Hindawi
Journal of Sensors
Volume 2023, Article ID 9809731, 1 page
https://doi.org/10.1155/2023/9809731

Retraction

Retracted: Stress Analysis of Concrete Materials Based on Finite Element Analysis

Journal of Sensors

Received 19 December 2023; Accepted 19 December 2023; Published 20 December 2023

This article has been retracted by Hindawi following an investigation undertaken by the publisher [1]. This investigation has uncovered evidence of one or more of the following indicators of systematic manipulation of the publication process:

(1) Discrepancies in scope
(2) Discrepancies in the description of the research reported
(3) Discrepancies between the availability of data and the research described
(4) Inappropriate citations
(5) Incoherent, meaningless and/or irrelevant content included in the article
(6) Manipulated or compromised peer review

The presence of these indicators undermines our confidence in the integrity of the article's content and we cannot, therefore, vouch for its reliability. Please note that this notice is intended solely to alert readers that the content of this article is unreliable. We have not investigated whether authors were aware of or involved in the systematic manipulation of the publication process.

Wiley and Hindawi regrets that the usual quality checks did not identify these issues before publication and have since put additional measures in place to safeguard research integrity.

We wish to credit our own Research Integrity and Research Publishing teams and anonymous and named external researchers and research integrity experts for contributing to this investigation.

The corresponding author, as the representative of all authors, has been given the opportunity to register their agreement or disagreement to this retraction. We have kept a record of any response received.

References

[1] R. Yang and S. Yuan, "Stress Analysis of Concrete Materials Based on Finite Element Analysis," *Journal of Sensors*, vol. 2022, Article ID 1826598, 11 pages, 2022.

Hindawi
Journal of Sensors
Volume 2022, Article ID 1826598, 11 pages
https://doi.org/10.1155/2022/1826598

Research Article

Stress Analysis of Concrete Materials Based on Finite Element Analysis

Rui Yang and Shengli Yuan

Department of Civil Engineering and Architecture, Xinxiang University, 453000 Xinxiang, China

Correspondence should be addressed to Rui Yang; yangrui01@xxu.edu.cn

Received 1 July 2022; Revised 19 July 2022; Accepted 23 July 2022; Published 12 August 2022

Academic Editor: Yuan Li

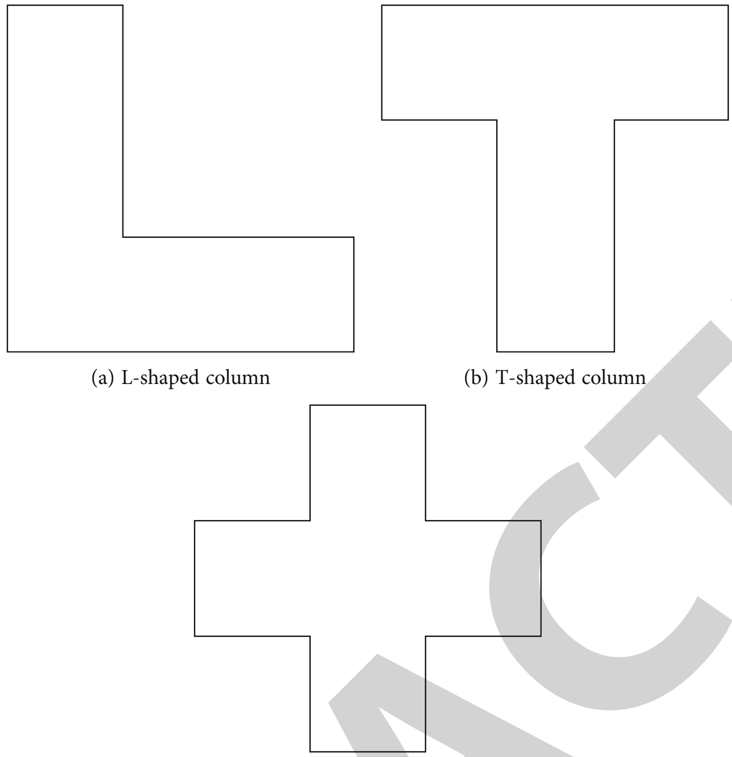
China's construction industry standard "technical code for concrete special-shaped column structure" has been implemented since 2006, and the use of special-shaped walls in the actual construction of buildings is more common. The cross-section wall can effectively reduce the protruding angle of the building, thereby expanding the effective building area of the room, reducing the proportion of the building components themselves, and making the structure layout of the building more beautiful. In order to analyze the dynamic characteristics of eccentrically compressed prestressed concrete beams, the finite element simulation of different reinforcement parameters, longitudinal reinforcement diameter, and reinforcement ratio was constructed. Through comparison, the load flexibility curve change law of concrete stress cloud and reinforcement stress cloud under various working conditions and the influence change law of test mold are studied, and the influence change law of dynamic characteristics under the above four conditions and unidirectional bias pressure is clarified.

1. Introduction

Concrete is a composite material widely used in the construction industry and other fields. For a long time, scientists and engineers have conducted a systematic research on all aspects of condensing ten materials. However, concrete is traditionally regarded as a uniform material, mainly by studying the macroscopic properties of materials. Since the middle of the last century, the micromechanics has provided a necessary theoretical basis to explore the essential characteristics of concrete materials from the fine view and from the microscopic level. Advanced experimental instruments help us to further understand the paper formation and internal structure of condensing ten materials. In the past ten years, the computer bin and the computing speed make the numerical method become an important means to analyze the characteristics of concrete materials. Therefore, it is of theoretical and practical significance to study the characteristics of concrete materials from the fine view or micro-level [1].

Columns with unconventional cross-section types usually have T-shaped, L-shaped and cross-shaped cross-section types, and the length to thickness ratio of the column limb shall not exceed four [1]. The main section types are shown in Figure 1. People can not only learn from it but also be different from each other through its use [2].

With the promulgation of China's "technical code for concrete special-shaped column structure" in 2006, the application of special-shaped beam structure in actual construction is more common. The utilization characteristics, design aesthetic rationality, and excellent bearing characteristics of the reinforced concrete special-shaped column structure are organically integrated to create a perfect indoor environment for modern consumers. It conforms to the characteristics of room design and is reasonably connected with the wall (refer to infilled wall), which is sought after by real estate developers and many consumer groups. In recent years, there is a wide application of reinforced concrete special-shaped columns in the actual building construction system [3].



