Journal of Constructional Steel Research 119 (2016) 144–153

Contents lists available at ScienceDirect

Journal of Constructional Steel Research

Local buckling behavior of welded stub columns with normal and high strength steels

Gang Shi *, Kelong Xu, Huiyong Ban, Cuocuo Lin

Key Laboratory of Civil Engineering Safety and Durability of China Education Ministry, Department of Civil Engineering, Tsinghua University, Beijing 100084, PR China

ARTICLE INFO

Article history:
Received 14 July 2015
Received in revised form 15 November 2015
Accepted 8 December 2015
Available online 19 December 2015

Keywords:
Local buckling
Axial compression member
Steel strength
Finite element analysis
Design guidance

ABSTRACT

High strength steels (HSSs) are increasingly applied in structural engineering due to their benefits in terms of mechanical performance and economy. Compared with normal strength steel (NSS) axial compression members, HSS members possess more critical local buckling behavior since its component plates may be designed as being more slender, and different mechanical properties of HSS, in terms of increased yield strength and reduced ductility, may result in different local buckling behaviors. In this paper, finite element (FE) analysis is performed to investigate the local buckling behavior of welded box section and I-section stub columns under axial compression with both NSS and HSS being incorporated, where initial geometric imperfections and welding-induced residual stresses are accurately simulated. Through this work, variation rules of post-buckling ultimate stress and local buckling stress of the axial compression members with steel yield strength and width-to-thickness ratio are clarified. By comparing the FE analysis results and existing test results with the corresponding design methods in ANSI/AISC 360-10, Eurocode 3 and GB 50017-2003, it is confirmed that the design methods for the local buckling behavior of welded box section and I-section stub columns under axial compression need to be modified. New design formulas are proposed to take the influence of the steel strength into consideration.

1. Introduction

Generally steels with a nominal yield strength $f_y \geq 460$ MPa are called high strength steels (HSSs). Compared with normal strength steel (NSS) structures, HSS structures exhibit many advantages, including higher safety, less material usage, lower energy consumption and more environmental friendliness [1]. Nowadays, HSS has been widely used in building and bridge structures, such as the Landmark Tower in Japan, the Reliant Stadium in U.S., the Sony Center in Germany, the Star City Complex in Australia, the Bird's Nest Stadium in China, and the Millau Bridge in France [1,2].

With an increase of the yield strength of steel, ultimate bearing capacity of the axial compression members becomes more sensitive to their stability behavior rather than the strength. Some experimental investigations have been conducted to investigate the effects of various geometric parameters, configuration details and material properties on the local buckling behavior of HSS axial compression members. Nishino et al. [3] carried out local buckling tests of welded box section stub columns fabricated from A514 steel ($f_y = 690$ MPa). Usami and Fukumoto [4,5] tested the overall, local and interactive buckling behavior of square and rectangular box section columns made from SM58 steel ($f_y = 460$ MPa) and HT80 steel ($f_y = 690$ MPa). Rasmussen and Hancock [6,7] studied the overall and local buckling behavior of BISALLOY 80 steel ($f_y = 690$ MPa) columns with I, box and cruciform sections. Clarin [8] experimentally investigated the longitudinal residual stresses and local plate buckling of a series of welded box section columns made from WELDOX 700 steel ($f_y = 700$ MPa) and WELDOX1100 steel ($f_y = 1100$ MPa). Shi et al. [9,10] carried out an experimental study on the local buckling behavior of welded box section and I-section stub columns fabricated from Q460 ($f_y = 460$ MPa) and Q960 ($f_y = 960$ MPa) HSSs, and equations in accordance with various national standards for predicting the buckling strength and ultimate strength were compared with the test results. In addition, some numerical studies have also been conducted by using finite element (FE) method. FE models validated against test results were respectively established by Tang et al. [11] with ABAQUS and Shi et al. [12] with ANSYS to analyze the local buckling behavior of HSS stub columns.

Based on the literature review, most of the existing research is focused on experimental studies, while very limited information is available regarding accurate numerical analyses. Furthermore, all the aforementioned experiments only concerned the ultimate loads of specimens except those by Nishino et al. [3], which measured both the ultimate loads and the local buckling loads.

A numerical study is conducted in this paper to investigate the local buckling of welded square box section (labeled as B specimen) and I-section (labeled as I specimen) axial compression member with different steel grades. Using the validated FE modeling method, FE models are developed for B specimens and I specimens under axial compression to

* Corresponding author at: Department of Civil Engineering, Tsinghua University, Beijing 100084, PR China.
E-mail address: shigang@tsinghua.edu.cn (G. Shi).

