Application of Optimized Finite Element Algorithm in Structural Mechanics Simulation System

Jie Wang* and Nana Liu

{*Corresponding author: wangjieinfo@163.com}

{545195968@qq.com}

Chongqing College of Architecture and Technology, Chongqing, China

Abstract. The experimental research on dynamic systems using digital computers has achieved good results. However, for some complex dynamic systems, it is often difficult to express solutions and analysis using mathematics or calculations, and the program is complex and not very intuitive. This has led to difficulties in dynamic experiments in certain fields. There are many similarities between power system and mechanical system, so it must use circuit theory, Network theory, Cybernetics and field theory to analyze and experimentally study its dynamics. Among the commonly used finite element analysis methods, this article discusses a method for solving structural force monitoring problems. Compared to conventional methods, this method does not rely on the increase of grid points in the area around the drill bit, and can adjust the number of grid points to any number, effectively solving numerical simulation problems in large-scale and multi-point situations. The life and death element method can effectively analyze the stress of spatial structural stress systems and has high application value (in the detection of the ratio of life and death element method to design strength, the data is 43.19%, while the actual detection is 41.10%, which is relatively close).

Keywords: Structural Mechanics Simulation System, Finite Element Algorithm, Computer Simulation Analysis, Modern Engineering

1 Introduction

The use of digital computers for testing the power system has achieved good results. However, for some complex dynamic systems, their solutions and analysis are often difficult to express using mathematics or computers, and the process is very complex and not intuitive enough. This has brought difficulties to dynamic experiments in some regions. There are many similarities between motor system and mechanical system, so it is very necessary to use circuit, network, Cybernetics, field theory and other theories to carry out dynamic analysis and experimental research on them.

For the monitoring and research of mechanics, scholars have long made relevant research. Moghadam P Z used multi-dimensional computer analysis to create the first

ICSETPSD 2023, November 17-18, Coimbatore, India

DOI 10.4108/eai.17-11-2023.2342781

interactive terrain map based on MOF. He established a high-throughput model of 3385 MOFs for the first time internationally, which includes 41 different network structures. Secondly, he proposed a set of unconstrained machine learning methods to automatically predict the mechanical properties of metal organic framework Strength of materials. On this basis, he used molecular dynamics simulation methods to conduct a detailed analysis of the crystallization pressure in different regions of the hypertonic space. The integrated mechanical screening method developed by him can effectively improve the mechanical properties of materials and provide a basis for optimizing their mechanical properties [1]. Deringer V L introduced "MeshGraphNets", a grid based simulation learning system that utilizes graph neural networks. By learning the Deringer VL language, the relevant information of the grid graph can be transmitted to subsequent simulation programs, thereby fine-tuning the dispersion of the grid graph. The research results show that the established mathematical model can well simulate the dynamic characteristics of various Physical system, such as aerodynamics, structure and human tissue. This method has good adaptability and can achieve autonomous learning for more complex states by analyzing dynamic characteristics. This method is also highly efficient, and its execution speed is improved by 1-2 orders of magnitude compared to using simulation. This method expands the scope of problems that neural network simulators can solve and hopes to improve the performance of complex scientific simulation problems [2]. Younesian D conducted a comprehensive review of different elastic and viscoelastic fundamental theoretical models in oscillating systems. This review covers from the simplest foundation models to the most complex foundation models, and comprehensively tracks recent theories on the topic of mechanical foundations. He thoroughly discussed why each theory was developed, what limitations each theory contained, and which methods were applied to eliminate these limitations. In addition, he briefly reviewed the corresponding theories about structures supported by this foundation. Subsequently, he introduced popular solutions. Finally, he reviewed several important practical applications related to linear and nonlinear foundations. Younesian D provides a detailed theoretical background, and also provides physical understanding of applications in Structural mechanics, nanosystems, biological equipment, composite structures and aerospace based mechanical systems on the basis of different types. He finally proposed a new idea of an intelligent foundation based on nanogenerators, which can be used for energy collection and self-powered sensing applications in future smart cities [3]. Although the above studies have made contributions to the study of mechanics, the research on computer simulation systems for mechanics is not in-depth enough.

This paper discusses the Structural mechanics simulation system based on the optimized finite element algorithm. The technology of this system is more advanced, and the accuracy of data detection is higher than other current technology systems.