(a) L-shaped column

(b) T-shaped column

(c) Cruciform column

FIGURE 1: Section style of shaped column.

The special-shaped wall frame system includes the cast in situ reinforced large-diameter concrete special-shaped wall frame and special-shaped column frame shear wall system used for light infilled walls and partitions. The reinforced concrete special-shaped column structure system is a special engineering structure system produced in the process of modern building structure design and construction in China, which makes the development and research of the reinforced concrete special-shaped column structure system in China ahead of the world [4]. Since the in-depth research on the technical system of reinforced concrete special-shaped column structure, considerable progress has also been made. For example, at present, the national research and development of the reinforced concrete special-shaped column structure system focuses on the fact that experts have carried out a lot of relevant experimental research and development work on the reinforced concrete special-shaped column structure, and for the corresponding regional-reinforced concrete special-shaped columns, they also put forward suggestions and opinions for the local technical standards of reinforced concrete special-shaped columns, for example, the industrial standard of Liaoning Province Technical Specification for reinforced concrete special-shaped column structure [5].

The research on the structural system of special-shaped reinforced concrete columns in China is very different from that in China. For example, the foreign research focuses on the offset, while the domestic research focuses on the earthquake resistance. At the same time, because of the existence of multiple limbs, the special-shaped column also has the problem of geometric asymmetry, resulting in the obvious brittleness of the structure of the concrete special-shaped column, and the deformation energy of the special-shaped column is weaker than that of the general rectangular column. Although there are the above problems in the special-shaped wall section column, based on the current design specifications of China, some places with relatively low seismic stiffness and meeting the corresponding height width ratio and elevation conditions can still be used in the actual construction. In the normal application of special-shaped columns, the probability of fracture caused by seismic effect is very small, so it is very important to increase the research on the dynamic characteristics of special-shaped columns under bias pressure [6]. The specific work of this paper is shown in Table 1.

2. State of the Art

ABAQUS finite element analysis software: ABAQUS is one of the most widely used analysis software in the professional field, with high-precision numerical simulation ability and super strong calculation ability. ABAQUS can generate a large number of analysis elements by using finite element software, which can be applied in different industries [7].

ABAQUS finite element software has the following features over other software in the field of numerical simulation.

(1) Easy to use and comprehensive functions

ABAQUS finite element analysis software is one of the most powerful engineering research and design application software, which can be used to model and study systems and fluids. These finite element analysis programs have brought effective calculation methods to relevant technical researchers, shortened the research process, and greatly improved the efficiency. In the actual project of using ABAQUS software to analyze the model, researchers can directly display the modeling results and the structural characteristics of simulated parts in the program.

The important data analysis modules of ABAQUS/FEA software include ABAQUS/explicit and ABAQUS/standard templates, as well as ABAQUS/CAE templates, which are the basic user interfaces of graphic solutions. Each data analysis software module has different classification methods, calculation methods, and key points. The main advantage of ABAQUS/obvious template is that it can visualize the whole dynamic scoring calculation process. However, in fact, ABAQUS/obvious template is in the calculation process. The advantage of ABAQUS/standard analytical model is that it can run on a solver that simultaneously calculates multiple eigenvalues, which is also a more commonly used method in linear dynamic analytical calculation. Finally, the advantage of ABAQUS/CAE standard analytical model is that it can establish more detailed numerical simulation model more quickly.

(2) Comprehensive cell library and material model library

There are about 400 types of basic units in the unit library of thermodynamic properties of various metal materials [7]. The element library in ABAQUS finite element software can roughly contain eight basic elements [8].

ABAQUS has a variety of material model libraries to model the thermodynamic properties of various materials. At the same time, we can define different material models according to different failure criteria and inherent structural relationships. Scientific researchers can select appropriate material model library from ABAQUS according to the structural characteristics of the special model materials they are developing. Through previous application research and exploration, people found that ABAQUS FEA software can more realistically simulate the characteristics of various materials in engineering practice.

(3) Strong compatibility

ABAQUS finite element analysis software system has strong compatibility. The software has a complete system interface, which can provide a good connection for the later secondary development, thus saving a lot of manpower, material resources and investment. ABAQUS finite element software has been applied to many problems and analysis [9].

(4) High operability

In ABAQUS FEA program, the whole working process mainly includes pre-processing (modeling, analysis and calculation) and post-processing. Each of the above links can be gradually realized through various components in the graphical interface, which is practical and does not require much work. The commonly used ABAQUS/CAE analysis model includes parts, characteristics, assembly, steps, interactions, loads, grids, jobs and dozens of modeling operation modules [10].

(5) Accurate nonlinear analysis

The calculation process of ABAQUS finite element software can not only solve the problem of studying model materials under various characteristics and test pieces under various stress environments, but also the relevant calculation tools have added the load information required by the research into the whole calculation process in the form of load steps, so as to obtain the correct nonlinear research conclusion more accurately. Similarly, in the ABAQUS FEM nonlinear calculation process, the whole calculation process can be regarded as a loading increment that changes continuously with the increase of time, and then, through iterative calculation, the final result is obtained. In the process of nonlinear calculation by ABAQUS finite element software, the software can intelligently change the rate of load increase and the convergence rate of control calculation, so that more accurate results can be obtained through the nonlinear research of ABAQUS software in practical problems [11]. The block diagram of the analysis of the concrete material using the ABAQUS software is shown in Figure 2.

ABAQUS finite element analysis software is a powerful and widely used engineering simulation application software. ABAQUS FEM software has a cell library that can simulate the characteristics of all building geometric components and a model library that uses the characteristics of various building metal materials, which can simulate the characteristics of metal materials used in all typical buildings. For example, the mechanical properties of reinforced concrete special-shaped beams under the influence of eccentric load can be analyzed using ABAQUS finite element analysis software [12].

