains induced by distortional deformation of connected members [1].

With progression in computational capability and availability of reliable modeling tools, complex boundary value problems can be handled. Use of finite element (FE) modeling to investigate the joint behavior at elevated temperature has attracted interest in recent years. Accurate modeling as well as realistic simulation of degradation of joint characteristics with increase in temperature is required in order to achieve acceptable results. Factors affecting the accuracy of FE modeling include the meshing of the configuration (the optimum mesh size), simulation of bolts, choice of elements, modeling of concrete slab and shear studs, material behavior and most importantly modeling of the contact and gap elements [2].

Several studies have been carried out to model the load deflection behavior of beams column joints [3–11] indicating that numerical modeling can provide realistic and accurate prediction of joint behavior and capacity. These studies showed that both bare steel and composite connections can be properly modeled to provide realistic results.

After success in modeling of joints' behavior using the finite element scheme, attempts started to model behavior of steel

* Corresponding author.
E-mail addresses: aljabri@squ.edu.om (K.S. Al-Jabri), m091846@squ.edu.om (P. Pillay), waris@squ.edu.om (M.B. Waris), tasneem@squ.edu.om (T. Pervez).

http://dx.doi.org/10.1016/j.compstruct.2016.01.069
0263-8223/© 2016 Elsevier Ltd. All rights reserved.

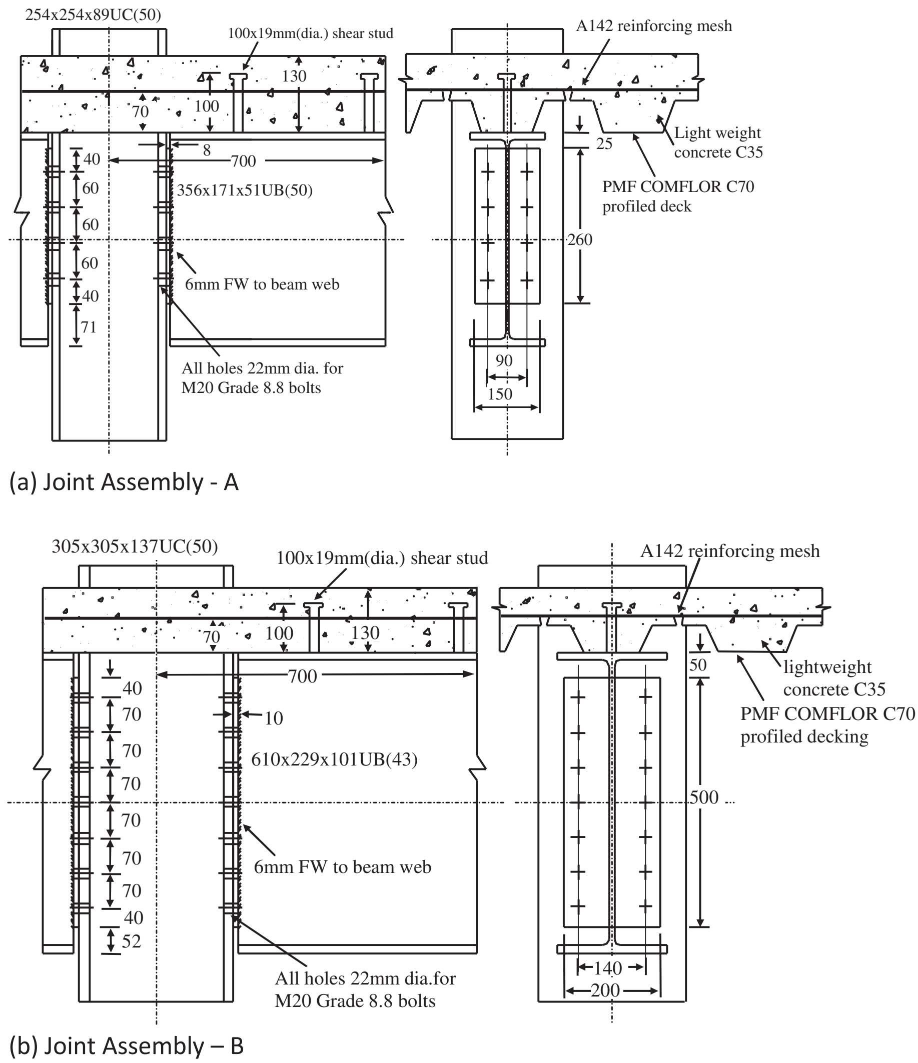

Fig. 1. Details of the composite cruciform joint assemblies [1].

connections under fire. An early attempt on modeling the steel connection under fire was made by Liu [12,13]. He developed a finite element program FEAST to simulate various types of bare steel connections in fire. A close agreement was observed with experimental data. The discrepancies between the results were suggested to be partly attributed to limited data of temperature distribution, actual stress–strain relationships and difference between adopted material properties. El Hossieny et al. [14] developed a three-dimensional finite-element model to simulate the response of extended endplate at both ambient and elevated temperatures. Close agreement was obtained with experimental work and subsequent parametric studies were conducted to investigate the influence of connection behavior at normal and elevated temperature. Spyrou et al. [15] modeled T-stub specimens at elevated temperature using ANSYS. Their results demonstrated that three-dimensional analysis was more accurate than two-dimensional modeling. Rahman et al. [16] used ANSYS to study the behavior of fin-plate connections in fire using a transient time-temperature fire load. Despite realistic results being predicted by the model, no experimental data was used to validate the model. Al-Jabri et al. [17,18] and Shirh et al. [19] performed detailed simulations on a series of fire tests on flush endplate connections using ABAQUS and ANSYS. Predictions agreed well with the experimental results [1,20], however, there were limitations in the research as only fixed bending moments in fire were considered and no simulation on joint-beam assemblies were considered [19]. Sarraj et al. [21] modeled steel fin plate connection at elevated temperature using ABAQUS. The results were validated with the experimental data available and results were found to be in close agreement with the experiments. Daryan et al. [22] used ANSYS to model bolted angle connections at elevated temperature. Yu et al. [23] investigated the behavior of bolted steel connections in fire with ABAQUS using explicit dynamic analysis. Convergence that was always a problem in static analysis was resolved in explicit dynamic analysis. Results were compared with static analysis and test results matched very well. Several other researchers

Table 1
Experimental test scheme [1].

Test case	Joint-A		Joint-B	
	Label	Applied moment [kN-m]	Label	Applied moment [kN-m]
Ambient (0)	–	Till failure	–	Till failure
1	A-01	34.0	B-01	80.0
2	A-02	46.0	B-02	134.0
3	A-03	62.0		
4	A-04	82.0		

Download English Version:

https://daneshyari.com/en/article/6705767

Download Persian Version:

https://daneshyari.com/article/6705767

Daneshyari.com