International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 10 Issue: 08 | Aug 2023 www.irjet.net p-ISSN: 2395-0072

Numerical study on behaviour eccentrically loaded double circular steel tubular short columns filled with concrete

N. D. Prakadhesh[1], Dr. M. P. Muthuraj[2]

[1]PG Scholar, Department of Civil Engineering, Coimbatore Institute of Technology, Tamilnadu, India
[2]Associate Professor, Department of Civil Engineering, Coimbatore Institute of Technology, Tamilnadu, India

---***---

Abstract

This paper presents a comprehensive comparative analysis between experimental results and numerical simulations using Abaqus software for concrete-filled double steel short columns. The study aims to validate the accuracy and reliability of the numerical model in predicting the structural behaviour of such columns under axial load and lateral loading conditions.

The experimental investigation involved the fabrication and testing of a series of concrete-filled double steel short columns, varying in dimensions and reinforcement details. Load-displacement responses, load-carrying capacities, and failure modes were recorded during the experiments. Subsequently, an Abaqus finite element model was developed to simulate the behaviour of the tested columns. Material nonlinearities, concrete confinement effects and steel-to-concrete bond were considered in the numerical simulations.

Comparisons between experimental and numerical results were performed in terms of load-displacement curves, load capacities and failure modes. The agreement between the experimental and numerical results was assessed quantitatively, and the discrepancies were analysed to identify potential sources of differences. Factors such as material properties, boundary conditions, and modelling assumptions were critically examined to understand their influence on the simulation outcomes.

The findings of this study contribute to the validation and refinement of numerical modelling techniques for concrete-filled double steel short columns. The research enhances the understanding of the structural behaviours of such columns and provides insights into the effectiveness of Abaqus software in simulating their response under varying loading conditions. This work is valuable for structural engineers and researchers involved in the design and analysis of steel-reinforced concrete structures, aiding in the development of more accurate and efficient design methodologies.

Keywords: Concrete Filled Steel Column, Strength, Deflection, Abaqus.

1. Introduction

Circular concrete-filled double steel tubular (CFDST) columns are being increasingly used in high-rise and super high-rise buildings due to their exceptional strength and ductility. These columns consist of an inner concrete-filled steel tube strengthened by an outer circular steel tube filled with concrete. This configuration provides effective confinement to both the core concrete and the sandwiched concrete between the two tubes. While circular CFDST columns loaded axially have been studied, there's a lack of research on their behavior under eccentric loads.

The paper aims to address this gap by presenting experimental and computational investigations into the performance of short CFDST circular columns subjected to eccentric loading. The study outlines a comprehensive approach to analyze and model the moment-curvature behavior and strength envelopes of these columns. The research also considers the influence of various parameters, such as different concrete strengths and tube configurations, on the structural performance of eccentrically loaded CFDST columns.

Existing design codes like Eurocode 4 and ANSI/AISC 360–16 have not provided specific guidelines for designing circular CFDST beam-columns due to limited experimental and computational studies. This paper seeks to contribute to the understanding of the behavior of CFDST columns under eccentric loading conditions and provide insights that can inform future design practices.

The investigation begins with a detailed presentation of the experimental test program and the obtained results. A nonlinear numerical analysis procedure is introduced to model the moment-curvature response and strength characteristics of CFDST columns. The accuracy of the numerical approach is verified through experimental validation. The paper then presents the findings of parametric studies conducted using the developed computer model, focusing on the structural performance of eccentrically loaded CFDST short columns.

Although studies on axially loaded CFDST columns have been conducted in the past, this paper represents a pioneering effort in investigating the behavior of short CFDST circular columns under eccentric loading. The research aims to contribute to the knowledge base for designing and optimizing circular CFDST columns in composite structures, shedding light on their performance in scenarios where eccentric loads are prevalent.

