Engineering Structures 88 (2015) 225–240

Contents lists available at ScienceDirect

Engineering Structures

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

Effects of truss behaviour on critical temperatures of welded steel tubular truss members exposed to uniform fire

E. Ozyurt, Y.C. Wang *

School of Mechanical, Aerospace and Civil Engineering, University of Manchester, UK

ARTICLE INFO

Article history:
Received 8 March 2014
Revised 10 October 2014
Accepted 19 January 2015

Keywords:
Circular Hollow Section (CHS)
Finite element (FE)
Critical temperature
Truss member limiting temperatures

ABSTRACT

This paper presents the results of a numerical investigation into the behaviour of welded steel tubular truss at elevated temperatures. The purpose is to assess whether the current method of calculating truss member limiting temperature, based on considering each individual truss member and using the member force from ambient temperature analysis, is suitable. Finite Element (FE) simulations were carried out for Circular Hollow Section (CHS) trusses using the commercial Finite Element software ABAQUS v6.10-1. The FE simulation model had been validated against available fire test results on trusses. The simulated trusses were subjected to constant mechanical loads and then increasing temperatures until failure. The elevated temperature stress–strain curves were based on Eurocode EN-1993-1-2. Initial geometrical imperfections were included, based on the lowest buckling mode from eigenvalue analysis.

The numerical parametric study examined the effects of truss type, joint type, truss span-to-depth ratio, critical member slenderness, applied load ratio, number of brace members, initial imperfection and thermal elongation on critical temperatures of the critical truss members.

These critical temperatures were then compared with the member-based critical temperatures, which were numerically calculated using ABAQUS but using the member forces obtained from ambient temperature structural analysis as would be the case in the current design method.

The results of the numerical parametric study indicate that due to truss undergoing large displacements at elevated temperatures, some truss members (compression brace members near the truss centre) experience large increases in member forces. Therefore, when calculating the member critical temperatures, it would not be safe to use the member forces from the ambient temperature structural analysis. Using the ambient temperature member force may overestimate the truss member critical temperature (based on truss analysis) by 100 °C.

Finally, this paper proposes and validates an analytical method to take into consideration the additional compression force due to large truss displacement. This is based on assuming a maximum truss displacement of span over 30.

1. Introduction

Hollow structural sections of all types are widely used in truss construction due to their attractive appearance, light weight and structural advantages. They are commonly used in onshore and offshore structures e.g. bridges, towers, stadiums, airports, railway stations, offshore platforms etc. For these structures, fire presents one of the most severe design conditions, because the mechanical properties of the steel degrade as the temperature increases.

For truss design, both the members and the joints should be checked. Truss member design at ambient temperature is relatively easy, involving mainly design checks for tension and compression resistance after performing static analysis to obtain the member forces. There is abundant amount of literature on the behaviour and strength and of truss joints at ambient temperature. Indeed, the CIDECT design guide [3] and Eurocode EN 1993-1-8 [4] present design equations to calculate the ambient temperature static strength of practically all tubular truss joints. In comparison, there are only a few research studies on welded joints at elevated temperatures. Among them, Nguyen et al. [5,6] carried out both experimental and numerical analysis on the behaviour of five full scale Circular Hollow Section (CHS) T-joints subjected to axial compression in the brace member at different temperatures. Meng et al. [7] and Liu et al. [8] presented a limited amount of experimental and parametric data of the structural behaviour of

---

* Corresponding author.
E-mail address: yong.wang@manchester.ac.uk (Y.C. Wang).

http://dx.doi.org/10.1016/j.engstruct.2015.01.032
0141-0296/

steel planar tubular trusses subjected to fire. Chen and Zhang [9] tested and numerically modelled a steel roof truss without fireproof coating under localised pool fire condition to obtain information on temperature distributions and displacements. However, no information was provided on failure mode or failure temperatures of the truss. Yu et al. [10] examined the mechanical behaviour of a steel T-joint under fire after impact loading. They observed increased failure temperature in the subsequent fire test compared to the fire test without impact. They attributed this increase to increased steel mechanical properties after high strain impact load. Jin et al. [11] experimentally investigated the parameters that affect post-fire behaviour of tubular T-joints. Based on their test results, they found that the load ratio in the brace member had no effect on the residual bearing capacity of the T-joint during both heating and cooling. Cheng et al. [12] observed that the critical mode of joint failure was plastification of the chord face for CHS T-joints at elevated temperatures with the brace member in compression. He et al. [13] carried out experimental tests to investigate fire resistance and failure mode of two full scale CHS K-joints under brace axial loading at brace ends. It was noted that the final failure mode of the two tests was due to local plastic yielding on the chord surface at brace-to-chord intersection area. The authors [14] recently developed and validated a design method for calculating the static strength of welded truss joints at elevated temperatures.