There are many reasons to change the dynamic characteristics of reinforced concrete under eccentric load, including limb length ratio, reinforcement strength grade, concrete reinforcement ratio, and load eccentricity [12]. At the same time, during the actual test, many interference factors may cause deviation and interference to the conclusions of some tests, and other factors in the process of some tests may also cause errors to the conclusions [13]. In the specific process, the results are directly affected by other reasons, such as the repeatability of the operation, the relatively large investment required in the experiment, and the reading errors of scientific researchers. Therefore, this section will focus on ABAQUS finite element analysis software to illustrate the accuracy and effectiveness of the FEA model constructed in this paper [14].

3. Methodology

3.1. Ontogenetic Model of the Material

3.1.1. Ontogenetic Model of Concrete. In the finite element software ABAQUS, there are two main methods applicable

Table 1: Research content and conclusion.

1	By using the finite element analysis software ABAQUS, the finite element model of reinforced concrete shaped column is established based on the test data in the relevant literature, and the accuracy of the model establishment is analyzed by verifying the load-displacement curve and ultimate bearing capacity obtained after the test
2	On the basis of verifying the correctness of the analytical finite element model, the cross-sectional form of reinforced concrete shaped column is designed, the relevant specimen parameters are set, and the specimen parameters required to be studied under each working condition are determined, and then, the finite element models are established separately to obtain the concrete stress cloud, reinforcement stress cloud, and load-displacement curve under the structural form of eccentricity, loading angle, concrete, strength and reinforcement ratio
3	Compare and analyze the changes of concrete stress cloud, steel stress cloud, and load-deflection curve of the shaped column under different working conditions and the damage pattern of the specimen, respectively, and determine the influence law of the mechanical properties of the shaped column under the above four working conditions and one-way bias pressure

to the simulation of cement as a building material, one is damage plastic simulation, and the other is diffusion crack simulation. It refers to dividing the reinforcement into plain cement and reinforced concrete according to the reinforcement requirements of building components. The modeling and analysis methods and research principles often used by these two forms of concrete are very different. Among them, the diffusion fracture mode is mainly applied to the theory of isotropic plasticity and directional failure strength to describe the nonlinear panel behavior of concrete and is mainly applied to analyze some types of monotonic loading on the vertical deformation of concrete. The research principle of damage plastic mode mainly adopts Kachanov's classical damage mode. The basic operation principle of damage plastic mode is to equivalent the damage degree of concrete material to the form of the most effective area variable. The specific parameters of damage deformation equation can be expressed as follows:

$$D = \frac{\left(A - \bar{A}\right)}{A}. \tag{1}$$

In the formula, A is the effective cross-sectional area of the material before damage. $\bar{A}$ is the effective cross-sectional area of the material after damage.

According to Lemaitre's hypothesis "strain equivalence assumption," the specific damage variables can be expressed as follows.

$$D = 1 - \frac{E'}{E}, \tag{2}$$

where E is the modulus of elasticity of the material before damage and E' is the modulus of elasticity of the material after damage.

Based on Equations (1) and (2), the specific expressions for the damage variables were regrouped by Yu, as shown below.

$$D = 1 - \frac{K}{K_0}. \tag{3}$$

The main structural simulation of buildings in this paper is the failure and plastic simulation of concrete structures in ABAQUS finite element software system. When ABAQUS finite element software is used to simulate the numerical simulation calculation process, it is assumed that the damage plastic model is used for simulation calculation, and the curve of the inherent structure model of cement is shown in Figure 3. The specific calculation method of cement damage and deformation has been explained, and the specific calculation process will not be explained. The corresponding inherent structure relationship of cement is deduced according to the above calculation formula [15].

3.1.2. Principal Structure Model of Reinforcement. The main dimension parameters of reinforcement can be obtained by measuring and determining the corresponding thermodynamic parameters. According to the main structure relationship of steel bar and iron obtained from the experimental results, the simulation curve is shown. As we can see, the main design performance curve of steel includes the following five sections, which can be divided into strength (OA), elastoplasticity (AB), yield (BC), strengthening (CD), and necking (DE) [16].

As shown in Figure 4, if the longitudinal axis f_p is a proportional limit, when the internal stress value of the test mold is insufficient or exceeds f_p, the inherent relationship of the steel at this stage also satisfies Hooke's law. When the internal stress can be removed from the test mold, the internal stress of the measured sample can also return to its original state, which indicates that the steel at this stage belongs to the most ideal elastic deformation stage. When the tensile stress on the specimen is between f_p and f_y, it means that the steel is in the elastic-plastic stage. The longitudinal axes f_y and f_u in the figure represent the yield point and tensile resistance of steel. When the stress of the sample exceeds the f_y value, the sample will also yield. When the pressure of the sample reaches the f_u value, the steel sample will also reach the necked state and then quickly break and lose strength [17].

The stress-strain relationship curve of the reinforcement can be more intuitive to see the intrinsic structure of the reinforcement, although the intrinsic structure of the reinforcement obtained from such a test can be more intuitive and realistic to describe the situation of the concrete at different stages of stress, but because such an intrinsic structure model is more complicated when calculated by ABAQUS

