Journal of Constructional Steel Research 67 (2011) 75–83

Contents lists available at ScienceDirect

Journal of Constructional Steel Research

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

Parametric analysis of mechanical behavior of steel planar tubular truss under fire

Meng Jin[a,*], Jincheng Zhao[a], Minglu Liu[a], Jing Chang[b]

[a] *Department of Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China*
[b] *Shandong Jiao Tong University, Jinan 250023, China*

ARTICLE INFO

Article history:
Received 8 March 2010
Accepted 24 July 2010

Keywords:
Fire test
Steel planar tubular truss
Critical temperature
Parametric analysis

ABSTRACT

This paper presents experimental and numerical results of steel planar tubular truss, which was exposed to the fire condition. The principal aim is to investigate the critical temperature of the truss under permanent load and the influence of geometric parameter. It is found that both experimental and numerical results demonstrate that the failure of the steel planar tubular trusses is due to the local buckling of the brace members which validates the accuracy of finite element model. In the parametric analysis, it is found that the geometric parameters of the truss have a significant influence on the load bearing capacity of steel planar tubular truss subjected to the fire. The parameters include wall thickness ratio τ, diameter ratio β and chord diameter/thickness ratio γ. The results demonstrate that the critical temperature of the truss can be improved significantly by the increase of the brace diameter and the wall thickness of the chord while changing the wall thickness of the brace has limited effects. It is indicated from the analysis that at the higher load level, the parameters τ, β and γ have a distinct influence on critical temperature of the steel planar tubular truss.

1. Introduction

In recent decades, steel tubular structures are widely used in space structures and offshore platforms for their simplicity, light weight, good mechanical performance and rapid erection. So far, the mechanical characteristic of the steel tubular structures have been extensively studied in recent years, such as tubular T-joints, tubular K-joints, etc. A detailed parametric analysis on the static strength, stress and a method to predict the strength of axially loaded tubular T-joints with internal ring-stiffeners were studied by Lee [1–3]. The experimental results of a full-scale completely overlapped joint tested to failure under lap brace axial compression were presented by Fung [4,5], and it is found that joint strength was increased as the gap was reduced by numerical simulation. Soh [6,7] investigated the behavior of completely overlapped tubular joints used in the eccentrically braced offshore jackets. Two full-scale specimens were tested, in which one for static strength with monotonic compressive loading and the other for seismic behavior under cyclic quasi-static loading. And the finite element analysis showed that the completely overlapped joint performed better than N-joints under seismic loading. Hoon [8] tested to study stress behavior of tubular joints reinforced with a double-plate under the combined action of 3 types of basic load. Chiew [9] carried out fatigue tests on three tubular T-joints subjected to different loads, which had the same geometrical parameters. Shu [10] conducted a nonlinear finite element analysis of multi-planar circular tubular KT-joints. Three analytical models of circular tubular X-joints based on the breakage mode were summarized by Lin [11]. Static tests on two multi-planar tubular KTT overlapped joints were carried out by Tong [12]. Based on the recovery characteristic model of tubular joints, proper numerical methodology were modified by Wang [13] to predict the responses under severe earthquake excitations. Wang and Chen [14] described the Quasi-static experimental study on the response of eight T-joint specimens at ambient temperature, and the results indicate that the strength at a certain deformation limit can be regarded as the ultimate strength of a T-joint under cyclic loading, and high triaxial tensile stress was main reason to result in the occurrence of weld cracking under low cyclic load level. However, most of the above studies of the steel tubular joints and structures were mainly concentrated on the mechanical behavior at ambient temperature. During a fire, high temperature affects the performance of structures since the structural steel is the heat-sensitive material. In recent decades, a significant amount of research works have been conducted to assess the behavior of steel structures subjected to fire. However, most of the research was limited to the conventional components or structures such as steel beams, steel columns, steel beam to column joints and steel frames. Less works have focused on the performance of fire resistance of the steel tubular structures.

* Corresponding author. Tel.: +86 21 34207998; fax: +86 21 62933082.
E-mail address: 54jinmeng@163.com (M. Jin).

0143-974X/$ – see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.jcsr.2010.07.012

Notations

D_R	Outer diameter of the right chord
D_L	Outer diameter of the left chord
d	Outer diameter of brace
T_R	Wall thickness of the right chord
T_L	Wall thickness of the left chord
t	Wall thickness of the left chord
β	Diameter ratio (d/D_R)
τ	Wall thickness ratio (t/T_R)
γ	Chord diameter/thickness ratio $(D_R/2T_R)$
F_u	Load bearing capacity of truss at 20 °C

In this paper, an experimental investigation is carried out to obtain the behavior of the steel planar tubular truss under fire. Details of the test works were presented by Liu and Zhao [15]. The setup of the experiments and the analysis adopting the finite element model to simulate the test are simply presented. This paper presents the main observations of fire resistance behavior of steel planar tubular truss with various parameters. In the numerical analysis, the finite element analysis is firstly compared with the test result at elevated temperature. Then the validated model is applied in subsequent parametric investigations to study the effects of three key factors (i.e., wall thickness ratio τ, diameter ratio β and chord diameter/thickness ratio γ) on the critical temperature of the trusses under a constant load.