Under fire condition, the current method for truss member design involves calculating the member force using static analysis at ambient temperature and then finding the critical temperature, defined as the maximum temperature at which the member can resist the applied load, using the ambient temperature member force. The member force – critical temperature relationship can be evaluated using design methods such as those in BS 5950 Part 8 [15] and EN 1993-1-2 [2]. However, the member force obtained from truss static analysis at ambient temperature may not be correct at elevated temperatures due to large deformations of the truss. The review in the previous section indicates that there has been little study to investigate how truss member forces change at elevated temperatures and how such changes affect the member critical temperatures. These are the topics of the present paper.

The specific scope of this paper is to investigate whether the member-based fire resistance design approach is safe, and if not, to develop a modified member-based method to take into consideration truss behaviour.

2. Validation of finite element model

The results of this paper are based on numerical simulations using the general finite element package ABAQUS/Standard v6.10-1 [1]. For validation, the fire tests of Edwards [16] and Liu et al. [8] were simulated and compared with the test results.

Figs. 1 and 2 show the tested trusses. Failure modes and displacement-temperature curves were compared.

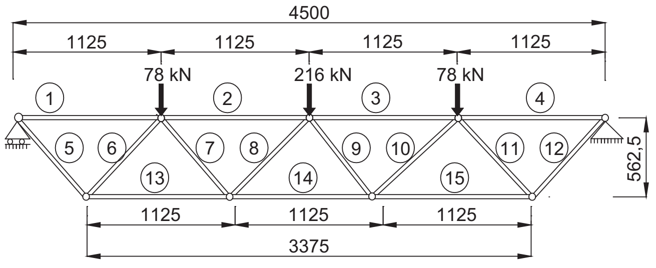

Fig. 1. Test Girder B of Edwards [16] (dimensions in mm).

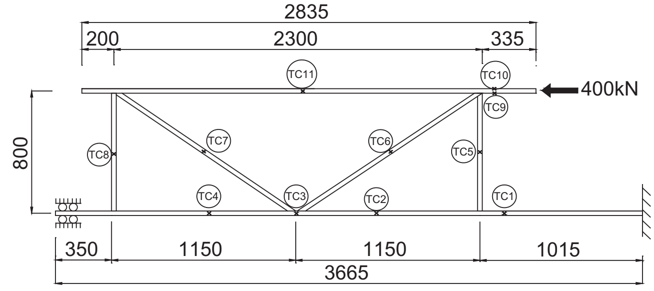

Fig. 2. Test specimen SP1 of Liu et al. [8] (dimensions in mm).

2.1. Material properties

Table 1 summarises the member sizes and material grades. For both tests, the ambient temperature mechanical properties were based on their coupon results. Extensive temperature measurements of the truss members were made in both tests and the recorded temperatures were used in the numerical analysis.

The elevated temperature engineering stress–strain curves were based on Eurocode EN-1993-1-2 as shown in Fig. 3 [2]. In the ABAQUS simulation model, the engineering stress–strain curve was converted into a true stress and logarithmic strain curve to consider nonlinear effects of large displacements by using the following equations [17]:

$$\varepsilon_T = \ln(1+\varepsilon) \tag{1}$$

$$\sigma_T = \sigma.(1+\varepsilon) \tag{2}$$

where ε_T, is the true strain, ε, is the engineering strain, σ_T, is the true stress and σ, is the engineering stress.

2.2. Finite element type and initial imperfection

For the chord and brace members, ABAQUS element types S4R (4 noded shell element) or B21 (2 noded line element) may be used. In the case of modelling using shell elements, quadratic wedge solid elements (C3D15) instead of shell elements were used for the weld to allow accurate meshing of the weld geometry [18]. At the weld-tubular section interface, the brace and chord members were tied with the weld elements using the ABAQUS "tie" function with surface to surface contact. The brace and chord members were chosen as the master surface and the weld elements the slave surface. Owing to symmetry in loading and geometry, to reduce computational time, only half of the truss was modelled when using shell elements, with the boundary conditions for symmetry being applied to the nodes in the various planes of symmetry as shown in Fig. 4.

Eigenvalue buckling analysis was performed on the numerical models in order to define the possible buckling modes for compressed members in the trusses. Lanczos was chosen as eigensolver together with the request five buckling modes [1]. Initial imperfections were included, based on the lowest buckling mode from eigenvalue analysis. The maximum initial imperfection was according to EN 1993-1-1 [19].

2.3. Comparison with test results

2.3.1. Test SP1 of Liu et al. [8]

Fig. 5 shows the recorded test temperatures (see Fig. 2 for locations of the thermocouples). These temperature curves were used in the numerical simulation.

The SP1 truss failed due to buckling of the diagonal brace member in compression. Fig. 6 compares the observed failure mode of