http://dx.doi.org/10.1016/j.jcsr.2015.12.020
0143-974X/

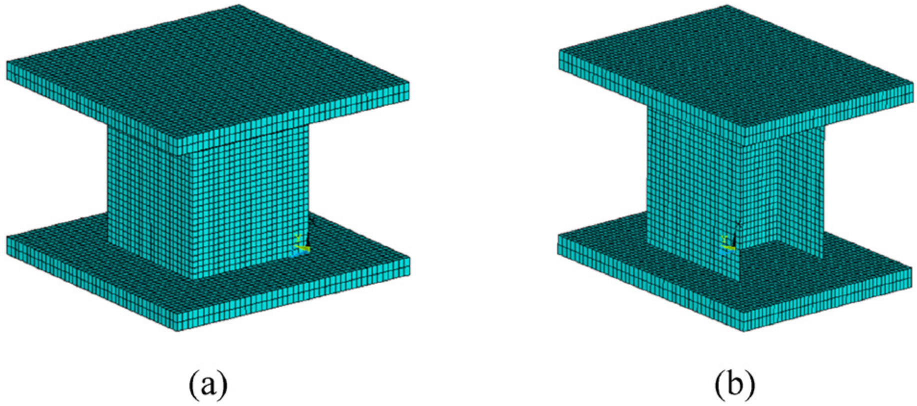

Fig. 1. Typical FE models: (a) B specimen; (b) I specimen.

perform parametric analyses, where the initial geometric imperfections and the residual stresses of specimens are accurately considered. Based on the FE analysis results, the variation rules of post-buckling ultimate stress and local buckling stress with steel strength and width-to-thickness ratio of component plates are elucidated, and are further compared with corresponding design methods in accordance with different specifications for steel structures. Improved design approaches are also proposed accordingly.

2. FE analysis

2.1. FE model validation

FE modeling was performed using general-purpose finite element software, ANSYS. The column model was meshed by using the 4-node finite strain shell element, SHELL181 with functions of linear eigenvalue buckling analysis, nonlinear large strain buckling analysis and input of initial stresses. A 40 mm thick rigid end plate established at either end of the columns was meshed by the 3-D structural solid element, SOLID95, which was used to transfer the load and to prevent the stress concentration. Only translational degrees of freedom of the nodes shared by both the column elements and the end plate elements were coupled.

Fig. 1 shows typical FE models, where each component plate was meshed into 22 elements along the width direction, and the element size along the axial direction was around 20 mm. It was found through mesh refinement trial that when the element number along the width direction was increased up to 44 and the element size along the axial direction was reduced to 5 mm, the deviation of the numerical results was within 1%. Therefore the former mesh method was demonstrated to be accurate enough and it was applied in the FE analysis. Because all the specimens concerned herein were stub columns which could not fail by overall buckling, both ends of the specimens were fully fixed except the axial directional translation of the top end for loading application.

The shape of the local initial geometric imperfection was defined by using the first eigenvalue buckling mode, as shown in Fig. 2. According to GB 50205-2001 [13], the amplitudes of initial geometric imperfection were $h_0/200$ for the component plates of B specimens and the webs of I specimens, and $B/100$ for the flanges of I specimens. The local initial geometric imperfections were applied in the FE models by using the command UPGEOM to update the geometry of the model.

The welding-induced residual stress was also considered in the FE modeling. The distributions and calculation formulas of residual stresses proposed by Ban et al. [14] were employed in this paper, as shown in Fig. 3. To simplify the distribution pattern applied in the FE meshes, a step shape calculated according to the stress equilibrium, drawn in dotted lines in Fig. 3, was employed in this work. The simplified residual stresses were applied in the FE models by using the command INISTATE, which implies that the 5 integration points of each element possess the same value of initial stress for representing the residual stress.

Four steps were included in the FE analysis in this paper. Firstly, the FE model of a stub column specimen with end plates was established

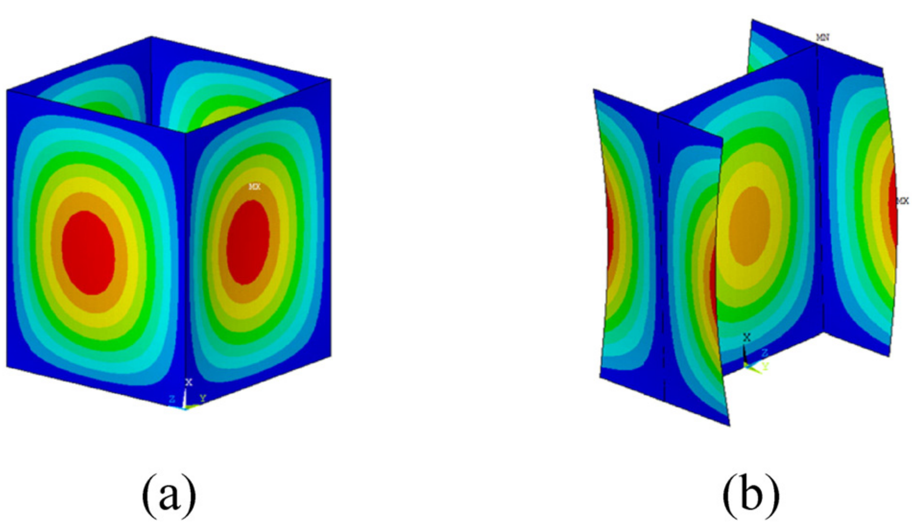

Fig. 2. Geometric imperfection model: (a) B specimen; (b) I specimen.