2 Finite Element Algorithm and Structural Mechanics Simulation System

2.1 Structural Mechanics Simulation System

The principle and composition of the computer testing system for structural dynamics simulation testIn the dynamics simulation test, various dynamic responses of the structure (acceleration, velocity, displacement, force, sound and other physical quantities) are converted into dynamic electrical signals through sensors. After being adjusted and amplified, it is input into a computer testing system, which controls the collection, processing, and result output of dynamic signals [4-5]. The characteristics of dynamic test are high frequency requirements and large data flow, especially for dynamic tests conducted on large structures [6-7].

(1) Divided by function

The computer dynamic testing system consists of the following three main parts (as shown in Figure 1):

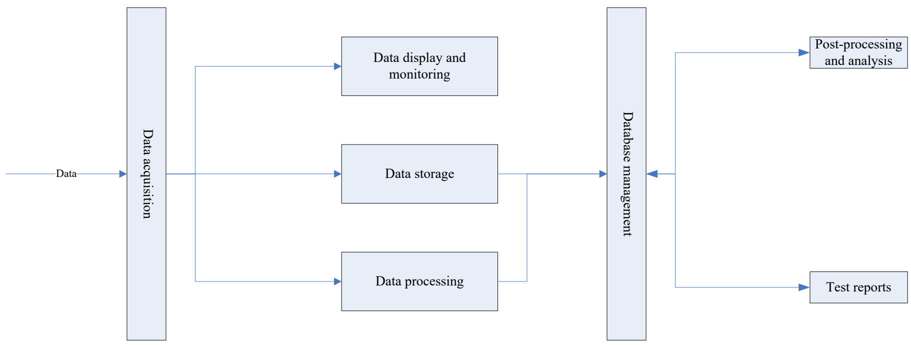

Fig.1 Functional schematic diagram of the computer testing system for dynamic simulation experiments

a. Data collection and transmission;

b. Data processing and storage;

c. Result output and management

(2) Divide according to structural composition

The computer dynamic testing system can be divided into the following three major parts (see Figure 2).

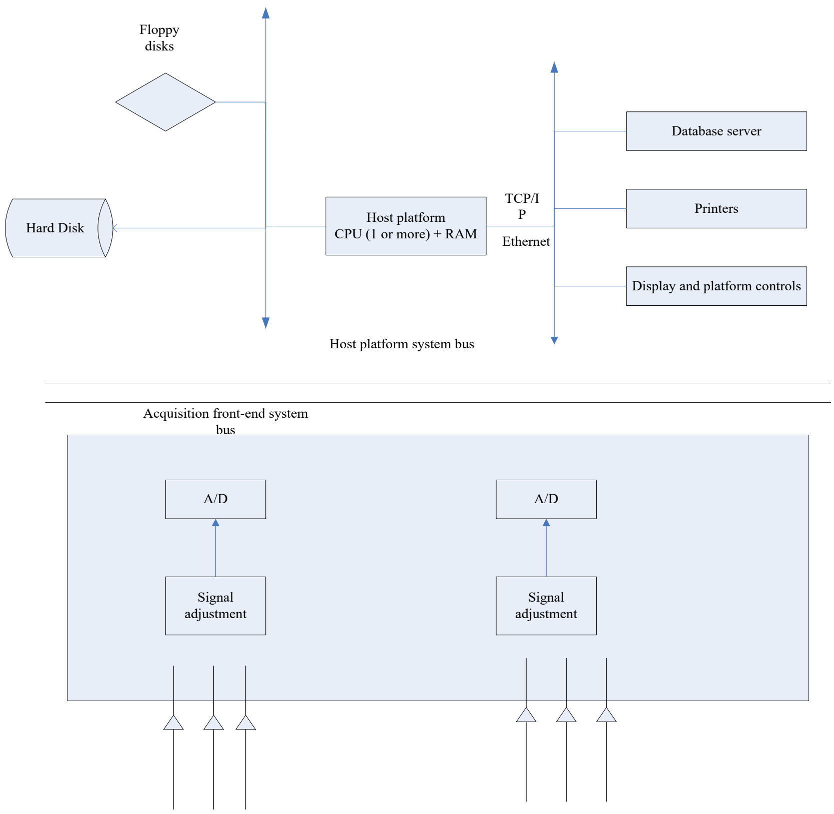

Fig.2 Schematic diagram of the hardware composition of the computer testing system for dynamic simulation experiments

a. Computer software with various dynamic signal testing and analysis functions includes random signal analysis, sinusoidal signal analysis, impact signal analysis, and so on. They are the soul of computer dynamic testing systems.

