Application and Prospect of Analysis of Structures Based on Finite Element Method

Chen Liang *

Department of Civil Engineering, Qingdao University of Technology, Qingdao, Shandong, China

* Corresponding Author Email: 100480@yzpc.edu.cn

Abstract. The structural requirements of buildings are becoming more stringent and sophisticated as a result of the quick advancement of science and technology. Due to the limitations of the traditional structural mechanical analysis methods, the calculation results often have large deviations from the actual, and therefore cannot simulate the structural forces and displacements in a more realistic way. The emergence of the finite element method (FEM) makes these complex and irregular problems can be solved well. This paper analyzes the specific applications of FEM and its effectiveness on three typical space structures, namely, space trusses, foundation pits, and steel buildings. The results show that in the analysis of large trusses, the three-dimensional nonlinear finite element analysis using relevant analysis software has good accuracy in displacement calculation, seismic simulation, static force analysis, and structural optimization simulation. In the application of foundation pits, this method can also well simulate the force and deformation characteristics of soil under different working conditions. In steel structure, this method can be applied to steel design, steel slip, and steel top plate force analysis. By discussing the finite element method and related software applications, this paper expects to provide ideas for the analysis of some complex problems in the civil engineering industry and to provide some predictions and suggestions for further advancements.

Keywords: Finite element method; structural analysis; nonlinear analysis; spatial modal.

1. Introduction

As Scientific and technological advancements continue to progress, the structural forms in engineering construction are also developing and progressing, and the difficulty of their structural analysis is also increasing, resulting in many complex space structures, such as large space trusses, or problems related to the force analysis of continuous media, such as for the analysis of foundation soil. Traditional mechanical analysis has gradually failed to meet the requirements, finite unit method analysis may be a suitable choice. Since its introduction, the finite unit method has undergone a long period of development, with the continuous expansion of the scope of application, the problems to be analyzed have become large and complex, and gradually evolved some of the "semi-analytical" numerical methods such as the finite strip method, combined strip - element method, finite line method, boundary unit method, etc. The applications also range from elastic plane problems to elastic-plastic, composite space, plate and shell problems gradually; from static equilibrium problems to stability problems; from single linear problems to nonlinear problems; from single medium to multi-media coupling. Therefore, the application of finite element analysis may have wider applicability.

Previous research on finite element methods has mainly focused on the study of mathematical theory or its application to different industries, lacking analysis for the civil engineering industry. However, for specific structures in civil engineering, only theoretical analysis is not enough, and the establishment of an efficient, convenient, reasonable, and easier-to-calculate finite element analysis model is the key focus of the work. Although there are some recent specific finite element modeling designs, these emerging designs need to be analyzed and selected for reasonableness to show whether they are instructive for reality. In contrast to other studies and reviews, this paper shows the specific work that can be performed by this method, analyzes its plausibility and efficacy, and summarizes its multiple applications in the civil engineering industry.

This paper focuses on two parts, namely the principles of finite element analysis (FEM) and the analysis of its application in three types of building structures. Firstly, the principle and development of FEM will be discussed, and the steps of its use in civil engineering. Secondly, specific examples of the application of the method will be shown to demonstrate the versatility of its application and to evaluate the soundness of the results.

2. FEM

2.1. Basic concept of FEM

The FEM is a highly effective and widely used method for numerical calculation. The field of scientific computing frequently involves solving differential equations of various types, and it is generally difficult to obtain analytical solutions for these differential equations. However, a computer-aided solution program can be designed after discretizing the differential equations using the FEM, which can be interpreted as replacing complicated problems with simpler ones and then try to find solutions. Specifically, the solution is derived by first solving the requirements (such as the equilibrium conditions of the structure) for the overall fulfillment of this domain, which is seen as the solution domain consisting of numerous small interconnected subdomains called finite elements. It is worth stating that although the results obtained are approximate solutions rather than accurate solutions, the results are in relative proximity to the real situation in reality.

