6th National Congress on Civil Engineering,
April 26-27, 2011, Semnan University, Semnan, Iran

Modeling of welded angle connections in fire

Amir Saedi Daryan [1], Mahmoud Yahyai[2],

1- PhD. candidate of Structural Session, Civil Engineering Dept. K.N.Toosi University, Tehran, Iran
2- Associate Professor, Civil Engineering Dept. K.N.Toosi University, Tehran, Iran

Amir_saedi_d@yahoo.com

Abstract

In this paper, the behavior of welded angle connections is studied at elevated temperatures using ABAQUS finite element software. In this study, steel members and connection components are considered to behave nonlinearly; the degradation of steel properties with increasing temperature is considered according to EC3 recommendations. The results of finite element and experimental tests conducted on welded angle connections in furnace fire conditions are compared, and the obtained failure modes and moment-rotation-temperature characteristics are in good agreement with those associated with experimental tests. Temperature loading similar to real fire condition is applied to each specimen to study the behavior of these connections in fire condition and the stiffness change with temperature increase under real fire condition is studied. The numerical results show that, in addition to material properties and connection geometry, the applied moments and temperature have significant effects on the stiffness of this kind of steel connection.

Keywords: Welded angle connections, Finite element modeling; Fire modeling; Elevated temperature

1 INTRODUCTION

The temperature sensitivity of steel is a weakness in steel structures. Since the mechanical properties of steel significantly deteriorate at high temperatures, the load capacity of steel structures under condition of a structural fire will decrease intensively. The failure of beam-column joints frequently induces the collapse of steel structures. However, the behavior of steel beam-column connections at elevated temperatures is very complicated and has not been fully studied. Different methods, including finite element analysis, are used to study the behavior of steel beam-column connections at elevated temperatures. The finite element method (FEM) provides an attractive means to investigate the beam-column joints in more detail than experimental tests would usually allow. A number of 3-D FE models have been developed (by various authors) that take into account any geometrical and material nonlinearities. Several studies have focused on the influence of elevated temperatures on end-plate connection behavior; a brief description of some of this research follows.

Liu [1, 2] was the first to attempt to use FEM in modeling connection behavior at elevated temperatures. He developed a finite element model (FEAST) to predict the behavior of different types of joints at elevated temperatures. The beam, column, end plate and stiffeners were modeled using eight-nodded shell elements and considered the nonlinear behavior of the material with nonuniform thermal expansion across a section as well as large deformations in fire. The stress-strain-temperature characteristics were adopted based on recommended values from experimental tests. Close agreement was observed with experimental data for different types of joints. A 3-D FEM was developed by El-Houssieny et al. [3] to simulate the response of extended end plates at both ambient and elevated temperatures. Results from the developed model compared well with the experimental results and subsequent parametric studies were conducted to investigate the influence of connections on the behavior of sub frame elements in fire. Silva and Coelho [4] presented an equivalent elastic model to evaluate the response of steel joints under bending and axial force.

Sarraj et al. [5] also developed 3-D ABAQUS models of fin plate connections, which include the important contact interaction between the bolts and the fin plate and beam web. The models were validated against lap joint data at ambient temperature and a fire test conducted by Wald et al. 2005 [6] at the Czech Technical University. Sarraj has used the FE modeling to develop a component spring model assembly.

A finite element model was developed by Al-Jabri et al. [7] to study the behavior of flush end plate bare steel joints at elevated temperatures using the general purpose finite element software ABAQUS. The finite element model was used to establish the moment-rotation characteristics of the flush end plate bare steel joints with a concentrated force at elevated temperatures. The joint components were modeled using 3-D brick elements, while contact between the various components was modeled using Coulomb friction. Material nonlinearity was considered to model steel members and the joint components. Degradation of steel