Dynamic Response Analysis of Cable-Stayed Bridge Structure Under Moving Load Based on Finite Element Method

Zihan Rong
Chongqing Jiaotong University, No.66 Xuefu Avenue, Nan 'an District, Chongqing, China
632101151915@mails.cqjtu.edu.cn

ABSTRACT

The characteristics and laws of the response of the vehicle to the cable structure of the cable-stayed bridge under different driving conditions are widely concerned by the engineering community. In this paper, the finite element model of cable-stayed bridge is established by using the nonlinear finite element software Abaqus, and the stress variation characteristics and laws of cable are simulated when the vehicle driving conditions on the bridge are different, so as to provide the basis for the design improvement of cable under moving load.

Keyword: Finite Element Method, Dynamic Analysis, Cable-Stayed Bridge, Moving Load, Stress Analysis

1. INTRODUCTION

As a numerical method for engineering structure analysis, finite element analysis has been widely used in the research and design of cable-stayed bridges. With the continuous progress of finite element software, from the early bar model to the relatively accurate solid model, the solid element model can bring higher precision, but also increase the pressure of calculation, so the multi-scale model has become an important research direction of long-span bridges. By establishing the finite element model of cable-stayed bridge, the response of the bridge structure under different dynamic loads can be simulated and the safety performance of the structure can be evaluated.

In recent years, many scholars have conducted preliminary studies on the stress of bridge structures under moving vehicle loads. In addition, some scholars have attempted to identify the parameters of vehicles driving on the bridge by utilizing dynamic response of bridge[1]. According to the theory of strain influence line under moving vehicle load, Bitao Wu et al[2]studied the relationship between local element bending stiffness and long gauge strain, and proposed a new method for bridge damage identification and bearing capacity evaluation; Pipinato Alessio[3] outlines the principle and method of dynamic response of bridges to moving loads and analyzes the current response of medium and short span bridges to moving loads according to European norms.; Sang-Hyo Kim et al[4]proposed a three-dimensional numerical model, which can more accurately show the dynamic response of curved bridges to moving vehicles.

The future development trend of cable-stayed bridge is shown in the following aspects[5]: (1) Lightweight bridge deck; (2) Diversification of tower structure (inverted Y-shaped or diamond-shaped); (3) New type of cable.

The purpose of this study is to study the structural response of cable-stayed bridges under different dynamic loads by using finite element analysis software Abaqus. By simulating the dynamic response of cable-stayed bridge under moving load, it can provide a powerful tool and reference for the design, evaluation and optimization of cable-stayed bridge.

2. FINITE ELEMENT ANALYSIS METHOD AND MODEL ESTABLISHMENT

2.1. Dynamic simulation theory based on finite element method

With the complexity of engineering structures, the analytical solution of dynamic problems by theoretical methods becomes more and more complex. The finite element method is proposed to solve this problem reasonably. At present, the finite element method has been widely used in engineering mechanics, thermodynamics and other fields. The solution process of the finite element method is divided into the following steps: Firstly, the structure is discretized, the continuous medium to be solved is divided into a finite number of simple elements, and the finite element mesh is established, as shown in Fig.1.

International Conference on Smart Transportation and City Engineering (STCE 2023),
edited by Miroslava Mikusova, Proc. of SPIE Vol. 13018, 130182K

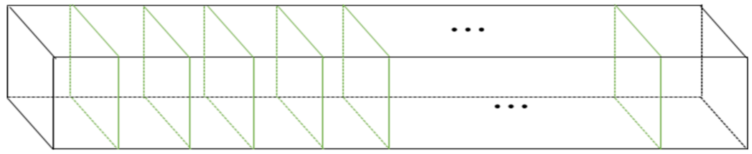

Fig.1. Continuous medium discretization diagram

Then the element model is established, and an appropriate mathematical model is selected within each finite element, such as linear elastic model or nonlinear model, to describe the mechanical behavior of the medium. The global equations are established according to the model, and the global equations are obtained by assembling the local equations of each finite element. The approximate solutions of field quantities such as displacement and stress are obtained by solving the global equations by numerical methods (such as Newmark-β method). Then, according to the approximate solution, the required physical quantities, such as stress distribution, deformation and displacement field, are calculated and analyzed.

Newmark-method is a generalized method of linear acceleration method. The Newmark-method can be considered as a generalized algorithm that summarizes the average constant acceleration and linear acceleration algorithms. Newmark-method has quasi-static incremental equation form and different types of quasi-static total equation form. The most commonly used methods in finite element dynamic analysis are central difference method, Newmark method and Wilson-θ method.

