Journal of Constructional Steel Research 125 (2016) 227–238

Contents lists available at ScienceDirect

Journal of Constructional Steel Research

Resistance of cold-formed high strength steel circular and polygonal sections - Part 2: Numerical investigations

Anh Tuan Tran [a,*], Milan Veljkovic [b], Carlos Rebelo [c], Luís Simões da Silva [c]

[a] Luleå University of Technology, Sweden
[b] Delft University of Technology, The Netherlands
[c] ISISE, Department of Civil Engineering, University of Coimbra, Portugal

ARTICLE INFO

Article history:
Received 23 November 2015
Received in revised form 6 June 2016
Accepted 17 June 2016
Available online 2 July 2016

Keywords:
High strength steel
Cold-formed sections
Geometrical imperfections
Wind turbine towers
FEA

ABSTRACT

This paper is the second part of the study on the cold-formed high strength steel circular and polygonal sections intended to be used in tubular wind towers. Results from 32 numerical finite element analysis (FEA) models were compared with and calibrated against results of the tests on 32 corresponding specimens. The FEA results agreed well with the experimental results in terms of resistances and load-displacement curves. Further investigations on the numerical models were performed. Yield stress used in the FEA significantly affected the resistances of the numerical models. Using 0.2% proof stress leaded to higher resistance than the experimental results. Corners significantly influenced buckling behaviour in the polygonal section models. Analyses of an oval opening in the tubular specimens showed that peak stresses around the opening were considerably higher in the polygonal section models than in the circular section models. Finally, investigation of sensitivity to geometrical imperfections indicated that failure modes of numerical models with geometrical imperfections according to EC3 significantly differed from those of tested specimens and numerical models with geometrical imperfections obtained from the 3D scans.

1. Introduction

Finite element analysis (FEA) is particularly useful for studying structural behaviours. It is widely used to investigate structures' behaviours and extend understanding obtained from test results. Baniotopoulos et al. have used Strand7 software and STATIK-3 software to investigate the behaviour of a prototype 1 MW 44 m high wind turbine tower in Laconia, Greece [1]. The diameters at the base and top of this prototype are approximately 3.3 m and 2.1 m, respectively, and wall thicknesses vary from 18 mm at the bottom to 10 mm at the top.

The tower structure was investigated under gravity, seismic and wind loadings. The cited authors presented stress distributions and design recommendations for the tower. FEA has also been used to study design issues associated with use of square cross-sections in wind turbine towers instead of circular cross-sections in [2], under dynamic and static fatigue loads, and load cases according to IEC 61400-1 (2015) [3]. Maximum and Von Mises stress distributions in the square cross-sections have been presented. Lee et al. have used FEA to predict the lateral buckling load of a 45 m high wind turbine tower [4]. The FEA results were compared with data obtained from a 600 kW wind turbine that collapsed in JeJu, Korea, in 2010.

Cold-formed and high strength steel structures have been extensively studied by numerical models using the finite element method (FEM). Notably, Abaqus software has been used for two-step numerical simulation of the buckling behaviour of high strength steel cold-formed purlins [5], involving elastic analysis to obtain buckling modes, and then nonlinear Riks analysis with introduction of geometric imperfections. FE results have been compared to the test results. FE simulations of the behaviour of axially compressed high strength steel (S460) columns with H-sections have also been compared with results from tests on six specimens (particularly the ultimate strengths obtained), and a parametric analysis of 72 numerical models has been applied to develop buckling design curves for such columns [6]. The behaviour of cold-formed steel perforated sections in compression has been examined in comparative numerical FE and experimental investigations of 16 stub columns, and the sensitivity to various geometrical imperfections in the numerical models has been examined [7]. The local buckling analyses of 460 MPa high strength steel welded section stub columns using the FE program ANSYS (considering four box section members and nine I-section members under axial compression) have been published [8]. The results have showed good agreement with experimental data. Parametric analyses with 13 square box specimens and 33 I-section specimens have been also performed in the same study, then results from the experiments and parametric analysis have been compared to recommendations in three design codes: GB 50017-2003 [9], AISC

* Corresponding author.

