Stress Finite Element Modeling Analysis of Simply Supported Beam Bridge

Xinwei Zhang*, Yaxiong Li, Xinzhi Yang and Yuqing Wang

Xi’an High-tech Institute, Xi’an, Shaanxi, 710025, PR China
*kins265@163.com

Abstract. In order to quickly and accurately analyze the stress and strain of simply supported beam bridge, overcome the problems of insufficient weight of finite element analysis software, single analysis type and difficult expansion, a bridge stress and stress weakness analysis method based on finite element analysis is proposed. This method improves the analysis accuracy by independently designing the analysis program and using symbolic computation, and can optimize and supplement the existing analysis software according to the demand. The example verification shows that this method can effectively analyze the bridge stress, accurately analyze multiple stress weaknesses of the model, and display its position. The accuracy and speed of the analysis are considerable.

1. Introduction
Simple-supported beam bridge has been widely used in bridge construction because of its simple structure, convenient erection and suitable for various geological conditions. As the key link of bridge design, construction and maintenance, stress analysis has been studied by many scholars. In [1], it puts forward a stress calculation method of pre-compression bending steel-concrete composite simply supported beam bridge. In [2], it realized the calculation method of measured stress in bridge construction monitoring engineering by programming. In [3], it analyzed the stress of bridge structure through static load test and compared it with theoretical data. In this paper, the finite element analysis method is used to analyze the stress and stress weakness of the simply supported beam bridge.

2. Finite element modeling analysis
Finite element analysis is an effective calculation tool for approximate solving complex partial differential equations. Simply speaking, a continuous region or model is discretized, and the model is divided into a finite number of elements. Then the combination of these elements is calculated, and finally the analysis results of the overall model are obtained[5].

2.1. Structural separation
Structural discretization is the first step of finite element analysis. For the simply supported beam bridge, the structure is more square, and the boundary is mainly rectangular. Therefore, the eight-node hexahedron element is used for discretization to ensure good subdivision effect and analysis accuracy. Each element node of the eight-node hexahedral element can move along three axes. The basic structure of each node and its corresponding node displacement is shown in figure 1.
Each node of the element has three degrees of freedom, so for eight nodes, there are 24 degrees of freedom. So the nodal displacement array $q^e$ and nodal force array $F^e$ of such an element can be expressed as

$$q^e = \begin{bmatrix} u_1 & u_2 & w_1 & u_3 & u_4 & w_2 & \cdots & u_8 & u_9 & w_3 \end{bmatrix}^T$$

$$F^e = \begin{bmatrix} F_{x_1} & F_{y_1} & F_{z_1} & F_{x_2} & F_{y_2} & F_{z_2} & \cdots & F_{x_8} & F_{y_8} & F_{z_8} \end{bmatrix}^T$$

The unit contains eight nodes that can be displaced, so the displacement field function needs to set at least eight undetermined position coefficients. According to the basic principle of determining this displacement mode, the displacement mode of each node of the unit is calculated and derived as follows

$$u(\xi, \eta, \zeta) = a_i + a_2 \xi + a_3 \eta + a_4 \zeta + a_5 \xi \eta + a_6 \xi \zeta + a_7 \eta \zeta + a_8 \xi \eta \zeta$$

$$v(\xi, \eta, \zeta) = b_i + b_2 \xi + b_3 \eta + b_4 \zeta + b_5 \xi \eta + b_6 \xi \zeta + b_7 \eta \zeta + b_8 \xi \eta \zeta$$

$$w(\xi, \eta, \zeta) = c_i + c_2 \xi + c_3 \eta + c_4 \zeta + c_5 \xi \eta + c_6 \xi \zeta + c_7 \eta \zeta + c_8 \xi \eta \zeta$$

A total of 24 correlation coefficients are determined by 8 node coordinates, of which $(\xi, \eta, \zeta)$ is the coordinates in the local coordinate system[6]. After finishing, the coordinates are projected to each axis, and the mapping relationship is

$$u = \sum_{i=1}^{8} N_i(\xi, \eta, \zeta) x_i \quad v = \sum_{i=1}^{8} N_i(\xi, \eta, \zeta) y_i \quad w = \sum_{i=1}^{8} N_i(\xi, \eta, \zeta) z_i$$