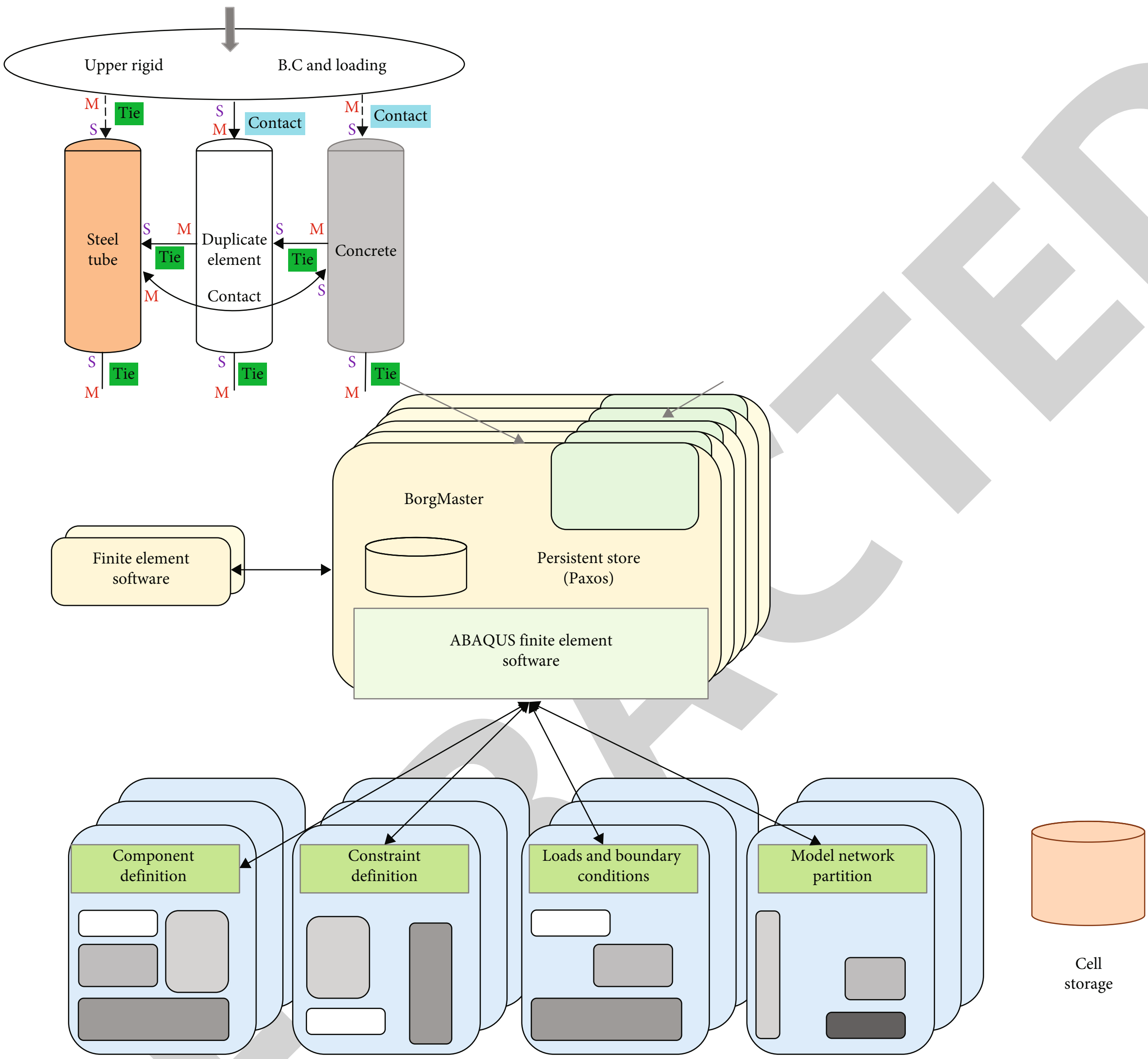


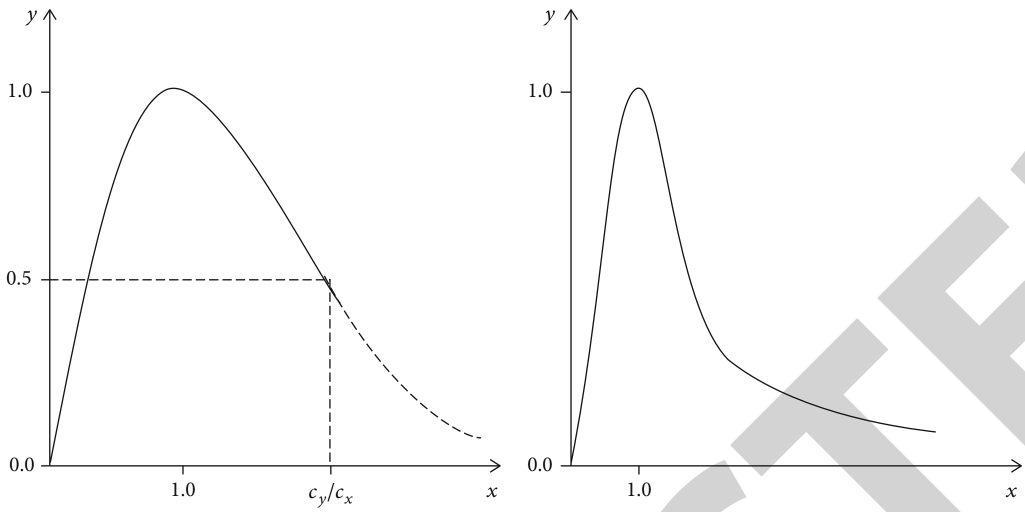
Figure 2: Stress analysis of concrete materials based on finite element analysis.

finite element software, it will lead to a complicated and tedious process of solution, so it is not widely used in the actual analysis [18]. The bilinear model main structure model found by people mainly includes two types [19]. The first type is the most ideal double elastic-plastic main structure model, as shown in Figure 5. (the first type is an ideal elastic-plastic transient model, as shown in (b), which is a transient model obtained without considering the reinforcement effect; The other is the bilinear through reinforcement model. As shown in Figure 5(a), the Mises yield criterion is that under certain deformation conditions, when an equivalent force point exceeds a certain value, it will enter the plastic state from that point [20].

3.2. L-Shaped Anisotropic Section Column Shape Center Formula. Since this paper focuses on the mechanical properties of the prestressed reinforced special-shaped column under eccentric load, the central part of the formwork of the special-shaped column must be determined before the discussion and research. And the displacement and volume of the normal regular section at the center of the shape can be easily expressed. Therefore, people can find and derive the method of describing the static moment, that is, the static moment algebra of each moment component of the heterogeneous section to one axis is equivalent to the static moment of the heterogeneous section to the same axis, and then deduce that the static moment of a heterogeneous section can be described as:

$$S_x = \sum_{i=1}^{n} A_i \bar{y}_i, \tag{4}$$

$$S_y = \sum_{i=1}^{n} A_i \bar{x}_i. \tag{5}$$



(a) (b)

Figure 3: The principal structure relationship curve of concrete.