2. Experimental study

2.1. Specimens and test setup

As illustrated in Fig. 1, a steel planar tubular truss specimen is made up of a left chord, a right chord, two horizontal braces and two diagonal braces. Geometric size of the members is shown in Table 1. The members of truss are made of Q345B steel which is commonly used in Chinese construction industry. Through material tensile tests, the material properties of Q345B steel are determined, as shown in Table 2, which is the basic of numerical study. The whole appearance of the fire test setup is shown in Fig. 2.

2.2. Experimental results

The tests on SP1 and SP2 aim to study the failure mechanisms and fire resistance of the steel planar tubular truss under different constant load 400 kN and 600 kN, respectively. The axial load is kept constant during the whole heating. The relationship of displacement versus specimen temperature of SP1 and SP2 during fire is shown in Fig. 3.

3. Numerical study

3.1. Finite element model

In order to interpret the test results and obtain a better understanding of the behavior of steel planar tubular truss subjected to fire, a numerical simulation on tubular truss specimens is carried out using the finite element analysis program ABAQUS. Regarding the finite element model, a four-node doubly curved shell element with six degrees of freedom per node (S4R) is adopted. Five Gauss integration points are used along the direction of the shell element thickness. The dimensions of the finite element model are kept consistent with the test specimen. In order to get more accurate numerical results, mesh size is refined at the joint intersection area. The finite element model and refined mesh of the CHS (circular hollow section) joints are shown in Fig. 4. From the literature [1], the residual stresses resulted from the welding were neglected, because its effect on the static strength of the truss has been found to be insignificant. In this paper, the stress–strain relationship of steel at elevated temperature adopted by Eurocode 3 [16], as shown in Fig. 5, and the effect of temperature on the material are taken into consideration.

The boundary conditions for the finite element model of the truss are the same as that of the test specimen. The upper end of the right chord is fixed and the bottom end is constrained in horizontal

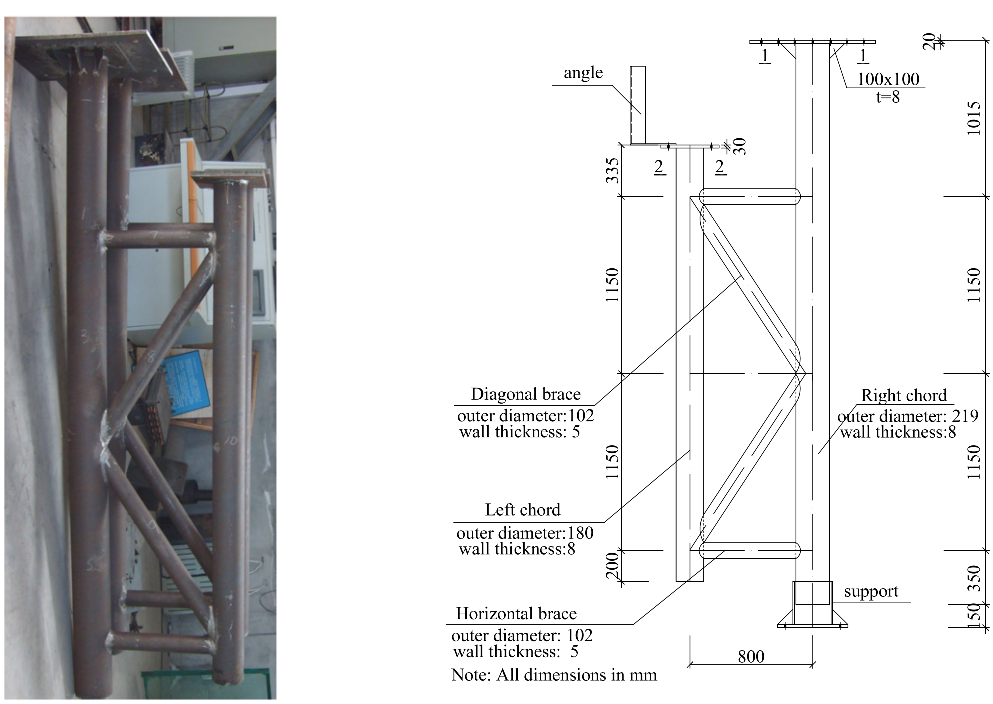

Fig. 1. Details of the test specimen.