The basic principle of the FEM is to discretize the continuous solution domain into a set of combinations of cells and to represent the unknown field function to be solved on the solution domain in pieces by the approximate function assumed in each cell, which is usually expressed by the numerical interpolation function of the unknown field function and its derivatives at each node of the cell [1]. Thus, a continuous infinite degree of freedom problem becomes a discrete finite degree of freedom problem.

FEA refers to the simulation of real physical systems (geometric and loading conditions) using mathematical approximations. Using simple and interacting elements, it is possible to approximate a real system with an infinite number of unknowns. Given that most real-life problems are difficult to obtain accurate analytical solutions, simulations using FEM are not only adaptable to a variety of sophisticated shapes but also have high computational accuracy, thus becoming a feasible and effective approach to engineering analysis.

2.2. Developing History of FEM

Finite elements are those separate components that can collectively describe an actual continuous field. The idea of finite elements was created and adopted for centuries ago, for instance, when finding the perimeter of a circle, a polygon can be used to approximate the circle and thus find a relatively accurate answer, yet it wasn't until lately that it was offered as a methodology. The development of the FEM can be divided into three stages: proposal (1943), development (1944-1960), and refinement (1961-1990). In order to solve the St. Venant's torsion problem, Courant used the piecewise continuous function defined on the triangular area and the principle of minimum potential energy for the first time in the 1940s. 1943 marked the birth of the FEM with Courant's mathematical paper "Variational Decomposition Method for Equilibrium and Vibration Problems" and the major breakthrough in engineering by Aceras. In the early stage of development of the FEM (1944-1960), The initial simpler algebraic expression of the FEM was derived, and the research on the segmentation of units and the selection of its type was carried out gradually, also significant progress and breakthroughs were made in the study of convergence of solutions. The first successful attempt of modern FEM was in 1956, when Turner, Clough, and others extended the application of steel frame displacement method to the elastic mechanic plane problem in the analysis of aircraft structures, and gave the correct answer to the plane stress problem with triangular cells. Clough coined the phrase "FEM" for the first time in 1960, which marked the end of the early development stage of FEM. The development of the FEM in its refinement stage (1961-1990) was characterized by, first, the establishment of a rigorous mathematical and engineering foundation; second, the extension of the application to fields other than structural mechanics; third, the further study of convergence and the formation of a systematic error estimation theory; and fourth, the development of the corresponding commercial software packages. It is worth mentioning that Feng Kang's paper "Difference format the based on variational principle" published in Applied and Computational Mathematics in 1965 is a symbol of the systematic founding of the FEM in China independently of the West [2].

As a result of its efficiency, practicality and effectiveness, the FEM has gained a great deal of attention among scientists conducting mechanical research. After just a few decades of efforts, along with the rapid progress and popularity of computer technology, the application of FEM has rapidly spread from strength analysis as well as structural engineering calculations to almost every field of science. At the same time, researchers have developed finite element simulation software such as ANSYS and COMSOL (used in the pre-design phase instead of experimental testing, which can greatly save costs), making FEM a colorful, widely used, practical, and efficient numerical analysis method.

2.3. Steps of using FEM in structural analysis

Partitioning: The region that has to be considered is segmented and partitioned into a limited number of elements. Although the geometry of the elements (units) is arbitrary in theory, tasks in two dimensions often utilize triangular or rectangular models as their basic elements, while problems in three dimensions may involve tetrahedral or polyhedral units, etc. Each element's vertex is also referred to as a node.

Elements(units)analysis: Through the process of piecewise interpolation, a linear interpolation function is constructed by expanding a function to be solved at whatever position in a discrete unit with the unit's shape function and its' value at the discrete grid point.

Solving approximate variational equations: The function of the entire continuum can be roughly represented by the ensemble of all unit field functions. The numerical resolution of the FEM is accomplished by solving this discrete system of equations in a sensible manner, which is based on the energy equation or the weighted residual equation, and which establishes an algebraic system of equations with limited quantities of the variables to be determined.