In Equations (4) and (5), the symbols A_i and $\bar{x}_i$, n, and $\bar{y}_i$ denote the area of a certain rectangular cross-section composing a heterogeneous section and the coordinate position of the shape center of this section, respectively, and n denotes the number of rectangular sections composing a heterogeneous section. Therefore, according to the knowledge of theoretical mechanics, it is known that the specific coordinate equation of the form center of a heterogeneous section in the right-angle coordinate system is shown as follows.

$$\bar{x} = \frac{S_y}{\sum_{i=1}^{n} A_i}, \quad \bar{y} = \frac{S_x}{\sum_{i=1}^{n} A_i}. \tag{6}$$

The cross-sectional form of the shaped column analyzed in this study is L-shaped, and the specific form is as follows. Based on the above calculation method of the formula for the form center of a column with a shaped section, we can derive the specific coordinates of the form center of a column with an L-shaped section as $\bar{x}$ and $\bar{y}$, as shown in the following equation.

$$\bar{x} = \frac{b^2 h + h_f'\left(b_f' - b\right)\left(b_f' + b\right)}{2bh + 2h_f'\left(b_f' - b\right)}, \quad \bar{y} = \frac{bh^2 + h_f' 2\left(b_f' - b\right)}{2bh + 2h_f'\left(b_f' - b\right)}. \tag{7}$$

The limb thickness of the column limb along the x-axis in Figure 6 is h_f', and the limb height is b_f' and the limb height along the y-axis is h and the limb thickness is b

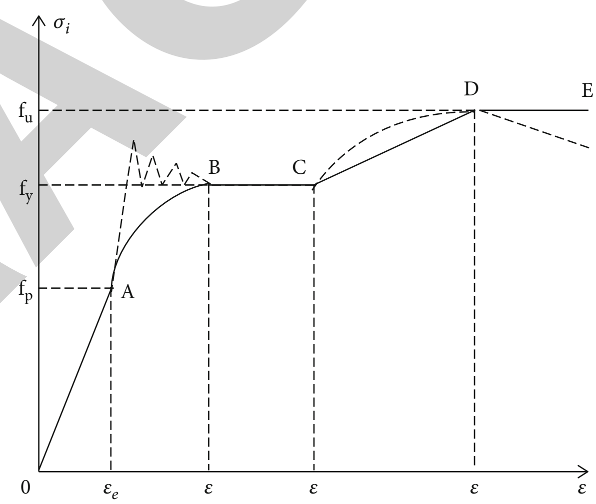


Figure 4: Stress-strain relationship curve for reinforcing steel.

3.3. Citation of Reference Test Data. The parameters of the finite element simulation of the reinforced concrete section column of the L-shaped frame are consistent with the standard data studied in the references, and a measurement sample in the references is selected to simulate and test and analyze the rationality of the internal problems of the simulation test. The longitudinal reinforced concrete structure size is also the same, that is, eight frame cement sizes of HRB 335 with a length of 12 mm are used, and the reinforced concrete ratio is C30, The compressive strength of frame cement is 34.57 MPa.

3.4. Simplification of L-Shaped Column Model. According to the computational mechanical model tested in Ref., we need to simplify the mechanical model of reinforced concrete L-shaped section column under eccentric load.

3.5. Establishment of Finite Element Model. This paper focuses on the dynamic characteristics of eccentrically

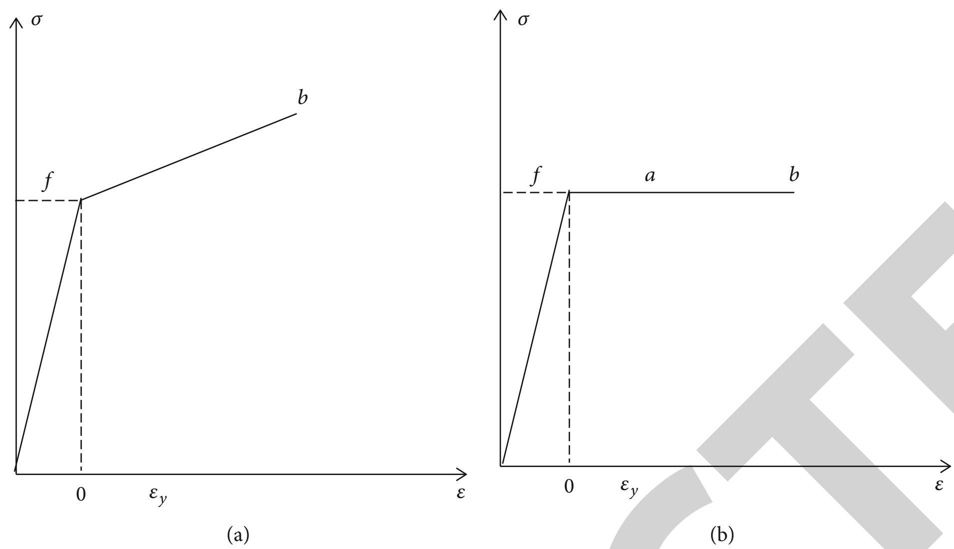


Figure 5: Simplified stress-strain principal structure relationship curve for steel reinforcement.

loaded steel-concrete L-shaped beam. In order to apply load to the L-shaped column, a rigid pad is placed at the top of the L-shaped column, and the load is applied here for control during the simulation of finite element specimens. The purpose is to prevent the stress concentration phenomenon in the subsequent loading process and thus affect the results of the simulation test and also to provide researchers with a more intuitive understanding of the specific shape of the specimen under eccentric loading.

When we build the FEA model (that is, the finite element analysis model) of reinforced concrete L-shaped column for operational analysis, it is assumed in advance that the bonding occlusion between the steel and concrete is good and can work together, and the existence of relative slip between the steel and concrete is also not considered.

3.5.1. Creating Parts and Defining Properties. The concrete L-shaped column, longitudinal reinforcement, hoop, steel plate spacer, and other base components of the reinforced concrete L-shaped column are created separately by using the "Component" module in the finite element analysis software ABAQUS, and then, the "Property" module in the finite element software is used to define the material properties of each part of the specimen (reinforcement, concrete, end, spacer, etc.), which contains the mechanical properties such as density, intrinsic structure relationship, and Poisson's ratio, as shown in Table 2. The relevant data are shown for the plastic loss model of concrete. Finally, after setting, the section properties of each component are assigned to all drawn components, respectively, so that the drawing and section properties of each component are completed.