The dynamic response of the finite element structure can be obtained by solving the dynamic equilibrium differential of the following structure:

$$[M][\ddot{X}] + [C][\dot{X}] + [K]X = [F(t)] \tag{1}$$

Where, $[M]$, $[C]$, $[K]$ are mass matrix, damping matrix and stiffness matrix of structural element respectively; $[X]$ is the displacement vector matrix of structural nodes, $[F(t)]$ is the load vector matrix at the structural node.

In practical engineering, it is difficult to express damping. Generally, the linear combination of structural mass matrix and stiffness matrix is used, that is, Rayleigh damping:

$$[C] = \alpha [M] + \beta [K] \tag{2}$$

Where, α and β are called mass damping coefficient and stiffness damping coefficient respectively.

Numerical Newmark-β method for solving linear dynamic equations

The following is an case of MCK equation with degree of freedom n = 3 (Fig.2).

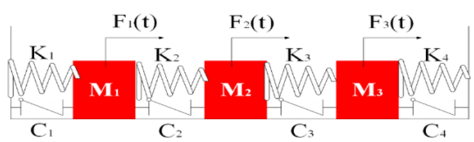

Fig.2. The schematic diagram of the model with n = 3 degrees of freedom

$$\begin{cases} M_1\ddot{X}_1 + (C_1 + C_2)\dot{X}_1 - C_2\dot{X}_2 + (K_1 + K_2)X_1 - K_2X_2 = F_1(t) \\ M_2\ddot{X}_2 - C_3\dot{X}_3 + (C_2 + C_3)\dot{X}_2 - C_2\dot{X}_1 - K_3X_3 + (K_2 + K_3)X_2 - K_2X_1 = F_2(t) \\ M_3\ddot{X}_3 - C_3\dot{X}_2 + (C_3 + C_4)\dot{X}_3 - K_3X_2 + (K_3 + K_4)X_3 = F_3(t) \end{cases} \tag{3}$$

Rewrite the equations into matrix form:

$$\begin{bmatrix} M_1 & 0 & 0 \\ 0 & M_2 & 0 \\ 0 & 0 & M_3 \end{bmatrix} \begin{Bmatrix} \ddot{X}_1 \\ \ddot{X}_2 \\ \ddot{X}_3 \end{Bmatrix} + \begin{bmatrix} C_1+C_2 & -C_2 & 0 \\ -C_2 & C_2+C_3 & -C_3 \\ 0 & -C_3 & C_3+C_4 \end{bmatrix} \begin{Bmatrix} \dot{X}_1 \\ \dot{X}_2 \\ \dot{X}_3 \end{Bmatrix} + \begin{bmatrix} K_1+K_2 & -K_2 & 0 \\ -K_2 & K_2+K_3 & -K_3 \\ 0 & -K_3 & K_3+K_4 \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \end{Bmatrix} = \begin{Bmatrix} F_1(t) \\ F_2(t) \\ F_3(t) \end{Bmatrix} \tag{4}$$

Using the following assumptions:

$$\dot{X}_{i+1} = \dot{X}_i + \left[(1-\delta)\ddot{X}_i + \delta \ddot{X}_{i+1} \right] \Delta t \tag{5}$$

$$X_{i+1} = X_i + \dot{X}_i \Delta t + \left[\left(\frac{1}{2} - \alpha \right) \ddot{X}_i + \alpha \ddot{X}_{i+1} \right] (\Delta t)^2 \tag{6}$$

Where, δ and α two parameters should be determined according to the accuracy and stability of the integral.

The calculation accuracy of Newmark-β depends on the size of time step Δt. The determination of the time step must consider the load change and the length of the natural vibration period of the system. In order to ensure the contribution of the high frequency components of interest, Δt is usually required to be less than $\frac{1}{7}$ of the minimum structural natural vibration period that has a significant effect on the response. The stability study also shows that the Newmark-β is unconditionally stable when $\beta \geq \frac{1}{4}$; When $\beta < \frac{1}{4}$, it is conditional stable, and the stability condition is: for $\beta = 0$, $\frac{1}{12}$ and $\frac{1}{6}$ these three cases, they must be less than 0.318,0.390 and 0.551 respectively. $T_{\min}$ is the minimum natural vibration period of the structure.

After finishing, we can get only the displacement value at the moment, and then get the value of velocity and acceleration at the moment.

2.2. Element selection method for numerical model

Truss element and Beam element are two types of elements commonly used in finite element analysis to simulate and analyze members or beams in structures. There are some differences between them in simulating member behavior.

The Truss element is a one-dimensional element used to simulate members or slender members in a structure. It idealizes the bar as a thin bar without mass and volume, only considering the effect of axial force. Therefore, the Truss element only focuses on the tensile or compressive behavior of the bar during the modeling process, ignoring the bending and shear effects of the bar. In addition, the Truss element has axial displacement freedom, and its stiffness is determined by geometric properties (length, cross-sectional area) and material properties (elastic modulus).