3. Application of FEA

3.1. Application of FEA on Trusses

Using the FEM, Chaojiang Fu [3] proposes a new solution format for the large displacement static analysis problem of space truss structures. Rather than relying on nodal displacements to characterize the situation, the format instead uses nodal positions. The strain is calculated directly by using the Cartesian coordinate system in fixed space, and a simple engineering strain metric is used. The calculation results of the algorithm are compared with those obtained by the arc length method, and satisfactory results are obtained, showing that the method has good accuracy.

Sucheng Rui [4] analyzed the results by modal calculations, including frequencies and vibration patterns, and used ANSYS software to solve the modal solutions for large complex space grid structures such as large stadiums, as shown in Fig. 1 and Fig. 2, to obtain the first 10 orders of intrinsic frequencies and vibration patterns of the large stadium structures. The analysis is of great significance for the calculation of seismic and earthquake resistance, and provides a theoretical basis for the design of large complex space grid structures, as well as a valuable reference value for the FEA of large complex space grid structures.

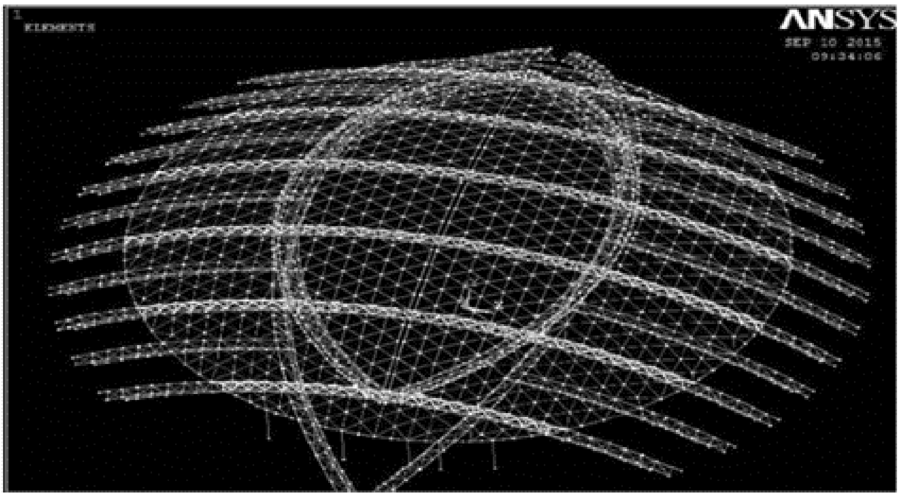

Fig 1. Space grid unit [4].

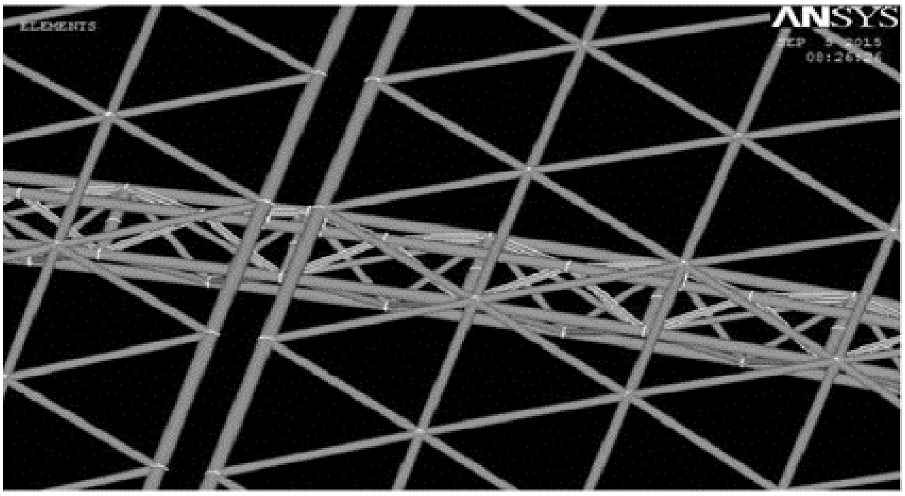

Fig 2. Partial space grid unit [4].