http://dx.doi.org/10.1016/j.jcsr.2016.06.014
0143-974X/

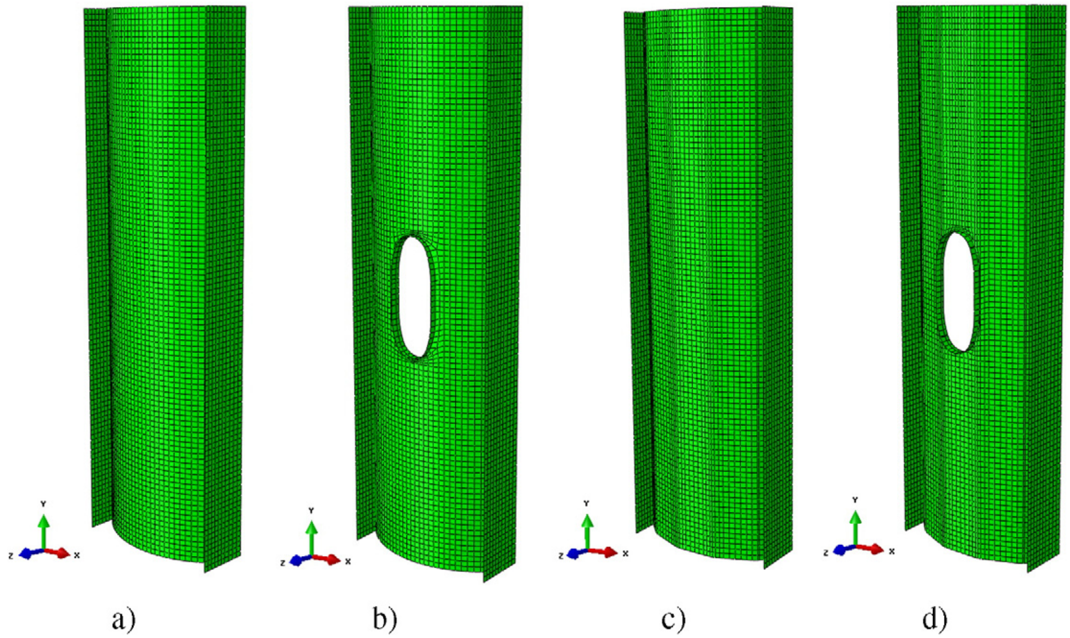

Fig. 1. Mesh of numerical model of: a) circular section without opening; b) circular section with opening; c) polygonal section without opening and d) polygonal section with opening.

360-10 [10] and Eurocode 3 [11]. The cited authors have concluded that using design codes for the flanges of I-section members lead to underestimation of the ultimate stress.

In the study presented here numerical investigations were performed on 32 cold-formed high strength steel circular and polygonal sections tested under compression in [12], using Abaqus [13]. First, FE models were constructed and calibrated against the experimental results. Then we examined the significant influence of yield strength on resistances of the numerical models, effects of the polygonal cross-sections' folding zones on their buckling behaviour, and behaviours of openings in the circular and polygonal section models. Finally, sensitivity to geometrical imperfections in the numerical models was analysed by introducing geometrical imperfections according to EC 3 and real geometrical imperfections obtained from 3D scans of the specimens.

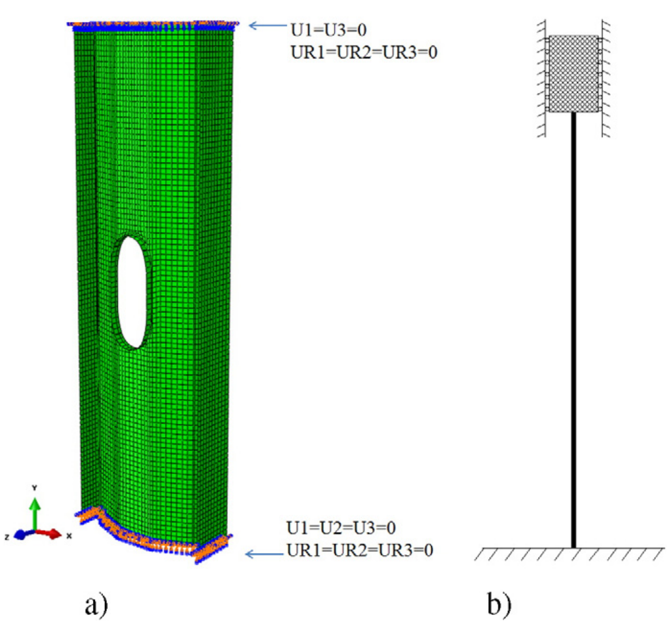

Fig. 2. Boundary conditions.

2. Finite element model

2.1. Finite element type and mesh

The 32 specimens have circular or polygonal cross-sections, they are 4 or 6 mm thick, and are either entire or have oval openings, as previously described in detail [12]. The thicknesses of the 4 mm and 6 mm specimens are approximately 0.4% and 0.6% of their lengths, respectively, and in both cases approximately 1.4% of their widths. Therefore, use of shell elements is suitable for the numerical models. Several types of shell elements are available in the commercial FEA program Abaqus [13]. The one used here is S4R, a shell element with four nodes and reduced numerical integration which is suitable and widely used for buckling and Riks analyses.

Fig. 1 presents typical FE meshes of the circular and polygonal numerical models with and without openings. The numerical models

Table 1
Comparison between FEA and test results for 4 mm specimens.

	Geometry of specimen		Ultimate load (kN)		
	tp	Width	FEA	Experiment	Ratio
Specimen	(mm)	(mm)	Pult,FEA	Pult,test	Pult,FEA/Pult,test
Cir-4-1000-1	4.01	306.2	2195	2115	**1.04**
Cir-4-1000-2	4.01	306.0	2189	2103	**1.04**
Cir-4-1000-3	4.03	306.5	2201	2098	**1.05**
Cir-4-1000-4	4.01	307.0	2168	2124	**1.02**
Cir-D-4-1	3.98	307.0	1547	1636	**0.95**
Cir-D-4-2	4.01	307.3	1542	1613	**0.96**
Cir-D-4-3	4.03	306.8	1543	1634	**0.94**
Cir-D-4-4	3.99	306.8	1543	1640	**0.94**
Pol-4-1000-1	4.00	303.9	2090	2102	**0.99**
Pol-4-1000-2	4.00	304.1	2017	2092	**0.96**
Pol-4-1000-3	4.02	304.1	2017	2081	**0.97**
Pol-4-1000-4	4.01	304.9	2094	2097	**1.00**
Pol-D-4-1	3.97	303.5	1647	1668	**0.99**
Pol-D-4-2	3.99	305.0	1660	1640	**1.01**
Pol-D-4-3	4.00	304.2	1665	1653	**1.01**
Pol-D-4-4	4.01	305.8	1667	1651	**1.01**
Mean					**0.99**
COV					**0.04**