Analysis of Concrete-encased CFST Beams Under Pure Bending After Exposure to Standard Fire

Wen-dong DUN, De-bin REN and Qing-xin REN*
School of Civil Engineering, Shenyang Jianzhu University
Shenyang 110168, China
*E-mail: renqingxin@sjzu.edu.cn

This paper presents the bending behavior of concrete-encased concrete filled steel tube (CFST) beams after exposure to ISO-834 standard fire. A three-dimensional finite element analysis (FEA) modeling is developed for sequentially coupled heat transfer and pure bending by ABAQUS. Two groups (8 specimens) of post-fire concrete-encased CFST beams with different CFST ratio or different time under the standard fire were experimentally investigated and a set of test data were used to verify the FEA modeling. Finally, the predicted curves of moment versus deflection are in good agreement with the test data.

Key words: Concrete-Encased CFST Beam; Finite Element Analysis (FEA); Post-fire; Pure Bending.

1. Introduction

The concrete-encased concrete filled steel tube (CFST) beam is a composite member. The steel tube is filled of concrete and its outside is encased by concrete too, the cross-section is shown in Figure 1. In this composite section, the outer concrete can act as sacrificial layers to protect the steel tube and internal part under fire. And the steel tube also increase the bending capacity of the core concrete under bending moment. However, there are few analyses on the behavior of the composite member after exposure to fire.

![Figure 1. Cross-section of concrete-encased CFST beams.](image-url)
2. **FEA Modeling**

2.1. **Temperature field**

The FEA modeling is built in the commercial FE program ABAQUS software. The beams are heated 30min or 60min under ISO-834 standard fire. The thermal conductivity, specific heat, thermal expansion and other thermal properties of steel and concrete presented by Han (2015) [1] were adopted in this paper. The temperature of an element in cross-section was assumed to be equal to the temperature at its center. The 4-node heat transfer quadrilateral shell elements were used for steel tube; the 8-node linear heat transfer brick elements were used for shaped concrete; the 2-node heat transfer link elements were used for longitudinal bar and stirrup. The beam exposure 4 faces to fire, the thermal boundary conditions are shown in Figure 2.

![Figure 2. FEA modeling in fire.](image)

2.2. **Material properties**

The basic parameters of beams: $L=1800$mm (the effective length of beam), $L_1=900$mm (the pure bending length), $B=200$mm (the width of cross-section). In order to verify the validity of modeling and analysis, 8 beams in two different groups with different time at exposure to fire or with different CFST ratio were designed for fire and bending tests.

An equivalent stress–strain model presented by Han et al. [1] and the residual compressive strength of concrete heated to a maximum temperature $T$ and having cooled down to the ambient temperature of 20 °C may be found
Fracture energy versus displacement cross crack relation is used to describe the tensile behavior of concrete.

A typical stress–strain curve for steel is given in Han et al. (2002)[3]. Elastic modulus ($E_s$) and Poisson’s ratio ($\gamma$) for steel after exposure to fire are taken as $2.06 \times 10^5$ (N/mm$^2$) and 0.283, respectively.

2.3. Finite element model

A finite element model was developed for the analysis of the bending behavior of the concrete-encased CFST beams after exposure to ISO-834 fire standard, as shown in Figure 3. In the stress analysis model, the same element meshing as the thermal analysis model was adopted (Tao et al.) [4]. The steel tube is modeled by reduced-integration shell element (S4R), the core concrete, outer concrete as well as the plates, is modeled by 8-node brick elements (C3D8R). The end plates contact with the steel tube by “Shell-to-solid coupling”, the penalty friction formulation method (Hard contact) is used to the interaction between steel tube and concrete, include outer and core concrete; and the pad plates are combined to outer concrete by “Surface to surface contact”(Hard contact). The uniform line load in the Y direction is applied on the middle of two top rigid pad plates.

3. Verification of FEA Modeling

3.1. Temperature field in fire

The fire time is set to 60 minutes. The measured temperature results got from Point 1, Point 2(Figure 1.) and furnace is compared to the predicted data in Figure 4. It can be found the predicted temperature curves are closed to the measured curves.
3.2. Mechanical behaviour after exposure to fire

The predicted results obtained by ABAQUS were compared to the test results as shown in Figure 5 and Figure 6. It can be seen that the predicted results agree well with the test results.
4. Conclusions

The results of the study are summarized below:

(1) FEA modeling which can predict the temperature and bending capacity for the post-fire CFST beams was developed.

(2) The bending test of post-fire CFST beams were carried out, the test results were used to validate the FEA modeling, include temperature and post-fire bending capacity.

5. Acknowledgments

The research reported in the paper is part of the Project supported by the National Natural Science Foundation of China (NO. 51208135), the Program for Liaoning Excellent Talents in University (No. LR2015054) and the Project sponsored by “Liaoning BaiQianWan Talents Program” (No. 201559). The financial supports are highly appreciated.

References


3. Linhai Han, Jing-Si Huoa, Yong-Chang Wang, Compressive and flexural behaviour of concrete filled steel tubes after exposure to standard fire (Journal of Constructional Steel Research, 2005) pp. 882–901.