Liquan Xiong [5] used the finite element software ABAQUS to perform a nonlinear 3D analysis of four contrasting indoor Ra CS beam column connections, and compared the analysis results with experimental results. A series of numerical simulation techniques are investigated, such as selecting cell types, defining material models for steel and concrete, and setting solution control options and solution modules. The results show that ABAQUS can simulate the behavior of the RCS beam-column connection under static loading through a reasonable choice of parameters.

Jiaheng Li [6] performed an analysis for a typical truss structure by using FEA software ABAQUS. Firstly, the finite element model of the space truss structure was established, and the stiffness analysis and modal analysis were carried out, and the first fifth order inherent frequency was obtained, and the combination of finite element and space truss structure analysis was well carried out.

Qimiao Xie [7] uses the FEM to analyze and solve the truss space grid structure. A spatial structure made up of rods and members connected by nodes in a certain configuration is known as a space grid structure. Usually, the number of rods and nodes of a space grid structure is huge and it is a kind of high super-stationary structure, so the FEM is used for numerical simulation analysis.

Jinbiao Liu [8] designed the three-dimensional structure of the truss support shell through ProE, and used ANSYS Workbench software to perform strength, stiffness and stability analysis of the support shell, and continuously optimized its structure so that its strength, stiffness and vibration resistance could meet the design requirements; it was processed and manufactured, and the real load engineering application could meet the requirements. Therefore, the structural optimization analysis method not only shortens the product development cycle, reduces the production cost, but also has a relatively high simulation accuracy, which provides a certain theoretical basis for the design of similar products.

3.2. Application of FEA on Foundation Pit

Double-row pile support is a relatively common form of foundation support. Fengyuan Wu [9] used the large finite element software ABAQUS to establish three-dimensional solid units, as shown in Fig. 3, and calculate the double-row pile support system to study the deformation characteristics of the structure and soil under different working conditions. The Mohr-Coulomb principal structure relationship in ABAQUS was used and the ground stresses were balanced before excavation of the soil. The FEA and calculation were carried out for a double-row pile support, and the reasonableness of the calculation results such as horizontal displacement of the row pile support structure and ground settlement were analyzed to verify the feasibility of ABAQUS in solving the problems of foundation pit excavation and double-row pile support, in order to guide the double-row pile foundation support in practical engineering.

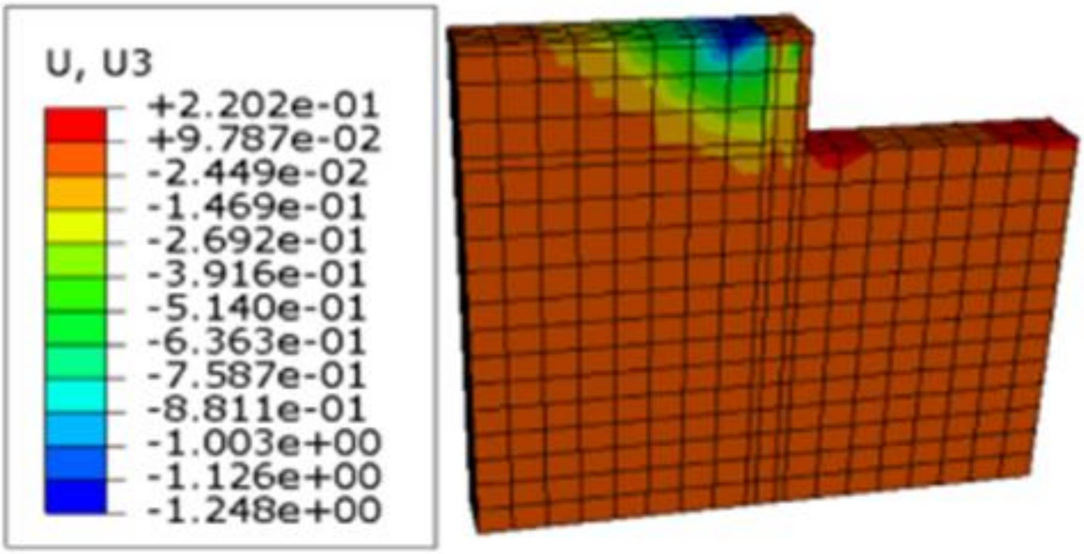

Fig 3. Vertical displacement after excavation of foundation pit [9].

