Finite element analysis of u-shaped reinforced concrete column

Jiuyang Li 1, Xiaoyu Wang 1*, Weinan Ma 2
1 School of Civil Engineering, Changchun Institute of Technology, Changchun, China
2 Sinohydro Northeast Survey and Design Research Co., Ltd.
Corresponding author’s e-mail: wxy18339168120@163.com

Abstract: Special shaped column is called frame column with special-shaped section. Under the condition of fully considering the strength and stiffness requirements of the whole structure, the U-shaped column is used in the new structural system of contemporary subway architectural pattern, which mainly uses special-shaped column instead of ordinary rectangular column to form the so-called "explicit frame", so as to promote the interaction between steel and concrete, and greatly improve the bearing capacity of components power. In this paper, the bearing capacity, bending deformation and failure mode of U-shaped reinforced concrete column are analyzed through the finite element software ABAQUS under the change of wall thickness and height, which provides a theoretical basis for the future research on the performance of U-shaped reinforced concrete column.

1. Introduction

With the rapid development of social economy, people’s design concept is becoming more and more novel. The main task of design is to ensure structural safety first, and then create conditions on the basis of which to improve and develop the architectural function, making the design and use of subway buildings more flexible and convenient. The rectangular and circular reinforced concrete column structure are used in China’s subway buildings, which is convenient for construction, but the bearing capacity is not good. It has these disadvantages, such as self-weight, seismic performance overall poor and inflexible architectural layout. Multi-storey and high-rise structures have been more and more widely used in practical engineering, in order to improve the bearing capacity and meet the requirements of various aspects and ensuring the bearing capacity and ductility of the structure within the safe range, it is necessary to appropriately increase the cross-section size of the rectangular column, so that it cannot be met at the same time.

In reinforced concrete special-shaped columns, the change of section size makes the constraint effect of internal concrete stronger, which is in a state of symmetric compression, and the bearing performance is greatly improved. The supporting effect of concrete increases the overall stability of steel bars and avoids local buckling, the ductility characteristics may also be improved. However, considering the complex stress of the components in the actual subway project, the research on the deformation and failure characteristics of reinforced concrete special-shaped columns is limited.

In order to determine the bearing capacity, bending deformation and failure mode of U-shaped reinforced concrete columns under certain conditions, the finite element analysis software analyzes the stress and deformation performance, which provides a basis for experimental research.
2. Abaqus software introduction
Finite element method is a widely used method in engineering simulation and plays a vital role in different industries and fields. As one of the largest finite element softwares in the world, ABAQUS can be simulated in a wide range. This software provides a large number of different types of element models, material models, and analysis processes, which can provide research in different fields for selection. The most significant advantage of this software is that it can almost solve all simple linear elastic problems, and can also provide corresponding solutions for some complex nonlinear problems. When using this software for linear analysis, the calculation efficiency is high. In the calculation process, the software can automatically select the load increment suitable for the model by analyzing the characteristics of different models, and adopt reasonable convergence criteria. In the calculation process, these parameters can be adjusted automatically and continuously in time to facilitate calculation. Finally, the maximum accurate solution effect is achieved, and the analysis efficiency is greatly improved. The analysis module of ABAQUS is divided into two parts: ABAQUS / Standard for static analysis and ABAQUS / Explicit for dynamic analysis. This paper mainly uses static analysis.

3. Abaqus finite element analysis process

3.1. Working condition of specimen
Finite element specimen size: In order to consider the influence of U-shaped column cantilever wall thickness and height on the bearing capacity, the parameters of finite element analysis are designed. U-shaped column cantilever wall thickness is 150mm, 200mm, 250mm; When the cantilever wall thickness of U-shaped column is 150mm, the height of U-shaped column is 1500mm, 2000mm, 2500mm respectively. A total of two groups of six finite element specimens are designed, and the calculation parameters are shown in Table 1 and Table 2.