b. The computer host platform and accessories include various high-performance CPU processors and memory with powerful processing functions, high-performance built-in or external hard disks and floppy disks, high-resolution displays, and fast printers. They are the heart of computer dynamic testing systems.

c. The dynamic signal testing and acquisition device includes sensors, front-end signal conditioning amplifiers, and high-performance analog-to-digital converters, which are the foundation of computer dynamic testing systems

2.2 Finite Element Analysis

2.2.1 Theoretical Basis

The finite element method is based on the principle of variability [8-9]. The most common principles of variation are the minimum potential energy principle, the minimum residual energy principle, and the mixed variation principle [10-11]. The finite element method adopts different conversion principles and obtains different field quantities. When applying the principle of minimum potential energy, it is necessary to make assumptions about the displacement field function within the element [12]. The displacement method can be used as a fundamental unknown quantity for analysis. When applying the principle of minimum residual energy, it is necessary to make a hypothesis about the stress field, which is the stress method. In the case of using mixed change theories such as Hellinger Reissner's theory of change, it is necessary to use the mixed change method due to the need to make assumptions about certain displacements and stresses in the theory of change. When the finite element method is used for transient analysis, the Variational principle of Hamilton principle is used, and the calculation results are given [13-14]. When conducting static analysis, it is generally feasible to use the displacement method for calculation. Therefore, this method has been widely applied [15].

2.2.2 Basic Idea of Finite Element Method to Deal with Linear Elasticity Problems

(1) Discretization of structure

When conducting finite element analysis, the components need to be discretized first [16]. On this basis, this article proposes a geometric modeling method based on finite element analysis. In the discretization process, each element is connected by a node, and the number and construction method of nodes are mainly determined based on the nature of the problem, the description of the deformation mode, and the accuracy of the calculation [17]. So the finite element method can be used to analyze a dispersed body composed of the same substance and combine it with several elements. Therefore, the results obtained by using the finite element method are relatively close to the actual situation [18]. It is obvious that smaller units (denser grids) can improve the fit of the discrete domain, improve computational accuracy, but also increase computational complexity [19]. In addition, during discretization, it is necessary to determine the grid type based on factors such as geometric dimensions, physical loads, and stresses.

(2) Selection of displacement mode

In finite element analysis, the displacement mode represents the relationship between the displacement and position of any point in the element. Because the mathematical operations of polynomials are relatively simple and easy to handle, they are generally chosen as displacement functions [20]. In finite element calculation, the selection of displacement function plays a crucial role in the performance and approximation effect of the solution results. The selection of displacement function is usually based on the following criteria (convergence conditions for finite element solutions)

A. The continuity of the internal displacement function of the element and the displacement on the common boundary of adjacent elements can be coordinated.

B. The displacement function of the rigid body motion criterion can be used to describe the deformation of the element.

C. The constant strain criterion displacement function can be used to characterize the constant strain condition of the structure.

2.2.3 Main Advantages of Finite Element Method

(1) These concepts are simple but not complex, and can be understood from intuitive physical models or explored through rigorous mathematical logic.

(2) It is very flexible and has a wide range of applications. It can be successfully applied to analyze complex problems involving complex boundary conditions, nonlinearity, heterogeneous materials, and dynamics. It can be extended to other mathematical equations with boundary values, such as heat conduction, electromagnetic field and fluid dynamics.

(3) There are currently some general applications for analyzing large structures, such as SAP, ANSYS, ABAQUS, etc. Due to these advantages, finite element methods are widely used and developed.

Finite Element Analysis System - ANSYS

In the early 1980s, a large number of large-scale finite element analysis software emerged internationally, including ANSYS, NASTRAN, ASKA, ADINA, SAP, etc. This method is simple and feasible, and can lay the foundation for the optimization design and performance analysis of various industrial products. This calculation method is simple and easy to implement, with accurate calculation results, which can lay the foundation for the design and performance analysis of various industrial products. ANSYS is currently the most widely used and powerful finite element software.