Beam element is also a one-dimensional element, which is used to model the beam or bar in the structure. Different from the Truss element, the Beam element considers the cross-sectional shape and rotational stiffness of the beam and can simulate the bending and shear behavior of the beam. In contrast, the nodes of the Beam element have six degrees of freedom, including three displacement and three rotation degrees of freedom. Its stiffness depends on geometric properties (cross-sectional shape, size) and material properties (elastic modulus, shear modulus).

In the study of cable-stayed structures, the following assumptions are often used : the cable structure only generates normal stress on its cross section, and the normal stress is evenly distributed on the cross section. In view of the fact that the stay cables in the bridge are usually twisted by multiple strands of steel cables, in the modeling process, this paper uses the characteristics of the Truss element to simulate the behavior of the stay cables, where the tension and compression of the Truss element correspond to the stress state of the stay cables. This modeling method can better describe the performance characteristics of stay cables.

The main reason for choosing shell elements for bridge deck modeling is that shell elements can simplify the geometry of the bridge deck and improve computational efficiency, while providing high-precision stress analysis and simplifying the definition of boundary conditions. Since the bridge deck is usually a thin plate structure, the shell element can better capture the plane strain effect and the bending characteristics of the plate structure, while reducing the model complexity and computing resource requirements. The choice of shell element makes the bridge deck modeling faster and more accurate, and it is easy to define the boundary conditions.

The main reason for choosing to use solid elements for bridge column modeling is that solid elements can accurately represent the complex geometry of bridge columns and provide high-precision stress analysis and dynamic analysis. The solid element considers the volume effect and three-dimensional deformation of the column, which can predict the stress distribution and structural response more accurately. In addition, the solid element also allows finer mesh division, captures local stress concentration and deformation characteristics, and improves analysis accuracy. The use of solid elements also facilitates the definition of boundary conditions and loading methods, providing flexible and accurate simulation. It should be noted that when choosing to use entity units, computing resource requirements need to be considered, because they have more nodes and degrees of freedom, and may require more computing resources and time. Therefore, in practice, it is necessary to comprehensively consider factors such as geometric complexity, analysis purpose, computational efficiency and resource constraints, and select the most suitable unit type for modeling.

2.3. Modeling of cable-stayed bridge and moving load

The commonly used finite element softwares are ABAQUS, ANSYS, Partran and so on. ABAQUS software is the first choice for nonlinear finite element calculation. In this paper, Abaqus modeling is used to simulate the cable-stayed bridge under different dynamic loads. Limited by computer hardware limitations and time cost considerations, a simplified bridge model is used for simulation. In the finite element software Abaqus, the pillar adopts three-dimensional, deformable solid element (C3D8R), the stay cable adopts Truss element (T3D2), the bridge deck adopts shell element (S4R), and the bottom of the pillar is fixed with fixed hinge support.

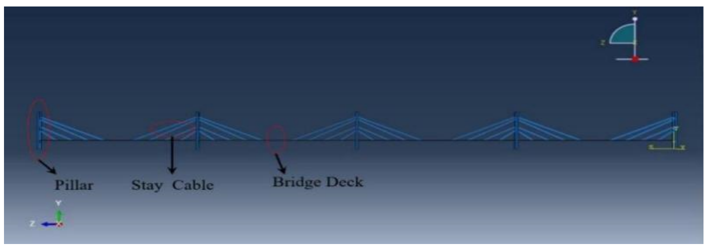

Fig.3. Cable-stayed bridge model

The cable-stayed bridge is a multi-tower multi-span cable-stayed bridge with a single span of 1000 m and a deck width of 50 m. The material parameters of the bridge are selected from Enshi Shizhou Bridge[6] in Hubei Province,China,16 stay cables are arranged in a single span, and the pillars are simulated by solid elements. C50 concrete is used, and the Young 's modulus of concrete is: 3.45×10^{10} N/m^2, the stay cable is simulated by Truss element, using Steel, the Young 's modulus is: 2×10^{11} N/m^2 [7] (Fig.3).

Define the material and cross-sectional properties of of the cable-stayed bridge (Fig.4).

The next step after creating the model part is to define and assign material and section properties to the part. Each region of the deformable body must specify a section property containing the material definition. In this model, the properties of Concrete and Steel will be created, and the material properties will be assigned to the pillar, bridge deck and stay cables, respectively.