3.5.2. Definition of Assemblies and Constraints. Assemblies refer to in the "Assembly" module, the L-shaped concrete column, longitudinal reinforcement, hoop, and end components are defined as independent states, and then the components with section properties are placed at the appropriate spatial locations using the "Move," "Rotate,"

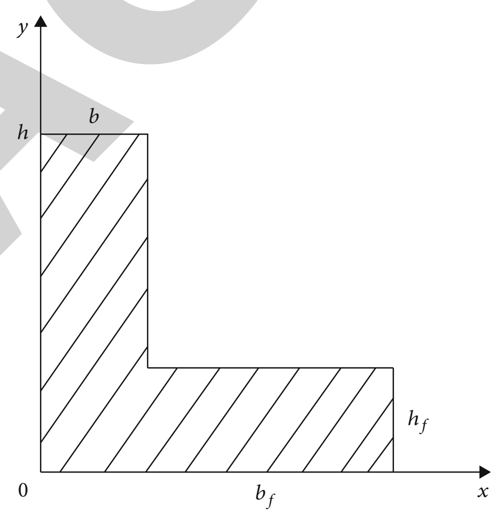


Figure 6: Schematic diagram of L-section coordinates.

and "Array" commands. Then, use the "Move," "Rotate," and "Array" commands to place the parts with section properties in the appropriate space of the reinforced concrete L-shaped column, and then, use the "Merge" command to combine the longitudinal reinforcing bars and hoops to form the reinforcing cage skeleton. The cage skeleton is also set to the independent state.

After assembling each component with properties, the contact and constraint relationships between the components are defined. First, the reinforcing cage skeleton is placed in the concrete L-shaped column as a "built-in area," in which the reinforcing cage skeleton is the "embedded" part, and the whole reinforced concrete L-shaped column model is selected as the main area, thus completing the constraint relationship between the reinforcing cage skeleton and the concrete. This completes the constraint relationship between the reinforcing cage skeleton and the concrete components. Then, the reference point RP-1 is created by using

Table 2: Table of plastic damage model parameters of concrete.

Expansion angle	Eccentricity	K	f_{b0}/f_{c0}	Viscous parameters
30°	0.1	0.667	1.16	0.0005

Table 3: Comparison of bearing capacity of specimens and deflection in columns.

	Experimental value (kN)	Analog value (kN)	Percentage of error (%)
Load carrying capacity	190	191	0.53
Deflection in the column	6.9	9.1	31

the toolbar “Reference Point,” and the displacement load in the reference document is applied to the created reference point RP-1, and the eccentricity of the reference point location is consistent with the eccentricity of the selected verification analysis specimen. Finally, set the constraint relationship between the reference point RP-1 and the rigid end pad as “coupling,” select the reference point as the “subordinate area” of the coupling constraint, and select the upper surface of the rigid end pad as the “main area” of the coupling constraint. This completes the definition of the constraint between the rigid end pad and the reference point RP-1.

Constraints refer to the concrete and rigid end pads which are then set as “binding” constraints, the specific meaning of the binding constraint is to assume that the rigid end pads and the top surface of the L-shaped column are well adhered to each other throughout the process of applying the load, and no peeling will occur between the two, where the top surface of the reinforced concrete L-shaped column is selected as the main surface and the rigid end pads are selected as the slave surface. The upper surface of the reinforced concrete L-shaped column is selected as the main surface and the rigid end spacer is selected as the follower surface.

3.5.3. Setting up Analysis Steps and Output Variables. Set the analysis step in the “Analysis Step” module, create the initial analysis step “Initial” and the analysis step “Step-1,” and then select the Newtonian solution technique method. As the solution technique for model research and analysis, set the initial incremental step, the minimum incremental step, and the maximum incremental step to 0.02, 1×10^{-7}, and 0.5, respectively, in the “Analysis Step,” where the maximum number of incremental steps is set to 20,000 and the total time length of calculation is set to 1. In the analysis step, enter the displacement load to be applied in “Step-1,” and then select “General, Static” type in order.

Finally, variable schemes such as displacement and reaction external force can be selected from “field input and output”. In the simulation analysis process, as long as the input and output variables are reduced, the time limit of the finite element software simulation operation samples can be effectively shortened, and the required calculation result information can be more conveniently obtained in the post-processing process. At the same time, because it is conducive to observing and obtaining the real displacement of the later calculation results, is set to “Set” (Set-1).

3.5.4. Defining Loads and Boundary Conditions. In order to simulate and analyze the full bearing capacity curve of the concrete L-shaped section width beam under eccentric load, the load method of the simulation test adopts the moving load method, and the pressure deflection relationship curve of the concrete L-shaped column is obtained by using the finite element software simulation analysis method. The boundary condition of the bottom surface of reinforced concrete L-shaped column is first defined in the “pressure” model. Finally, it is constrained on the six degrees of freedom of the bottom surface of the concrete special-shaped column to ensure that the concrete special-shaped column is completely stable and the FEA model can operate correctly.

3.5.5. Model Meshing. In the “grid” model, the reinforced concrete L-shaped column formwork is divided into grids, and the overall unit size of the whole model is set to zero zero zero two. However, since the rigid end pad on the upper part of the special-shaped column is not the key object of the simulation analysis, the end pad is set to facilitate loading. The global setting unit size is 1. In order to produce good convergence of the calculation results, the unit form is designed as a regular hexahedron element. Then, the reinforcement cage skeleton is divided into grids, and then, the element form of the reinforcement cage is determined by this method; that is, the reinforcement cage skeleton is determined as the truss element of T3D23D. The unit form of the rigid end pad and the reinforcement is selected as the C3D8R unit form, that is, the solid three-dimensional eight-node linear hexahedron unit form.

3.5.6. Job Submission Analysis and Postprocessing. After the completion of the meshing operation, enter the “Job” module, subsequent analysis each time you need to create a new job and then submit, and finally wait for the computer to complete the analysis job. In the process of submitting the simulation analysis job, you can also see the whole calculation process of the shaped column more intuitively through the “monitoring” function and can also be prompted at any time during the calculation of errors and warning information.

Table 4: Main reasons for the deviation of the test simulation.