ANSYS is currently the most advanced finite element calculation software in the United States. ANSYS is a multifunctional software that integrates mechanical, fluid, electrical, magnetic, acoustic, and other functions. ANSYS has been developed for over thirty years since its inception in the early 1970s, and has now developed into a fully functional finite element analysis system. The system is divided into three parts: preprocessing, analysis and calculation, and post-processing. The analysis and calculation section includes various analysis methods such as linear, nonlinear, and strongly nonlinear, as well as various analysis methods such as hydrodynamics, electromagnetic fields, sound fields, piezoelectric, and sensitivity analysis. The post-processing module can display and output the results of numerical simulation, such as Contour line map, gradient map, vector map, particle flow map, 3D profile map, translucent map (the internal situation of the structure can be observed), and the results of numerical simulation in the form of charts and curves.

ANSYS has over 100 models that can simulate various structures and materials. This project provides a widely applicable computational unit, material model, and algorithm that can efficiently handle various static, dynamic, oscillatory, linear and nonlinear structures, steady-state and transient problems. It can also handle thermal, coupled, static and time-varying structures, compressible and incompressible electromagnetic problems. This software has a good human-computer interaction interface and good programming performance, which can greatly reduce the workload of users in the process of establishing design models, establishing finite element models, analyzing and evaluating results, etc. This article adopts a centralized database structure to ensure the functionality and flexibility of each module.

2.3 Introduction to the Life and Death Unit Method

The basic idea of survival and death methods is to simulate specific operating conditions through "recovery" or "death" devices, because "recovery" means that the device exists and operates. 'Dead' means that the device is dead and not working throughout the entire phase analysis process. This allows for a good simulation of the actual construction phase.

1) In the/prep7 phase of ANSYS, the entire model is established, including those that need to be killed in stages, so there is no need to consider the later stages in the early modeling.

2) When killing an element, ANSYS would not delete the "killed" element from the model, but multiply its Stiffness matrix by a small coefficient, which is 1.0E-6 by default (this coefficient can be set with ESTIF). This is equivalent to multiplying this number by a very small number, which would be a very small number, approximately zero. The load of the idle battery would be Q, and Clonxi has no effect on the load vector (but it still appears in the battery load list). Similarly, the mass, damping, specific heat, and other similar effects of dead batteries are also zero. The mass and energy of the dead unit would not be included in the results of the model solution. When the battery is turned off, the deformation of the battery would also be 0. 3) The applicable analysis types are most static and nonlinear transient analyses

3 Experiment on the Unit of Life and Death Method

The installation of large-span complex steel structure rigid roofs often relies on temporary support molding to improve the structural stress performance. After the overall stiffness of the structure is formed, special processes are used to remove the temporary support and complete the transformation of the stress system. The roof structure on the north side of a certain project in Xinjiang is a rigid large-span spatial steel structure, consisting of 7 types, 26 arc trusses, and 14 circumferential ring trusses to form a spatial structural stress system. The top elevation of the roof reaches 78.300m, with a maximum span of 110m.

The relevant calculation formulas are as follows:

$$\partial = \frac{F}{S} \qquad (1)$$

Among them, ∂ is the stress; F is the acting force; S refers to the area of action.

$$\partial = \frac{\Delta L}{L} \qquad (2)$$

Among them, ∂ is the stress; ΔL is the deformation amount; L is the original length of the value

To gain a specific understanding of the stress changes of the structure during the critical construction phase of dismantling braces, strain monitoring is carried out throughout the unloading process of the structure. In addition, considering that the mechanical analysis of such large rigid structures during construction is usually simplified to a geometric nonlinear mechanical model, the physical properties of structural steel are still expressed by the elastic Constitutive equation in Classical mechanics. Therefore, the maximum stress at key measuring points can be calculated through measured strain, and the numerical analysis results can be simultaneously listed in the table (Table 1)

Table 1. Changes in maximum stress of the structure

	Before removing the brace	After unbracing	Change in stress	Ratio to design strength
Actual observations	0	121.12	119.62	41.10%
Direct modelling method	-81.01	19.99	99.99	31.12%
Life and death cell method	- 89.13	35.99	131.23	43.19%

The structural position it is located in has a significant relationship, and a considerable number of components do not show significant stress changes before and after dismantling the support. In actual observation results, the stress changes at the measuring points vary, that is, their sensitivity to unloading varies. From Table 1, it can be seen that during the monitoring process, the maximum value of stress change occurs at the bottom chord position of the top ring truss. The stress change reached 119.62 MPa, which means that the change in stress value at the measuring point of the component during the dismantling process reached nearly 41.10% of its design strength. This indicates that the bottom chord of the top ring truss is the key part to be controlled during the unloading process; The results of the other two numerical analyses are 99.99MPa and 131.23MPa, which have important guiding significance for ensuring the safety of dismantling the support.