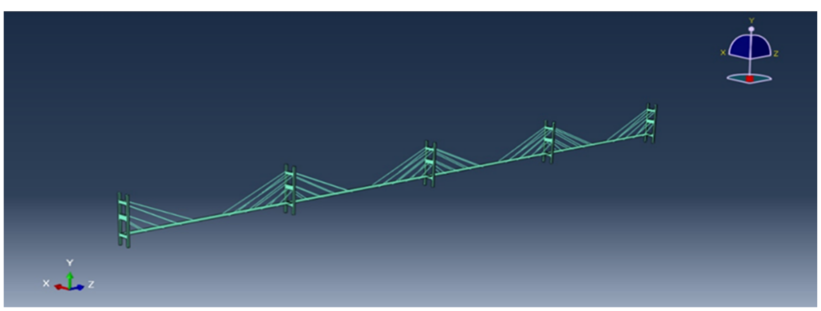

Fig.4. Model section attributes given

The 'dynamic-implicit' analysis step is set under the step module, the time length is set, and the geometric nonlinearity is turned on.

In this paper, the rectangular rigid body is used to represent the moving load and simulate the process of the car driving on the bridge deck. Under the interaction module, the tangential friction-free and normal hard friction are set, and then the cuboid is set as a rigid body.

Under the load module, the boundary condition at the bottom of the pillar is set to be rigidly fixed to achieve the purpose of fixing the support, and the rigid body is set to a speed of 100 (Table 1).

Under the grid module, the components are divided into meshes. The components after meshing are divided into 649 meshes. The grid element type of stay cable is T3D2, the mesh element type of bridge deck is S4R, and the mesh element type of pillar is C3D8R. The meshing results are as follows (Fig. 5):

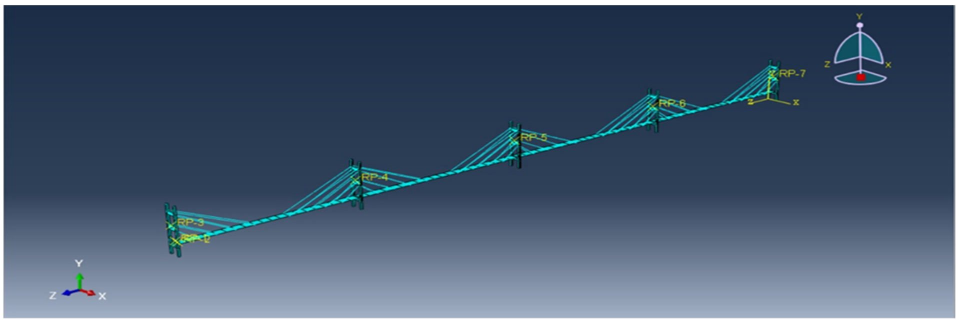

Fig. 5. Case meshing diagram

Table 1. The specific information table of three cases

Condition / Case	Number of Rigid Bodies	Velocity	Time Length	Motion Type
Case1	2	100	10	Unilateral motion
Case2	4	100	10	Opposite motion
Case3	2	100	10	Single reciprocating

3. ANALYSIS

After calculation, the result cloud of Case123 is as follows (Fig.6 to Fig.8):