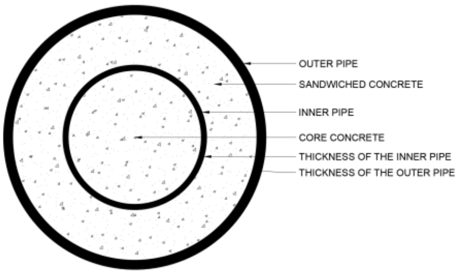


Figure 1 Plan View of the CFSCC

2. Finite Element Analysis

2.1 General

The numerical models in this study were constructed using the finite element analysis (FEA) software ABAQUS [5]. To validate the accuracy of these models, the obtained results were compared with experimental work and existing findings in the literature. This comparison served as a confirmation of the reliability and precision of the FE models. Additionally, a database was created through parametric analysis, which involved varying the load parameters to investigate their influence on the buckling strength. This approach allowed for a comprehensive assessment of how different loads affect the structural behavior and buckling capacity.

2.2 Material properties and Geometry

Concrete, Steel Tube and Steel Profile mechanical properties from Marsimoyi Terefe Bango.[6] are used to model the I Section components.

The behavior of concrete under various loading conditions was studied using ABAQUS, focusing on axial load-induced failure modes including compression crushing and tensile cracking. A continuum damage model rooted in plasticity was employed to examine and elucidate these failure mechanisms.

The Concrete Plasticity model encompasses the following key parameters:

1. The material exhibits distinct mechanical behaviors dictated by specific parameters. In the realm of plasticity, it showcases a Dilatation angle of 30 degrees, an eccentricity of 0.1, an fb0/fc0 ratio of 1.16, a K value of 0.667, and a viscosity parameter of 0. These characteristics govern the material's response to plastic deformation.

2. Under compressive forces, the material initiates yielding at a stress threshold of 25MPa. This behavior is pivotal for analyzing how the material reacts to external loads exerted upon it.

3. Conversely, the material's reaction to tensile forces is characterized by a yield stress of 2.25MPa. This parameter is of utmost importance in understanding the material's susceptibility to deformation or failure under tension.

4. Furthermore, the material's Poisson's ratio (v) is determined as 0.2. This value sheds light on the material's tendency to expand laterally when subjected to axial loading.

In essence, these parameters collectively define the material's mechanical properties, facilitating a comprehensive grasp of its performance under distinct loading scenarios, encompassing plasticity, compression, and tension.

I) Steel profile and Steel tube

Elastic-perfectly plastic modelling with Von Mises yield criterion is applied to the steel tube, steel profile capturing stress distribution from elastic to plastic regions during incremental loading.

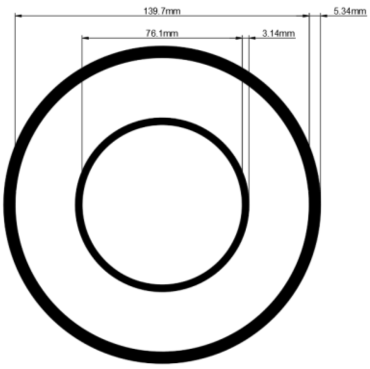


Figure 2 Dimensions of the CFSCC

Table 1 Detailed cross-section and Material properties of Steel tube

Eccentricity	Inner Tube		Outer Tube	
	D [mm]	Thickness [mm]	D [mm]	Thickness [mm]
e = 0	139.7	5	76.1	3.6
e = 10	139.7	5	76.1	3.6
e = 25	139.7	5	76.1	3.6
e = 30	139.7	5	76.1	3.6

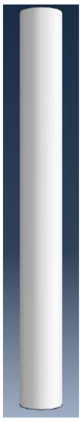
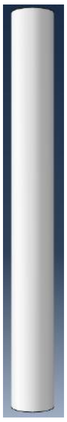
Note: Height of each column is 420 mm.

The parameters define for the steel tube nonlinearity modelling are the following:

1) Poisson's ratio, μ = 0.3
2) Density, ρ = 7860 kg/m^3
3) Young's modulus, Es = 210000MPa

In the modelling process, the cross-section dimensions are assumed to be uniform within the interval investigated in the previous study conducted by Marsimoyi Terefe Bango [6]. The material properties utilized in the modelling are consistently applied to all sections and their corresponding elements. For the purpose of the simulation, steel profiles are embedded in the composite column, aligning with the considerations made in the referenced paper. This approach ensures uniformity and allows for a direct comparison between the different sections analysed in the study. By maintaining these assumptions and standardizing the material properties and profiles used, the modelling process becomes more robust and facilitates a meaningful investigation into the behavior of the composite column.