The maximum stress listed in Table 1 comes from the bottom chord of the first truss that directly contacts the support frame. At the bottom column foot of the main truss, which is far away from the support frame, the change in stress before and after dismantling the support is not very significant (Table 2). At the same time, it was noted that the measurement points of the first truss not only had a change in stress magnitude, but also had completely opposite stress properties before and after the dismantling of the support (the compressive stress before unloading changed to the tensile stress after unloading). This is something that cannot be considered in general structural design.

Table 2. Maximum stress changes at the column base

	Before unbracing/MPa	After removal of bracing/MPa	Stress change/MPa	Ratio to design strength
Actual observations	1	-16.99	-16.99	6.12%
Direct modelling	-8.12	-30.12	-20.12	8.51%

method				
Life and death cell method	-7.99	-30.99	-25.10	8.01%

The use of universal finite element software to simulate the construction process has been proven to be a reliable technical support, especially for large rigid space structures similar to this project, construction simulation technology has many advantages. In order to correspond with the measured strain values, the numerical analysis results used two modeling methods to determine the amplitude of stress fluctuations. By comparing the three methods, on the one hand, it can reflect the stress changes during the unloading process of the structure. On the other hand, by comparing Table 1-2, it can be found that no matter which modeling method is used, there is no significant difference in the stress changes of the structure before and after dismantling the support.

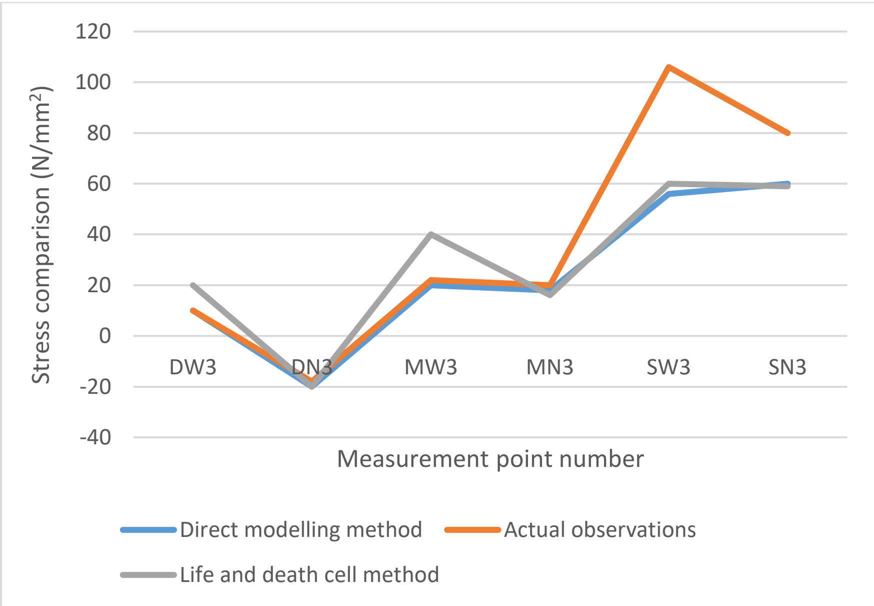

Fig.3 Comparison of stress at the base of the main truss column

As shown in Figure 3, for the stress data with measurement points DW3, DN3, MW3, and MN3, the direct modeling method almost perfectly fits the actual observation values. However, for the stress data with measurement points SW3 and SN3, the direct modeling method has a significant deviation from the actual observation values.

As shown in Figure 4, comparing the stress monitoring data of six measurement point numbers (QS2, QS3, QS4, QS5, QS6, and QS7), it is the life and death unit method that is generally closer to the actual observation values. Especially when detecting measurement point numbers QS5, QS6, and QS7, the accuracy of the direct modeling method is far less than that of the life and death unit method.