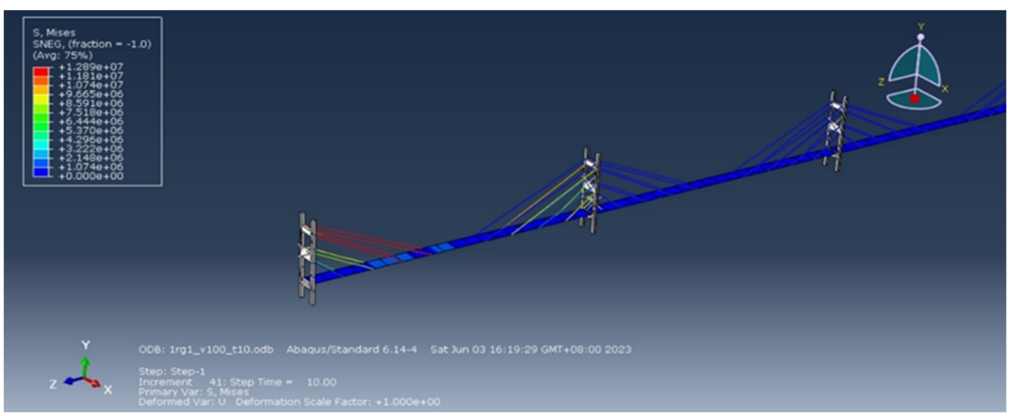

Fig.6. Case 1 Analysis result cloud chart

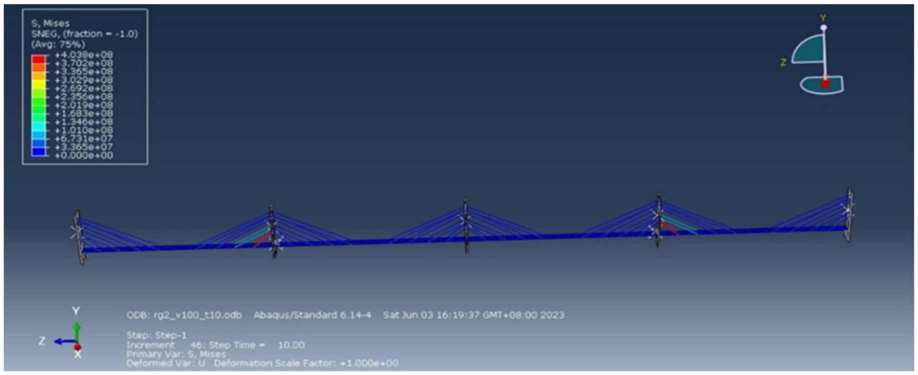

Fig. 7. Case 2 Analysis results cloud picture

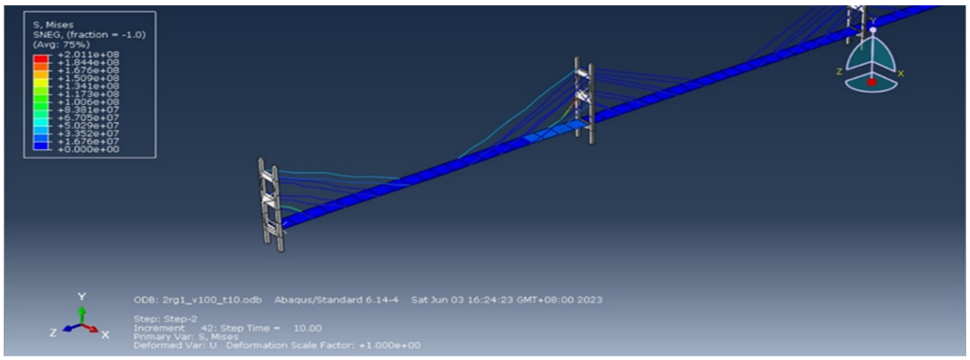

Fig.8. Case 3 Analysis result cloud chart

The principle of path selection : Case1 and Model3 are drawn from left to right, with four nodes as a group; Case2 draws the path from left to right in a group of eight nodes (Fig. 9).

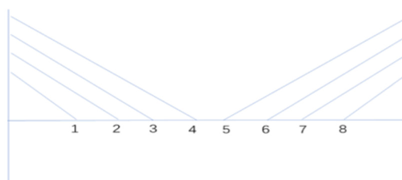

Fig. 9. Schematic diagram of single-span stay cable of cable-stayed bridge

Case 1 (Fig. 10): 2 rigid bodies, velocity 100, time length 10(Unilateral movement)