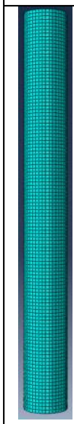
Table 2 Modelling of Outer Tube and Inner Tube

OUTER TUBE	INNER TUBE
	

2.3 Mesh convergence study and Element type

Mesh convergence theory in ABAQUS involves refining the discretization of a finite element model to ensure accurate and reliable numerical solutions. In a 15 x 15 x 15 mesh configuration, finer mesh elements are utilized, enhancing the representation of complex geometries and stress distributions. As mesh density increases, localized variations in stress, strain and displacement are better captured. The process entails progressively reducing element sizes while observing convergence trends in key output parameters, such as reactions or displacements. Convergence is achieved when further mesh refinement yields minimal changes in these outputs. A balance must be struck between computational efficiency and accuracy. Mesh convergence analysis verifies that results stabilize with finer mesh, ensuring dependable predictions. However, excessive refinement can lead to unnecessary computational overhead. In summary, performing mesh convergence studies for a 15 x 15 x 15 mesh configuration in ABAQUS establishes optimal discretization, guaranteeing credible simulation outcomes while optimizing computational resources.

Table 3 Meshing of Outer Tube and Inner Tube

CONCRETE	TUBE
	

2.4 Boundary and Loading conditions

In the context of a parametric study aimed at determining the buckling strength of simply supported concrete-filled double steel tube short columns subjected to a compressive load, a specific modeling approach is employed. The objective is to establish a robust simulation framework to analyze the buckling behavior of such columns under varying conditions.

To ensure structural integrity and accuracy in the simulation, a rigid region is created. This is achieved by strategically placing nodes at specific locations: at the top and bottom ends of the section centroid, "e = 10," "e = 25," and "e = 30." These nodes are then connected to the section's edge nodes, which are considered dependent nodes.

The key independent node, positioned precisely at the geometric centroid, plays a crucial role in the simulation. It is intricately linked to all dependent nodes, thereby facilitating comprehensive interactions within the model. Moreover, a master node, situated 30 mm above the specimen, is strategically designated to apply essential boundary and loading conditions in a consistent manner across all directions.

This meticulously devised approach ensures that the parametric study accurately captures the buckling strength of the concrete-filled double steel tube short columns. By effectively controlling variables and boundary conditions, the simulation offers valuable insights into the structural behavior of these columns and aids in the exploration of their buckling characteristics.

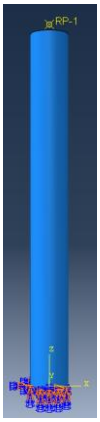
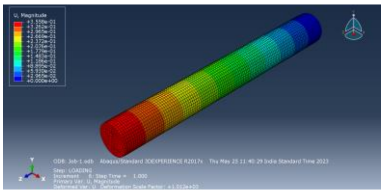
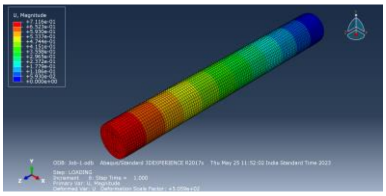
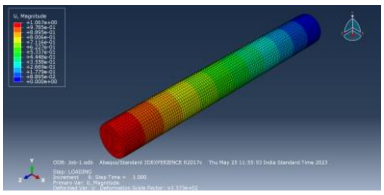
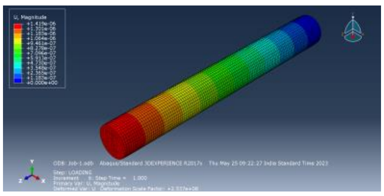