In the formula, the shape function is

$$N_i(\xi, \eta, \zeta) = \frac{1}{8}(1 + \xi_{i})(1 + \eta)(1 + \zeta) \quad (i = 1, 2, \cdots, 8)$$

As shown in figure 2, $\xi_i, \eta_i, \zeta_i$ is the node position coordinates of each unit, and the value is $\pm 1$. The symbol is determined by the quadrant of the local coordinate system where the node is located. The element-specific shape function matrix can be obtained from the above formula, namely

$$u = \begin{bmatrix} u & v & w \end{bmatrix}^T = N \cdot q^e$$

$$N = \begin{bmatrix} N_1 I & N_2 I & \cdots & N_8 I \end{bmatrix}$$

In the formula, $I$ is a unit matrix of order 3.

2.2. Element stiffness matrix

A geometric equation for solving a space physical problem in elastic mechanics, and a numerical expression of the displacement field of an element, that is

$$\varepsilon(x, y, z) = \begin{bmatrix} \varepsilon_{xx} & \varepsilon_{yy} & \varepsilon_{zz} & \gamma_{xy} & \gamma_{yz} & \gamma_{xz} \end{bmatrix}^T = [\partial] \cdot u$$

In the formula, $[\partial]$ is the operator matrix of the geometric equation, that is
\[\frac{\partial}{\partial x} 0 0 \frac{\partial}{\partial y} 0 \frac{\partial}{\partial z}\]  
\[0 \frac{\partial}{\partial y} 0 \frac{\partial}{\partial x} \frac{\partial}{\partial z} 0\]  
\[0 0 \frac{\partial}{\partial z} 0 \frac{\partial}{\partial y} \frac{\partial}{\partial x} \]  
(9)

From the above formula can be obtained

\[\varepsilon(x,y,z) = [\partial] N(x,y,z) q^e = B(x,y,z) q^e\]  
(10)

Moreover, due to the physical equation of space problem in the process of elasticity problem, an expression of stress field can be obtained

\[\sigma = D \cdot \varepsilon = D \cdot B \cdot q^e = S \cdot q^e\]  
(11)

In the formula, \(D\) is the elastic coefficient matrix. After obtaining the geometric matrix \(B(x,y,z)\) of an element, the element stiffness matrix of the element can be obtained, and its calculation formula is

\[K^e = \int B^T DB \, d\Omega\]  
(12)

Since the shape function is not the explicit expression of \(u, v, w\), the Jacobian matrix of spatial problem is introduced[7], and the calculation formula is

\[J = \sum_{i=1}^{8} \begin{bmatrix} \frac{\partial N_i}{\partial \xi} x_i \frac{\partial N_i}{\partial \eta} y_i \frac{\partial N_i}{\partial \zeta} z_i \\ \frac{\partial N_i}{\partial \eta} \end{bmatrix} \]  
(13)

The partial derivative of the shape function to the coordinate with dimension 1 is

\[\left[\frac{\partial N_i}{\partial \xi} \frac{\partial N_i}{\partial \eta} \frac{\partial N_i}{\partial \zeta}\right] = \frac{1}{8} [\xi_i(1+\eta,\eta)(1+\zeta,\zeta) \eta_i(1+\xi,\xi)(1+\zeta,\zeta) \xi_i(1+\xi,\xi)(1+\eta,\eta)]\]  
(14)

The partial derivative of the shape function to the whole coordinate is

\[\left[\frac{\partial N_i}{\partial x} \frac{\partial N_i}{\partial y} \frac{\partial N_i}{\partial z}\right] = J^{-1} \left[\frac{\partial N_i}{\partial \xi} \frac{\partial N_i}{\partial \eta} \frac{\partial N_i}{\partial \zeta}\right]\]  
(15)

Each element in the geometric matrix \(B\) can be obtained by the above formula.

In the triple definite integral area of the element, the element stiffness matrix of the spatial problem is

\[K^e = \int \int \int_{-1}^{1} B^T DB \, d\xi \, d\eta \, d\zeta\]  
(16)

The equivalent node load matrix is

\[F^e = \int N^T \bar{b} \, d\Omega + \int N^T \bar{p} \, dA\]  
(17)

In the formula, \(\bar{b}\) is the unit average volume force and \(\bar{p}\) is the unit average surface force.