1	When the FEA model was established with ABAQUS software, it was assumed that the bond between the reinforcement and concrete was good, and there was no slippage between the reinforcement and concrete before the specimen reached the ultimate bearing capacity, so there was a certain difference between the actual test force and the FEA simulation value and the test value of the reference literature.
2	We use ABAQUS software to simulate the FEA model, and the material's intrinsic structure relationship is applied to the FEA by reviewing other data and doing a lot of experiments with instrumental data, so the intrinsic structure relationship of the FEA simulation may be inaccurate with the experimental intrinsic structure data of the reference literature.
3	The determination of the cube compressive strength of concrete in the test is obtained under more ideal conditions, while in the actual test, there is a certain degree of error in the production of concrete cube specimens, and there may also be a certain degree of human error in the collection and collation of test data, which leads to a certain difference between the data obtained from the ABAQUS finite element simulation and the data from the reference literature.

Table 5: Analysis of results.

Eccentricity	With the increase of eccentricity, the height of the concrete flange compressed area decreases, the distribution of large compressive stress in the corresponding area gradually shifts from the middle of the column to the two ends of the column, and the shape of the large pressure distribution is also concentrated from the high through the column to the two ends of the column, respectively; the eccentricity has little effect on the stress of the reinforcement; the bearing capacity gradually decreases with the increase of the eccentricity, and the ductility has a small increase with the increase of the eccentricity.
Load angle	As the loading angle changes, the neutral axis of the L-shaped column rotates and the pressurized area changes from 0°, 45°, 80°, 100°, and 135° first gradually increases and then gradually decreases; in 80° and 100°, the number of hoops reaching yield is the highest, and the bearing capacity tends to rise first with the loading angle and then gradually decreases at 80° and 100°, reaches the peak, and then gradually decreases.
Concrete strength	With the increase of concrete strength, concrete stress, reinforcement stress, and concrete bearing capacity also increase gently, and the deflection corresponding to the ultimate bearing capacity decreases accordingly.
Longitudinal reinforcement rate	As the diameter of longitudinal reinforcement increases, the reinforcement ratio of longitudinal reinforcement increases, and the load is mostly borne by the reinforcement in the web part of the L-shaped column, and the compressive stress in the flange part decreases accordingly; the ultimate bearing capacity increases, but the corresponding deflection increases instead.

Finally, select the "visualization" template first, and then enter the postprocessing phase. In the postprocessing process, the strain range and stress of any part of the prestressed concrete L-shaped column can be obtained from the results calculated by the finite element software, and the numerical value can be obtained, and then, the load deflection curve of the vertical deformation can be drawn to extract the ultimate bearing capacity of the simulated vertical deformation, and the stress evolution process of the vertical deformation and the stress state of each part can be observed through the stress cloud of the simulated specimen, i.e., position distribution of reinforced concrete L-shaped section column sample. By observing the stress cloud in the simulated specimen, the corresponding pressure distribution of the concrete L-shaped section column specimen can also be observed.

3.5.7. FEA Convergence Criterion. If the result does not exceed the required error range, the iterative operation can still be continued. On the contrary, once the expected error level is exceeded, it can be considered that the FEA iteration result has converged or met the requirements, and the current analysis iteration of the calculation can be terminated, and the FEA result is finally obtained. Assuming the nonequilibrium force after the termination of the i-th iteration, the calculation equation is as follows:

$$\{\psi(\delta_i)\} = \{F(\delta_i)\}\bullet\{R\}. \tag{8}$$

The convergence requirement of the nonequilibrium force criterion can be expressed as

$$\|\{\psi(\delta_i)\}\|_2 \le \alpha_F \|\{R\}\|, \tag{9}$$

where $\{R\}$ is the horizontal vector of the external load, $\{F(\delta_i)\}$ is the nodal force vector obtained by integrating the cell after the i-th iteration, α_F is the tolerance for the convergence of nonequilibrium forces, which usually takes the value range from 0.1% to 5%, and the default value of the finite element software ABAQUS is 0.5%, and $\|\{\psi(\delta_i)\}\|_2$ is the Euclidean parametric number.

4. Section Results and Analysis

The following results analysis and model validation are carried out, mainly using the finite element software ABAQUS simulation analysis of the test piece as the main object of study, based on the parameters obtained from the finite element analysis software operations and the test results data measured in the test in the reference literature to verify whether the model of the simulation analysis is correct.

4.1. Load-Deflection Relationship Curve Comparison and Ultimate Bearing Capacity Comparison. The load-deflection relationship curve of the specimen simulated with ABAQUS finite element software is similar to the load-deflection curve in the reference literature, and the change pattern and distribution of the curve match well, so the intrinsic structure relationship model used in the simulated analysis specimen can be used to reflect the mechanical properties of the test specimen and the trend of internal mechanical changes, but the load-deflection relationship curve is not completely consistent with the test results. In order to ensure the correctness of the simulated specimen, it is necessary to further extract the ultimate bearing capacity of the specimen from the simulated specimen according to the need and then carry out a control analysis, and the proposed specific bearing capacity data are as follows. Table 3 shows the specific bearing capacity data.

It can be seen from Table 3 that the maximum deviation proportion between the bearing capacity of the simulation sample and the reference sample is 0.53% and the maximum deflection deviation proportion in the column is not 31%. Therefore, it can be seen that the error between the finite element simulation results of the two test L-shaped columns and the maximum ultimate strength in the experimental results in the literature is only 10%. This further demonstrates the consistency between the finite element analysis model established by the finite element software and the reference specimen, with good accuracy and precision, which can be used for subsequent analysis.

4.2. Error Analysis. It can be seen from the previous section that the load displacement curve calculated by referring to the load deflection relationship curve in this paper and the model established by the finite element software ABAQUS is only approximate to the ultimate bearing capacity of two reinforced concrete L-shaped columns, but not quite the same. Through analysis of the main factors of test simulation error, the following aspects are shown in Table 4.

4.3. Analysis of Results. In this chapter, the middle limb L-shaped prestressed concrete column in the literature is taken as the basic structure, and the validity of its construction mode is verified through the simulation of the test structure by the large-scale commercial general nonlinear finite element software ABAQUS. On the basis of correctly studying the concrete stress-strain curve and the reinforcement stress-stress curve, the four influence factors of eccentricity and loading angle are also studied. Based on the research results of the four influencing factors of concrete tensile strength and longitudinal reinforcement ratio and longitudinal reinforcement ratio, there are three aspects: concrete stress, reinforcement stress, and load-deflection curve. The following results were obtained, shown in Table 5.