Fig. 10. Case 1 Path diagram

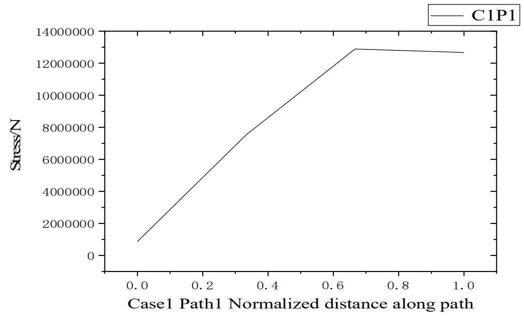

Fig.11. Case 1 Path 1 normalized stress diagram

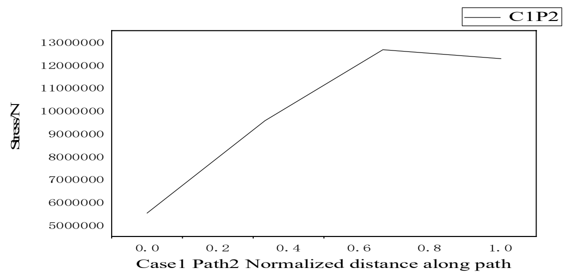

Fig.12. Case 1 Path 2 normalized stress diagram

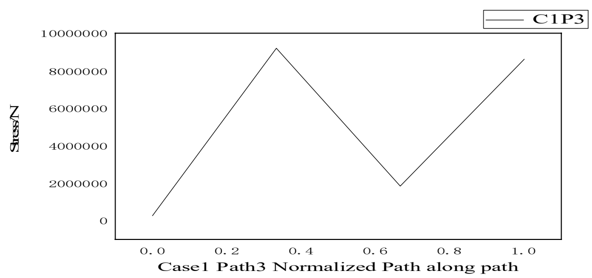

Fig. 13. Case 1 Path 3 normalized stress diagram

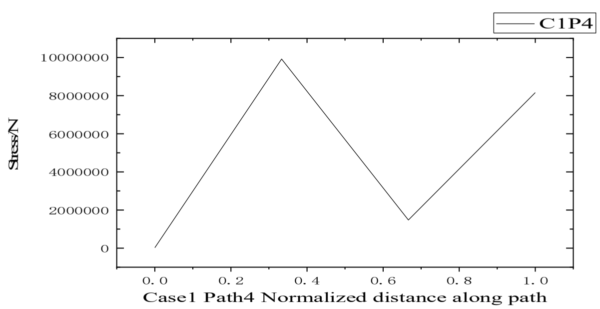

Fig. 14. Case 1 Path 4 normalized stress diagram

In Case 1 (Fig.11 to Fig. 14), when two small sliders move side by side, the force between the stay cables changes greatly. According to C1P1, C1P2, C1P3 and C1P4, it can be seen that the stay cables 3 and 6 are subjected to relatively large tension during the movement of the slider. Therefore, when there is only one-way dynamic load movement on the bridge deck, the largest force is not necessarily the outermost stay cable. Considering that in real cases, the dynamic load will not stop at the bridge deck, so the selection of the force value of the stay cable 8 is temporarily ignored.

Case 2 (Fig. 15): 4 rigid bodies, velocity 100, time length 10 (opposite motion)

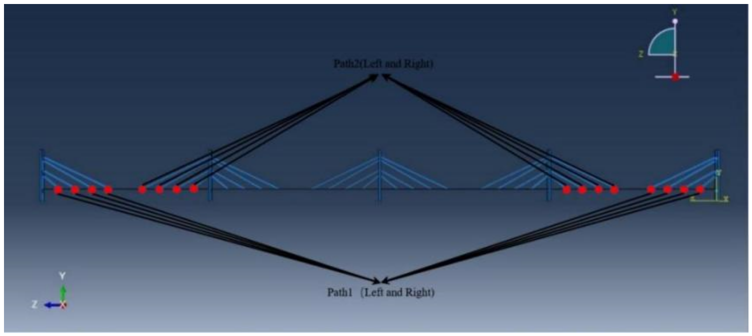

Fig. 15. Case 2 Path diagram

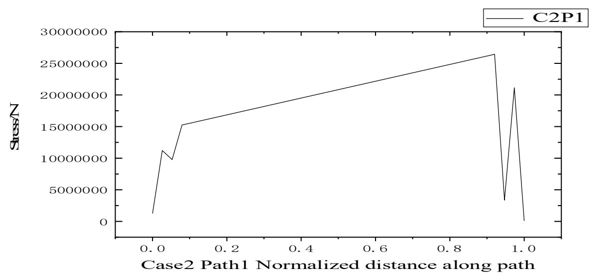

Fig.16. Case 2 Path 1 normalized stress diagram

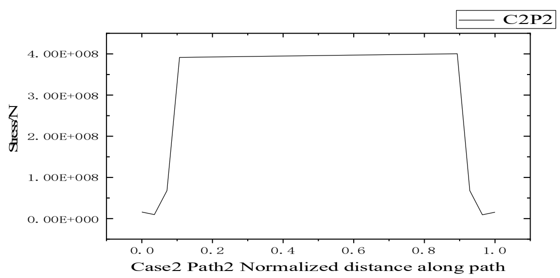

Fig.17. Case 2 Path 2 normalized stress diagram

In the second case (Fig.16 to Fig.17), two pairs of small sliders move in opposite directions. According to C2P1 and C2P2, it can be seen that the stay cables 4 and 5 are subjected to relatively large tension during the movement. Compared with the first case, two dynamic loads are added. The increase of gravity also leads to the increase of cable force. The dynamic response of the cable stress is generally increasing, and the influence of the value of the cable 8 is also ignored in the analysis.