Yanhui Zhang [10] used Geo Studio finite element software to analyze the designed foundation pit project, giving the stress-strain characteristics of the foundation pit soil, and analyzed that the stress at the end of the support pile is most concentrated at the junction of the strongly weathered rock layer and the medium weathered rock layer, and the plastic zone is easy to appear near the top of the pit by the load. The calculation results and monitoring data are consistent with each other, and the basic deformation trend is generally consistent with the actual situation.

With the increasing development of underground space and the increasing number of foundation pit projects, the leakage of foundation pit envelope often occurs to different degrees, and it is especially important to predict and deal with the leakage of envelope before excavation of foundation pit. Zhihua Luo [11] combines engineering cases of leaking pits, analyzes the water level change of pumping test of leaking pits, and further uses finite element simulation to analyze the leakage of pits, and gets the analysis technical means to study the leakage of pits combined with FEM, which provides guidance and reference to assist in judging the leakage of pits.

3.3. Application of FEA on Steel Structures

In order to avoid the interference of the steel structure foundation short column steel bar bend with the anchor bolts of column footing. Lepeng Wang [12] carried out finite element simulation of the angle and length of the base short column steel bar bend in order to avoid the interference of the steel structure foundation short column steel bar bend on the column foot anchor bolts, and derived the maximum stress and deformation of the steel bar as shown in Fig. 4, and combined with the relevant domestic and foreign literature and codes, proposed the design suggestions of the base short column steel bar bend. The results show that: because the short column is a compression bending member, the top steel bar bend of the short column is subjected to small forces, and the angle and length of the bend are not effective in restraining the deformation of concrete. In order to avoid the collision problem and not to violate the code requirements, the size and positioning of the reinforcement and anchor bolts should be checked in the design stage to deal with the collision problem in advance.