| Working condition | Test specimen number | Reinforcement ratio | Stirrup reinforcement ratio | Wall thickness/mm | Height /mm |
|-------------------|----------------------|---------------------|----------------------------|-------------------|-----------|
| group one         | 1                    | 0.29%               | 0.58%                      | 150               | 2500      |
|                   | 2                    | 0.29%               | 0.58%                      | 200               | 2500      |
|                   | 3                    | 0.29%               | 0.58%                      | 250               | 2500      |
|                   | 4                    | 0.29%               | 0.58%                      | 150               | 1900      |
| group two         | 5                    | 0.29%               | 0.58%                      | 150               | 2200      |
|                   | 6                    | 0.29%               | 0.58%                      | 150               | 2500      |

3.2. Properties and constitutive relations of concrete materials
The U-shaped concrete used in this test is C50 concrete, and the mechanical properties of C50 concrete materials are determined according to Code for design of concrete structures GB50010 - 2010. The mechanical properties of C50 concrete are shown in Table 3. The stress before yield should yield 360 MPa, which is considered to be in a fully elastic state. The mechanical properties of the steel bar are shown in Table 4.
Table 2. Mechanical properties of C50 concrete

| grade of concrete | Elastic modulus N/mm² | Poisson ratio $\lambda$ | Standard value of uniaxial tensile strength N/mm² | Standard value of uniaxial compressive strength N/mm² | Compression stiffness recovery factor | Recovery factor of tensile stiffness |
|------------------|-----------------------|-------------------------|-----------------------------------------------|-----------------------------------------------|----------------------------------|-----------------------------------|
| C50              | $2.69 \times 10^4$    | 0.2                     | 1.89                                         | 23.1                                         | 0.9                              | 0                                 |

Table 3. Mechanical property of steel bars

| Steel grade | Design tensile strength N/mm² | Elongation percentage N/m² | Elastic modulus N/mm² | Poisson ratio $\lambda$ |
|-------------|--------------------------------|---------------------------|-----------------------|--------------------------|
| HRB400      | 360                            | 7.5                       | $2.00 \times 10^5$    | 0.25                     |

3.3. Modeling process

The finite element analysis software ABAQUS is used to create the components of U-shaped concrete column in the ratio of 5:1 through the component creation (part module). Summarizes the experience of finite element analysis results, combined with the structural characteristics of reinforced concrete structure itself, using displacement coordination modeling method to establish the model.

The material properties are defined, and the material properties of each component in U-shaped concrete are defined through the property module. The definitions of concrete and steel are determined according to the above material properties and constitutive parameters, respectively.

Component assembly and interaction, the assembly module is used to assemble the various components of the model orderly, and then assembly positioning is carried out to form the assembly. Finally, through the interaction module, the through-length reinforcement, stirrup, tie bar and longitudinal reinforcement are bound into a whole embedded into the whole concrete component. The assembly of finite element model components is shown in Figure 1.

![Finite element components](image)

(a) cage of reinforcement (b) concrete

Figure 1. Finite element components

4. Finite element analysis process

Through the finite element software, the deformation of U-shaped reinforced concrete columns under different working conditions is analyzed. The bending forms of U-shaped columns under vertical load are shown in Figure 2. According to the working conditions in Table 1 and Table 2, the factors affecting the deformation of U-shaped columns are compared and analyzed, and the influence of wall thickness and height of U-shaped columns on the bearing performance of U-shaped reinforced concrete is obtained.
4.1. Influence of wall thickness and height variation on u-shaped reinforced concrete columns

4.1.1. Load capacity of lubricant-carrying properties. Under the finite element calculation model, the relationship between the bearing capacity and time of the specimen is shown in Figure 3. The analysis results show that under certain conditions, with the increase of U-shaped concrete column wall thickness, the maximum bearing capacity of the column increases gradually. With the increase of the height of U-shaped concrete column, the bearing capacity of the column increases gradually.

4.1.2. Damage pattern. This model simulates the vertical force along the centroid direction, under different wall thickness, axial compression deformation and failure characteristics are basically the same in monotonic loading process, this paper takes the equivalent plastic strain diagram Figure 4 of concrete and steel bar as an example to reflect the deformation and failure characteristics of the specimen.
From Figure 4, it can be seen that under axial compression, the U-shaped concrete column is in the middle of the drum failure, and there is no significant change in the upper and lower ends, so the deformation in the middle of the column is the largest.