Case 3 (Fig. 18): 2 rigid bodies, velocity 100, time length 10 (single reciprocating)

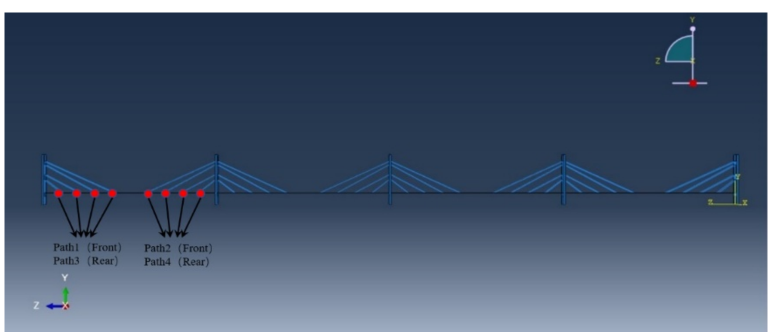

Fig. 18. Case3 Path diagram

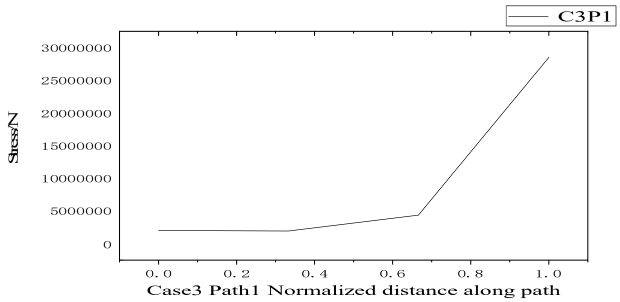

Fig. 19. Case3 Path1 normalized stress diagram

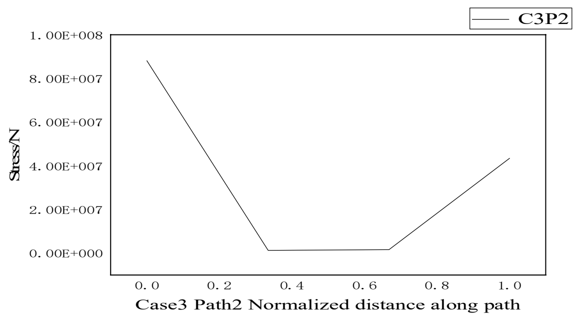

Fig. 20. Case3 Path2 Normalized stress diagram

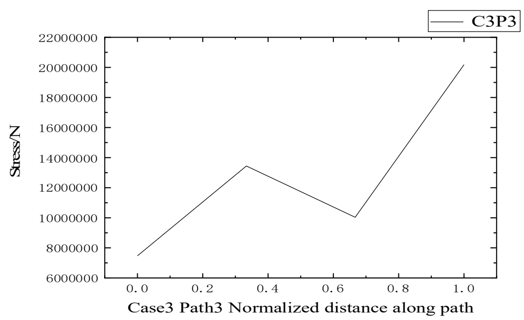

Fig. 21. Case3 Path3 Normalized stress diagram

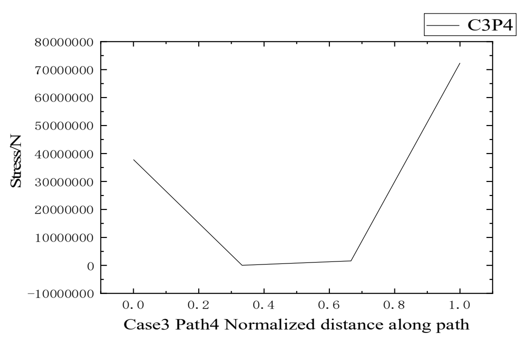

Fig. 22. Case3 Path4 Normalized stress diagram

In the third case (Fig. 19 to Fig. 22), the two sliders move back and forth. Because in the simulation of the results, the moving load on the left side does not move in the pre-direction, so the analysis of C3P2 and C3P4 is abandoned. According to C3P1 and C3P3, it can be seen that the cable 5 and 6 are relatively large, and the stress value of cable 8 is also ignored.

Through analysis, it can be seen that the stress of the stay cable gradually increases from left to right. Compared with Case 1, due to the round-trip movement, it can be regarded as the movement of vehicles on the two-way lane. The overall tension of the stay cable is relatively large, close to the reality and more realistic.

4. DISCUSSION

4.1. Regularity analysis

According to the stress analysis of the cable under the moving load under the above different cases, it can be seen that the tension of the cable 3,4,5 and 6 is larger than that of the other four. In comparison, the Case 3 has more reference value.