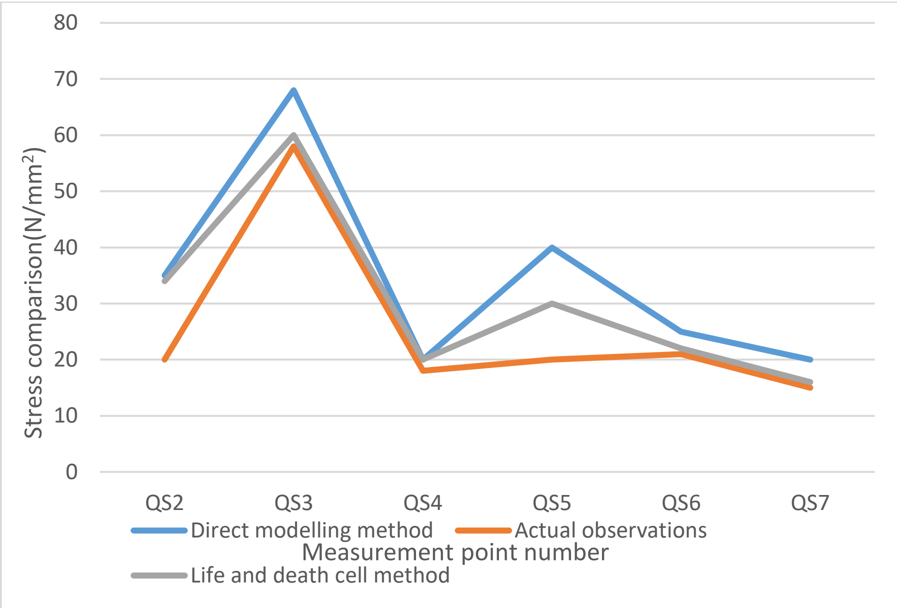

Fig.4 Comparison of Top Chord Stress of Ring Truss

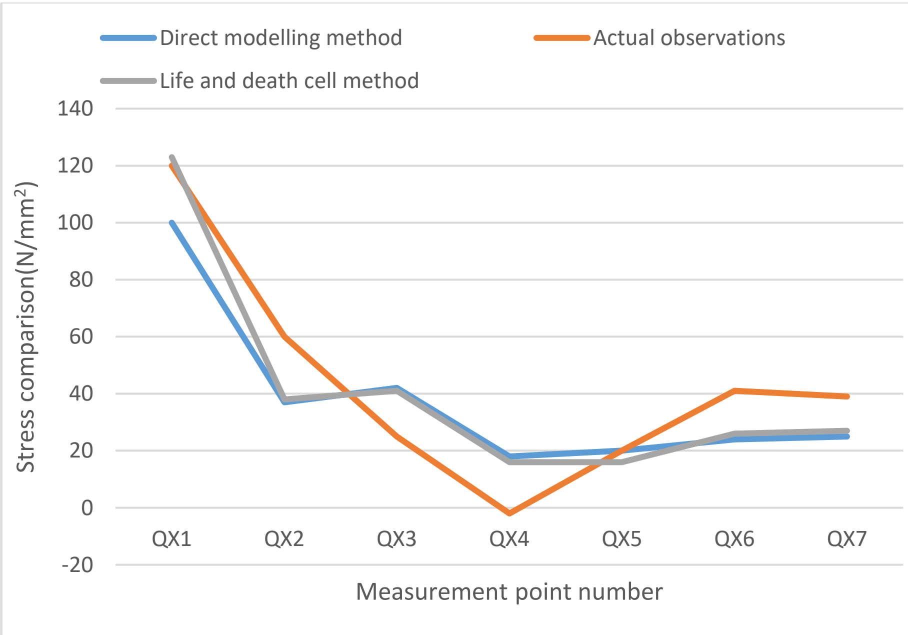

Fig.5 Comparison of Stress in the Bottom Chord of Ring Truss

As shown in Figure 5, the overall fit between the direct modeling method and the actual observation values is not as high as that between the life and death unit method and the actual observation values. From the figure, it can be seen that there is not much difference between the direct modeling method and the life and death element method, but the life and death element method has higher accuracy in calculating structural forces.

4 Conclusions

Using computer to build Structural mechanics simulation system can ensure the implementation of engineering safety. First, this paper introduces the simulation system of Structural mechanics, finite element algorithm, computer simulation analysis and modern engineering. At the same time, it discusses the ANSYS system based on the finite element algorithm, and compares the direct modeling method with the birth and death element method. It is verified that the accuracy of the Structural mechanics simulation system based on the optimized finite element algorithm is greatly improved, which is of research significance.