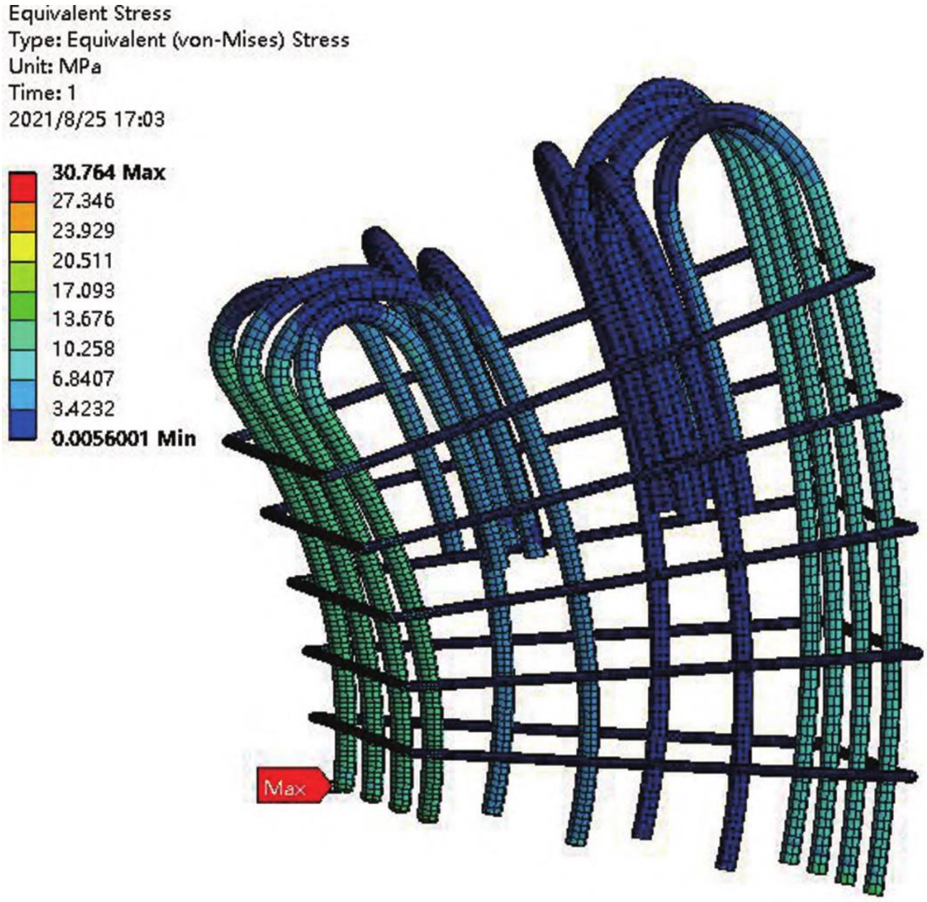

Fig 4. Schematic diagram of maximum stress and deformation of steel bars [12].

In order to reflect the stress-strain situation in the sliding process of the large-span steel structure more realistically. Zerong Lai [13] conducted simulation analysis of different stages of the cumulative sliding process of the steel structure through finite element software. The results show that the maximum stress area of the steel structure during the sliding process is located on the local range of the track around the loading point, and the maximum stress of the double-piece track beam system is larger than that of the single-piece track beam system; the maximum displacement area is located at the direct connection point between the track and the mesh shell, and the maximum horizontal displacement of the double-piece track beam system is smaller than that of the single-piece track beam system. In the process of slippage, each span of track beam shows the trend of strain change which decreases first and then increases, but it is much lower than the warning value. On this basis, this study proposes a safe and effective sliding construction scheme for large-span steel structures, which provides reference and reference for similar construction projects in the future.

The top tightening surface of the steel connection plate directly affects the node load transfer and the structural safety and service performance. In order to study the stress distribution law on the contact surface of the nodal joint plate. Yang Ya [14] used ABAQUS finite element software to simulate and analyze the stress on the contact surface of the nodal joint plate with the uncontacted length of the edge of the joint plate a0, the gap width of the edge of the joint plate δ0, and the real contact area ratio A_r / A_0 as the design parameters. The analysis results show that with the increase of the load, the uncontacted length of the edge of the joint plate and the edge gap width gradually decrease, and the contact area ratio gradually increases, and when the contact area ratio of the joint plate is greater than 70%, the decrease gradually decreases; with the increase of the load, the maximum stress ratio of the contact surface gradually decreases, and the larger the initial contact area of the contact surface, the smaller the maximum stress ratio of the contact surface, and the more uniform the contact surface stress is. tends to be more uniform.

4. Future Outlook

4.1. FEA Software

Large space trusses inevitably have to consider the influence of space fluid on structural stability as well as stiffness. Since the research and development of FEA program for solving non-solid mechanics and cross-discipline has been researched and developed for decades, the FEM and software for solving solid mechanics have been relatively mature, and now the frontier problems of research are non-solid mechanics and cross-discipline problems such as space fluid dynamics, compressible and incompressible fluid flow. Since this software requires mathematical and physical analysis to be powerful, it is more difficult to develop, so we hope to produce software for analyzing fluid and other non-solid mechanics and cross-discipline.

4.2. Plate building blocks

Currently, with the rapid development of the application of big data and machine learning techniques in the field of plate components, non-physical methods or artificial intelligence algorithms provide a new data-driven idea for the study of plate deformation and plate shape control. When using data-driven intelligent algorithms to analyze plate deformation control problems, the control system does not use any mathematical and physical models and a priori theoretical knowledge, but only the processing and analysis of data to achieve the establishment of the control cross-section. However, it may be accompanied by problems such as slow convergence into local minima. Therefore, by establishing a model based on FEM to analyze the actual problem first, the obtained analysis results can be used as training samples of the intelligent algorithm can effectively improve the learning speed and increase the feasibility of the algorithm. Therefore, the organic integration of finite element models and data-driven intelligent algorithms will be more beneficial to solve the complex plate shape control problems in the field.