5. Conclusion

In this paper, a finite analysis model is constructed according to the characteristics of sludge concrete. Analyze the stress situation of the structure under the least favorable load to obtain the strength index that the interlayer bonding layer material and the asphalt mixture should achieve. Comprehensive calculation results show that the shear breakage between layers is the main form of breaking; in the case of horizontal load and overload, the horizontal shear stress and longitudinal shear stress between the layers should be considered, and the angle of the shear stress force is between the longitudinal bias of 15-30 degrees. The stress cloud and load-displacement curve of steel reinforcement under the structural forms of concrete stress cloud, eccentricity, load angle, concrete strength, and reinforcement ratio are obtained, and the influence law of the mechanical properties of the special-shaped column under these four working conditions and unidirectional bias pressure is determined.

Data Availability

The labeled datasets used to support the findings of this study are available from the corresponding author upon request.

Conflicts of Interest

The authors declare that there are no conflicts of interest.

References

[1] X. Bi, Y. Yang, and Z. Zhang, "The mechanical analysis of cement concrete surface layer with load - transfer bar for joints based on computer," *Journal of Physics: Conference Series*, vol. 1744, no. 2, p. 022084, 2021.

[2] S. Samarakoon and B. Hodne, "Parametric study of different unbonded tendon layouts in pre-stressed concrete flat plates," *International Journal of Concrete Structures and Materials*, vol. 14, no. 1, pp. 12–33, 2020.

[3] Z. Si, Y. Li, L. Wen, and X. du, "Simulation and analysis of the temperature field and the thermal stress of an inverted-siphon concrete structure based on the contact friction element," *KSCE Journal of Civil Engineering*, vol. 24, no. 8, pp. 2449–2457, 2020.

[4] Z. Z. Wang, W. M. Gong, G. L. Dai, and L. F. Liu, "Analysis of bond behavior of FRP-confined concrete piles based on push-out test," *Advances in Materials Science and Engineering*, vol. 2020, 17 pages, 2020.

[5] M. Da and Z. Su, "Stress and deformation analysis of concrete-facing sand–gravel dam based on inversion parameters," *Geotechnical and Geological Engineering*, vol. 39, no. 2, pp. 1399–1408, 2021.

[6] A. Ghasemipanah and R. Z. Moayed, "Analysis of concrete piles under horizontal and vertical simultaneous loading with P-Y method and finite element analysis," *Geotechnical and Geological Engineering*, vol. 39, no. 8, pp. 5857–5877, 2021.

[7] D. S. Wang, K. J. Yang, H. Yang, and P. P. Zhang, "Computational analysis of laser cladding of preset MCrAlY coating based on ANSYS ii-stress field," *Materials Science Forum*, vol. 1020, no. 4, pp. 148–156, 2021.

[8] J. Jiang, X. Huang, and Z. Wang, "Finite element simulations of eccentric compression performance for GFRP confined reinforced concrete long columns," *Society of Photo-Optical Instrumentation Engineers (SPIE) Conference Series SPIE*, vol. 15, no. 21, pp. 25–41, 2021.

[9] J. Watkins, M. Carlson, and K. Shan, "Performance portable ice-sheet modeling with MALI," *Computer Science [J]*, vol. 10, no. 8, pp. 18–30, 2022.

[10] N. Revanna, C. Moy, and T. Krevaikas, "Verifying a finite element analysis methodology with reinforced concrete beam experiments," *Applied Mathematics and Applied Physics (English)*, vol. 8, no. 11, pp. 2549–2556, 2020.

[11] M. H. Lim and C. Lee, "A study on the heat and stress evaluation of reinforced concrete through high-frequency induction heating system using finite element techniques," *Sustainability*, vol. 13, no. 11, p. 6061, 2021.

[12] R. Jakubovskis and G. Kaklauskas, "Damage of bond in reinforced concrete: a detailed finite element analysis," *Structural Concrete*, vol. 22, no. 6, pp. 3228–3240, 2021.

[13] S. P. Parida, P. C. Jena, and S. R. Das, "Comparative stress analysis of different suitable biomaterials for artificial hip joint and femur bone using finite element simulation," *Advances in Materials and Processing Technologies*, vol. 2, pp. 1–16, 2021.

[14] S. A. Al-Taan and A. A. Abdul-Razzak, "Geometrical and material nonlinear finite analysis of fiber reinforced concrete slabs," *IOP Conference Series: Materials Science and Engineering*, vol. 978, no. 1, pp. 012041–012056, 2020.

[15] X. Long, M. Mao, C. Lu, R. Li, and F. Jia, "Modeling of heterogeneous materials at high strain rates with machine learning algorithms trained by finite element simulations," *Journal of Micromechanics and Molecular Physics*, vol. 6, no. 1, pp. 2150001–2150117, 2021.

[16] H. Chen, B. Xu, J. Wang, T. Zhou, X. Nie, and Y. L. Mo, "Parametric analysis on compressive strain rate effect of concrete using mesoscale modeling approach," *Construction and Building Materials*, vol. 246, p. 118375, 2020.

[17] Y. Niu, Y. Wang, and Y. Tang, "Analysis of temperature-induced deformation and stress distribution of long-span concrete truss combination arch bridge based on bridge health monitoring data and finite element simulation," *International Journal of Distributed Sensor Networks*, vol. 16, no. 10, 2020.

[18] F. Yu, Z. Song, I. Mansouri, J. Liu, and Y. Fang, "Experimental study and finite element analysis of PVC-CFRP confined concrete column - ring beam joint subjected to eccentric compression," *Construction and Building Materials*, vol. 254, pp. 119081–119131, 2020.

[19] V. K. Kytinou, C. E. Chalioris, and C. G. Karayannis, "Analysis of residual flexural stiffness of steel fiber-reinforced concrete beams with steel reinforcement," *Materials*, vol. 13, no. 12, p. 2698, 2020.

[20] E. M. Bouhjiti, J. Baroth, F. Dufour, M. Briffaut, and B. Masson, "Probabilistic analysis of concrete cracking using stochastic finite element methods: application to nuclear containment buildings at early age," *Materials and Structures*, vol. 53, no. 4, pp. 41–55, 2020.