Figure 3 Boundary Condition

2.5 Deformation analysis

Table 4 Deformation Result

LOAD	RESULT
Centroid	
e = 10	
E=25	
E= 30	

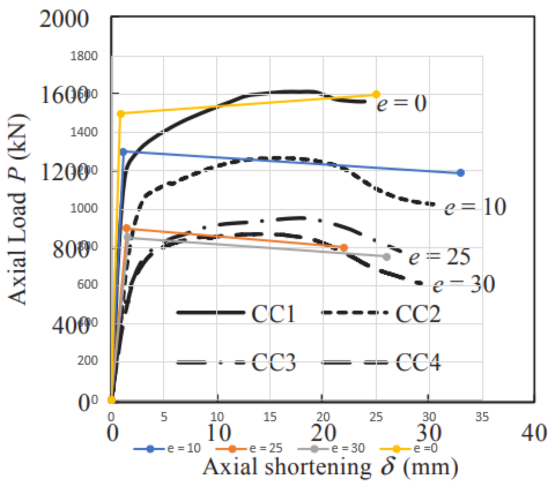
2.6 Verification

The validation process involves confirming the accurate representation of the specimen within the Abaqus software and subsequently comparing the simulation results with those obtained from a reference paper. The reference paper serves as the source of experimental data and includes a detailed description of the physical model as well as graphs illustrating its behavior.

To establish the fidelity of the Abaqus model, meticulous attention is given to ensuring that all relevant geometric, material, and boundary conditions are faithfully incorporated into the simulation. This involves a thorough examination of the model's configuration, including the positioning of nodes, elements, and connections.

Upon completing the simulation, the results obtained from Abaqus are meticulously compared with the data presented in the reference paper. The focus of this comparative analysis centers on identifying similar trends and patterns in the obtained graphs. The aim is to pinpoint instances where the simulated graph closely resembles the graph from the reference paper, validating the accuracy of the numerical model.

By undertaking this verification process, the study aims to confidently assert the reliability of the Abaqus simulation in replicating the behavior of the physical specimen. The successful alignment of simulation results with those of the reference paper underscores the robustness of the numerical approach and substantiates its capacity to predict the real-world response of the concrete-filled double steel tube short columns. Mizan Ahmed et. al. [7]



Graph 1 Verification of the Axial Shortening

5. Conclusion

In conclusion, the conventional steel composite column exhibited higher deformation compared to the shear connect used composite column. The presence of shear connectors in the latter column enhanced the interaction between the steel and concrete components, resulting in reduced deformations. The shear connectors effectively transferred shear forces and improved the composite action, leading to a stiffer and more robust structural response. This interaction between steel and concrete in the shear connect used composite column played a crucial role in mitigating deformation and enhancing the overall performance of the column system, making it a favorable choice in applications where deformation control is a critical consideration.

References

[1] Shane Abrahama , Lija R La , V Vandhana Devia, Behavior of Self-Cured Self-Compacting Concrete filled Short Channel Columns under Axial Compression (2022)

[2] Mathias Johansson and Kent Gylltoft, Mechanical Behavior of Circular Steel–Concrete Composite Stub Columns (2002)

DOI: 10.1061/(ASCE)0733-9445(2002)128:8(1073)

[3] Zhi-Lu Jia, Yan-Li Shi, Wen-Da Wang , Sun-Hang Ji, Compression-bending behaviour of steel-reinforced concrete-filled circular steel tubular columns with preload

DOI: https://doi.org/10.1016/j.istruc.2021.12.056

[4] Bo Yang, Le Shen, Kang Chen, Chen Feng, Xuchuan Lin, Mohamed Elchalakani and Shaoqian Xu, Mechanical Performance of Circular Ultrahigh-Performance Concrete–Filled Double Skin High-Strength Steel Tubular Stub Columns under Axial Compression

DOI: 10.1061/(ASCE)ST.1943-541X.0003291.

[5] ABAQUS (2011), "Standard User's Manual", Hibbitt, K.a.S., Inc., Vol. 1, 2 and 3, Version 6.11- 1, USA.

[6] M. Kaarthik, S. M. Rubhan Prakaash, Um Chandru, Experimental and Analytical study with stiffened web cold formed steel lipped columns with Batten Sheets

[7] Mizan Ahmeda , Qing Quan Lianga,∗ , Vipulkumar Ishvarbhai Patelb , Muhammad N.S. Hadic Behavior of eccentrically loaded double circular steel tubular short columns filled with concret.

DOI: https://doi.org/10.1016/j.engstruct.2019.109790

http://www.solidstatetechnology.us/index.php/JSST/article/view/1676