5. Conclusion

The FEM has developed rapidly in recent years in different problems, providing the prerequisite and basis for subsequent research in related fields. The full paper introduces the principle method of FEM and summarizes and analyzes its application in the field of civil engineering.FEM presents a new solution method for the analysis of 3D modeling of space truss structure in spatial problems. Utilizing FEA software such as ABAQUS, the 3D nonlinear analysis of mega space truss structure can accurately find its intrinsic frequency and vibration pattern, and also accurately calculate the displacement, etc., which facilitates the design optimization in reality. This facilitates the design optimization of the structure in reality. In the analysis of foundation pits, FEA can calculate the stress-strain characteristics of the foundation soil more accurately, which can guide the design of foundation support and production in practice. Although there are various FEA software, they all lack the analysis of spatial fluid dynamics. In addition, with the rapid development of big data and machine learning technology, how to integrate them with FEA software has become a new development direction. It is expected that in the future, there will be an internationally used software containing interdisciplinary analysis such as fluid dynamics, combined with big data analysis and simplifying the operation to facilitate people's use, which will eventually make the application of FEM more extensive and accurate. This paper makes some suggestions on the wide application of FEM in civil engineering and the further development trends.

References

[1] Yu, Dehao. Finite elements, natural boundary elements and Sim-geometric algorithms - important contributions of the Feng Kang school to the development of computational mathematics[J]. Higher Mathematics Research,2001(04):5-10.

[2] Feng Kang, the founder of Chinese computational mathematics[J]. Computational Mathematics,20 20,42(03):258-259+257.

[3] Fu Chaojiang, Pan Qinfeng. Nonlinear finite element solution format for large complex space truss structures[J]. Journal of Fujian Engineering College, 2012, 10(01):1-4

[4] Rui Su-Cheng, Mo Wei-Feng, Tang Jun-Fu, Guo Xiao-Zhe, Hua Yifan. Finite element modeling and modal analysis of space truss structure[J]. Fujian Building Materials, 2015(12):7-9.

[5] Xiong Lizhong, Fu Yanan, Mao Haitao, Men Jinjie, Guo Zhengchao, Wang Hailong. Nonlinear finite element analysis of nodes in reinforced concrete column-steel beam (RCS) combinations considering spatial constraints [J]. Structural Engineer, 2017, 33(06):22-28.

[6] Li Jiaheng. Finite element analysis of space truss based on ABAQUS [J]. Electronic Fabrication, 2017(17):48-49+83.

[7] Xie Qimiao. Approximate finite element model analysis of spatial mesh structure[J]. Sichuan architecture, 2018, 38(04):208-209+212.

[8] Liu Jinbiao. Structural optimization design of support shell based on finite element analysis [J]. Mechanical Engineering and Automation, 2015(05):48-49+52.

[9] Wu Fengyuan,Ji Xuezhuang,Zhao Zihao,Zhang Yichao. ABAQUS-based finite element analysis of deformation of double-row piles and perimeter soil of piles [C]. Proceedings of the 19th Annual Shenyang Scientific Conference. [publisher unknown],2022:373-377.

[10] Zhang Yanhui,Zhang Hongqiong. Research on finite element analysis of foundation pit design[J]. Juye,2022(05):176-178.

[11] Luo C. H. Finite element analysis of foundation pit leakage based on pumping test[J]. Engineering Quality,2021,39(06):65-68.

[12] Wang Lepeng. Finite element analysis and design suggestion of steel structure foundation short column steel bar bending hook[J]. Fujian Architecture,2022(04):50-53.

[13] Lai Zerong. Study on cumulative slip process of large span steel structure based on finite element analysis[J]. Guangzhou Architecture,2021,49(05):25-28.

[14] Yang Y, Tang X R, Tang L Y. Finite element analysis of contact surface performance of steel connection plates[J]. Journal of Changzhou Institute of Technology,2020,33(06):32-36.