It can be seen from the deformation and stress nephogram of the reinforcement in Figure that the stress values of the longitudinal reinforcement and the horizontal stirrup in the middle of the column are the largest, reaching 360 MPa. The stirrup is explicitly red in the figure. The reason why most of the longitudinal reinforcement in the figure is explicitly red is that under the action of axial load, each longitudinal reinforcement is uniformly loaded. When the load reaches the yield stress of the reinforcement, the longitudinal reinforcement in the figure is all red. The horizontal stirrup stress at the upper and lower ends of the specimen is small. It can be seen from the deformation diagram that the specimen has an outward protruding deformation in the middle of the column, and the horizontal stirrup has a transverse deformation.

It is not difficult to see from the deformation and stress nephogram of concrete in Figure 4 that under the action of axial load, the axial force gradually transfers from the upper part of the column to the middle part of the column. At this time, the concrete stress in the middle part of the column begins to gradually increase. When the ultimate compressive strain of concrete is reached, the concrete stress in the middle part of the column shows a downward trend. Therefore, in the stress nephogram of external concrete, the stress in the middle part of the column is the smallest, which is explicitly shown as the blue area in the figure.

5. Conclusions
(1) In this paper, the axial compression of U-shaped reinforced concrete columns is simulated and analyzed by using the finite element analysis software ABAQUS, and the stress-strain curves of the compression components are obtained. The mechanical properties and deformation characteristics of U-shaped concrete columns under two working conditions are compared. It is concluded that under the same other conditions, with the increase of height and wall thickness, the bearing capacity of U-shaped concrete columns has been significantly improved, and the ductility performance has also been significantly improved, showing the outstanding contribution of height and wall thickness to the bearing capacity and deformation capacity of components.

(2) In this paper, the failure characteristics and stress of the components are simulated according to the subway building. The failure mode of the whole U-shaped reinforced concrete column under the axial load is that the concrete is crushed, and the longitudinal reinforcement between the stirrups is lateral bulged. The failure mode of the whole column is that the middle bulge is moon-shaped. After the upper and lower ends are stressed, the force is effectively transferred from the upper and lower ends to the middle part. The load is concentrated in the middle position of the column, so the upper and lower parts are less damaged, and the middle part is moon-shaped.

(3) In this paper, the deformation curves of concrete and steel bar in the loading process are compared. Before the ultimate bearing capacity, the steel bar and concrete maintain a good synergy. When the specimen reaches the ultimate load, the steel bar plays a very important role in improving the ductility of the component.

References
[1] Rong,B., Chen, Z.H., and Zhou ,T. (2009) Research on axial compression strength of l-shaped shopt column composed of concrete-filled square steel tubes.,39:04-107.
[2] Rong,B., Chen, Z.H., (2009) Research on axial compression stability of L-shaped column composed of concrete-filled square steel tubes.,39:39-42.
[3] Li,K.W., Zhou, Y. (2012) Mechanical Properties Analysis of Steel Reinforced Concrete Column Based on ABAQUS, J.SiChuanJianZhuKeXueYanJiu.,38:67-69.
[4] Guo,Y.F. (2017) Seismic Performance Analysis for T Column with Concrete Filled Steel Tube Basing on ABAQUS.
[5] Zhou, J. (2017) Finite element analysis of steel reinforced concrete columns based on ABAQUS, J. Guangdong Building Materials, 33: 49-53.

[6] Ling, Y. M., Hang, L., Zhang, H. B., Xu, W. K., and Xu, Y. F. (2019) Finite element analysis of axial compression performance of L-shaped steel reinforced recycled concrete special-shaped columns.

[7] Zhou, Y. Y., Xiao, Y. M. (2020) Finite Element Analysis of Rear Steel Tube Reinforced Concrete Column Based on ABAQUS, J. Anhui Building Materials, 27: 77-79.

[8] GB50010-2010, design code for concrete structure [S].