4.2. Cause analysis

When the cable-stayed bridge is subjected to load, the mid-span will bear the maximum stress[8]. From the perspective of static equilibrium, the cable-stayed bridge is suspended on the bridge tower by the stay cable. In the static equilibrium state, the gravity and load of the bridge deck will be transmitted to the bridge tower through the stay cable, and the stay cable will produce tension. Since the mid-span stay cable is the longest in the entire bridge system, the load it bears is also the largest, resulting in the largest cable stress. From the perspective of the bending effect of the bridge deck, when the load is applied to the cable-stayed bridge, the bridge deck will undergo bending deformation. According to beam theory, the maximum bending moment of a curved beam occurs at the mid-span position. The main girder of the cable-stayed bridge can be regarded as a continuous beam, and the bending moment is the largest at the mid-span position. There is a certain inclination angle between the cable and the main girder, and the tension of the cable will be affected by the bending moment caused by the bending of the main girder. Therefore, the cable in the middle of the span will reach the maximum stress caused by bending. From the geometric effect of the stay cable, the stay cable of the cable-stayed bridge is connected to the bridge tower and the bridge deck at a certain angle. The geometric shape of the cable determines the projection length of the cable on the bridge deck, and the tension of the cable is proportional to its projection length. Due to the large inclination angle of the cable in the middle of the span, the projection length is shorter. Under the same tension, the tension on the cable per unit length will be larger, so the cable stress in the middle of the span is larger.

4.3. Dynamic analysis modeling

The purpose of dynamic analysis is to obtain the structural response of the actual structure under the action of traffic (such as car movement), and to calculate the stress of various components directly applied in structural design. Therefore, it is necessary to ensure the authenticity of the input dynamic excitation and materials. However, due to the limitations of this paper, the model cannot include every detail of the structure. The results and application of dynamic analysis must be based on a full understanding of the model assumptions and limitations.

5. CONCLUSION

In this paper, the finite element simulation of the stress of the cable-stayed bridge structure under dynamic load is carried out, and the stress characteristics and laws of the cable-stayed cable of the cable-stayed bridge are explored through the calculation results of the structural response:

In the analysis of the stress numerical line chart of the path node of the bridge deck and the stay cable, it is concluded that:

(1). The cable of the cable-stayed bridge is subjected to the maximum stress when the vehicle is driven to the middle of the bridge span. This is because the cable is subjected to the maximum axial tension when the vehicle passes through the secondary main span. Therefore, in the design and construction of cable-stayed bridge, it is necessary to ensure that the cable can withstand the maximum stress to ensure the safety of the bridge.

(2). During the driving of the vehicle, the stay cables near the mid-span are subjected to greater stress than other cables. This may be due to uneven weight distribution of the vehicle near the mid-span or other factors. In order to enhance the axial tensile capacity of stay cables, these cables can be reinforced. Reinforcement methods may include using stronger materials, increasing the diameter or number of cables, or using other structural improvements.

(3). The beam (main beam) arranged under the mid-span bridge deck can choose more solid materials or structures to support the bridge deck to prevent the collapse of the mid-span bridge deck and ensure the reliability of the bridge structure. Choosing a strong material or structure can increase the carrying capacity of the beam and enable it to withstand pressure from vehicles and other loads, thereby reducing the risk of bridge deck collapse.

(4). In the design and construction of cable-stayed bridges, special attention should be paid to the areas near the secondary main span and mid-span where vehicles pass through the bridge, because these areas require higher stress bearing capacity of stay cables and mid-span decks. Additional measures should be taken in these areas, such as increasing material strength, optimizing structural design or strengthening the support system to ensure the reliability of the bridge.

REFERENCES

[1] Deng L, Cai C S. Identification of parameters of vehicles moving on bridge [J].Engineering Structure,2009,31(10):2474-2485.

[2] Bitao Wu, Gang Wu, Caiqian Yang, Yi He. Damage identification and bearing capacity evaluation of bridges based on distributed long-gauge strain envelope line under moving vehicle loads [J]. Journal of Intelligent Material Systems and Structures,2016,27(17).

[3] Alessio P .Moving Loads Bridge Response: Structural Analysis and Eurocode Provisions[J].Advanced Materials Research,2012,629(629-629).

[4] Sang-Hyo Kim, Yong-Seon Lee, Kwang-Yil Cho. Analysis of Horizontal Reactions Due to Moving Vehicle Loads in Curved Bridges with Varied Support Conditions [J]. Advances in Structural Engineering,2005,8(5).

[5] Wu Zhongrong. Determining of Reasonable Completion State and Simulation Analysis of Construction Controlling of Cable-Stayed Bridges[D].Chongqing University,2014.

[6] Yang Xing, Zhang Ming, Zhou Shuixing. Application of Influence Matrix Method to Secondary Cable-Stayed Bridges[J]. Journal of Chongqing Jiaotong University (Natural Science Edition),2009,28(03):508-511.

[7] Li Jiashun. Research on Calculation Method of Cable-Stayed Bridge Construction Process Based on Specified Stress Finite Element Method [D]. Chongqing Jiaotong University,2021.DOI:10.27671/d.cnki.gcjtc.2021.000067.

[8] Han Huiming. Numerical and Experimental Study on Damage Detection of Beam Under Moving Loads [D]. Shandong Agricultural University,2017.