吉林大学学报(工学版) 2009 39 (01): 113-118 ISSN: 1671-5497 CN: 22-1341/T

运用ANSYS分析超高强度钢材钢柱整体稳定特性

施刚，石永久，王元清

清华大学 土木工程系，结构工程与振动教育部重点实验室，北京 100084

收稿日期 2007-7-18 修回日期 2007-11-10 网络版发布日期 2009-1-2 接受日期 2007-11-16

摘要 通过运用通用有限元软件ANSYS建立有限元模型，对5个超高强度钢材焊接工形截面轴心受压柱的整体稳定受力特性进行有限元分析。详细地介绍了建立有限元模型的具体方法，给出了分析其整体稳定特性的求解全过程，提出了输入构件几何初始缺陷和模拟截面残余应力的方法。通过将有限元计算结果与相应的试验结果进行对比，验证了本文建立的有限元模型的有效性。计算结果还表明，残余应力的变化对超高强度钢材焊接工形截面轴心受压柱整体稳定承载力的影响较小。

关键词 土木建筑结构 超高强度钢材 钢柱 整体稳定 ANSYS 有限元分析

分类号 TU391

Analysis on overall buckling behaviour of ultra-high strength steel columns by ANSYS

SHI Gang,SHI Yong-jiu,WANG Yuan-qing

Department of Civil Engineering,Key Laboratory of Structural Engineering and Vibration of China Education Ministry,Tsinghua University,Beijing 100084,China

Abstract The finite element analyses(FEAs) were performed using the general software ANSYS for the overall buckling behaviors of the 5 ultra-high strength steel welded I-seetion columns under the axial compression. The building procedure of the FEA model and the solution programs were introduced in detail. The method to simulate the initial geometric defects and the residual stresses in the cross-sections were proposed. The effectiveness of the performed FEA was verified by the comparison between the FEA results and the test results. The FEA also indicated that the residual stress only has little effect on the overall buckling resistance of the ultra high strength steel columns.

Key words civil engineering structure ultra high strength steel steel column overall buckling ANSYS finite element analysis （FEA）

DOI:

通讯作者 施刚 shigang@tsinghua.edu.cn