References

[1] Moghadam P Z, Rogge S M J, Li A. Structure-mechanical stability relations of metal-organic frameworks via machine learning[J]. Matter, 2019, 1(1): 219-234.

[2] Deringer V L, Bernstein N, Csányi G.Origins of structural and electronic transitions in disordered silicon[J]. Nature, 2021, 589(7840): 59-64.

[3] Younesian D, Hosseinkhani A, Askari H.Elastic and viscoelastic foundations: a review on linear and nonlinear vibration modeling and applications[J]. Nonlinear Dynamics, 2019, 97(1): 853-895.

[4] Li X, Lu L, Li J.Mechanical properties and deformation mechanisms of gradient nanostructured metals and alloys[J]. Nature Reviews Materials, 2020, 5(9): 706-723.

[5] Kumar A, Sharma K, Dixit A R. A review of the mechanical and thermal properties of graphene and its hybrid polymer nanocomposites for structural applications[J]. Journal of materials science, 2019, 54(8): 5992-6026.

[6] Wu Q, Miao W, Zhang Y.Mechanical properties of nanomaterials: A review[J]. Nanotechnology Reviews, 2020, 9(1): 259-273.

[7] Shang L, Zhang W, Xu K. Bio-inspired intelligent structural color materials[J]. Materials Horizons, 2019, 6(5): 945-958.

[8] Onyibo E C, Safaei B. Application of finite element analysis to honeycomb sandwich structures: a review[J]. Reports in Mechanical Engineering, 2022, 3(1): 192-209.

[9] Welch-Phillips A, Gibbons D, Ahern D P, et al. What is finite element analysis?[J]. Clinical Spine Surgery, 2020, 33(8): 323-324.

[10] Jabali I, Naagarjun L, Paal O.Investigating the Interaction of Stress Intensity Factors in Thin-Walled Cylindrical Vessels using Finite Element Analysis[J]. International Journal of Basic and Applied Sciences, 2023, 10(03): 740-744.

[11] Kardani N, Zhou A, Nazem M. Improved prediction of slope stability using a hybrid stacking ensemble method based on finite element analysis and field data[J]. Journal of Rock Mechanics and Geotechnical Engineering, 2021, 13(1): 188-201.

[12] Chen H, Fan D, Huang J.Finite element analysis model on ultrasonic phased array technique for material defect time of flight diffraction detection[J]. Science of Advanced

Materials, 2020, 12(5): 665-675.
[13] Liu M. Finite element analysis of pitting corrosion on mechanical behavior of E690 steel panel[J]. Anti-Corrosion Methods and Materials, 2022, 69(4): 351-361.
[14] Wu T, Antona-Makoshi J, Alshareef A.Investigation of cross-species scaling methods for traumatic brain injury using finite element analysis[J]. Journal of neurotrauma, 2020, 37(2): 410-422.
[15] Dal Piva A M O, Tribst J P M, Borges A L S.The ability of mouthguards to protect veneered teeth: A 3D finite element analysis[J]. Dental Traumatology, 2023, 39(3): 191-199.
[16] Spruegel T C, Bickel S, Schleich B. Approach and application to transfer heterogeneous simulation data from finite element analysis to neural networks[J]. Journal of Computational Design and Engineering, 2021, 8(1): 298-315.
[17] Tonin B S H, He Y, Ye N. Effects of tightening torque on screw stress and formation of implant-abutment microgaps: A finite element analysis[J]. The Journal of Prosthetic Dentistry, 2022, 127(6): 882-889.
[18] Spathopoulos S C, Stavroulakis G E. Springback prediction in sheet metal forming, based on finite element analysis and artificial neural network approach[J]. Applied Mechanics, 2020, 1(2): 97-110.
[19] Nath U, Yadav V, Purohit R. Finite element analysis of AM30 magnesium alloy sheet in the laser bending process[J]. Advances in materials and processing technologies, 2022, 8(2): 1803-1815.
[20] Elias D M, Valerio C S, de Oliveira D D.Evaluation of Different Heights of Prosthetic Crowns Supported by an Ultra-Short Implant Using Three-Dimensional Finite Element Analysis[J]. The International Journal of Prosthodontics, 2020, 33(1): 81-90.