2.3. Overall stiffness equation

Above, we have expressed the basic variable \((u, \varepsilon, \sigma)\) of the element by directly using the basic displacement variable array \(q^e\) based on the element node, and substituted it into the potential energy expression of the element, namely
\[ \Pi^e = \frac{1}{2} \int_{\Omega^e} \sigma^T \varepsilon d\Omega - \int_{\delta^e} b^T u d\delta - \int_{S^e} \bar{p} u dA = \frac{1}{2} q^T K^e q^e - F^e q^e \] (18)

Taking the first-order nonlinear extremum of the above equation for all nodal displacements \( q^e \), the element stiffness equation of a model element is obtained

\[ K^e q^e = F^e \] (19)

After the stiffness matrix \( K^e \) is obtained, the overall stiffness matrix \( K \) of the model can be obtained by regular combination, and then the stress and strain of the model can be solved by calculating the overall balance equation \( Kq = F \) in the model.

The element stiffness equation can also be assembled into the overall stiffness equation. The assembly rule is similar to the way that the element stiffness matrix is combined into the overall stiffness matrix, and the expression is

\[ Kq = F \] (20)

The nodal displacement can be obtained as

\[ q = \frac{K}{F} \] (21)

Through the node displacement, the stress, strain and other parameters of the model can be obtained.

3. Programming and case analysis

Based on MATLAB platform, the finite element analysis program is designed to analyze the bridge stress. In the design process, the calculation accuracy is improved to a certain extent by symbolic operation and unconventional numerical operation. In addition, various parameter expressions that are often used in pre-calculation, mainly including elastic coefficient matrix, Jacobian matrix and element geometric matrix, reduce a lot of repetitive work, improve the accuracy and operation speed of finite element analysis method, and achieve the effect of querying and displaying multiple stress weaknesses through a certain degree of accuracy range control. At the same time, in order to ensure good interactive performance, also designed a friendly, intuitive interface. In the following, the program is verified and analyzed by using the double-pillar support bridge model, and the results are compared with those of ABAQUS software.

3.1. Basic Situation of Double-prop Supported Bridge Model

For a plate-shell structure bridge, 326 nodes and 128 units are formed after dissection, as shown in the following diagram.

![Model Structure, Constraint, Load Diagram](image)

The model selects linear elastic material, its Young's modulus \( E = 1 \times 10^6 \text{ MPa} \), Poisson's ratio \( \mu = 0.25 \).

The constraint condition of the model is the complete constraint at the bottom of the bearing, and the load is its own gravity.
3.2. Mises stress nephogram based on original diagram and deformation diagram
Figure 4(a) is the Mises stress nephogram based on the original diagram obtained from the analysis of this program, and figure 4(b) is the analysis result of ABAQUS software.

![Figure 4. Comparison of Mises stress nephogram of model (original drawing)](image)

3.3. Stress weakness display
Figure 6 (a) is the schematic diagram of stress weakness obtained by finite element analysis using this program, and figure 6 (b) is obtained by ABAQUS software analysis.

![Figure 6. Comparison of model stress weaknesses](image)
It can be seen that ABAQUS analysis software can only find a weak point of stress in the case of more than two maximum Mises stress nodes, and this program can effectively avoid this problem by setting a certain precision control, and can mark nodes greater than the given Mises stress value. Such as this model is marked by Mises stress greater than 5000 nodes.

3.4. Analysis time comparison
Time consuming analysis of this program: 0.9484 seconds;
Time consuming of ABAQUS software analysis: 0.3 seconds

4. Conclusion
In this paper, the finite element analysis program is designed to realize the independent control of the finite element analysis of the eight-node hexahedral element in the whole process of the simply supported beam bridge, and also provides some special optimization for the analysis of the bridge structure. The program has the following advantages: First, the analysis accuracy is high. ABAQUS software and the program are used to analyze the results of the same model. Through visualization and comparison of the results, it can be clearly seen that the similarity of the analysis results is high. By comparing the node displacement data of the model, it is found that the deviation of this program relative to ABAQUS software is within 8.31%, indicating that the program analysis accuracy is good. The second is the program volume lightweight. This program is packaged into desktop executable file only 0.2% of the volume of ABAQUS software. Third, the analysis results are complete. When analyzing the stress weakness of the model, the problem that ABAQUS software can only display single stress weakness in the presence of multiple maximum stress points is solved by precision control.

Next, the following aspects of improvement need to be noticed. One is to further expand the unit type. In order to adapt to different models, different analysis unit types are needed according to their characteristics. Therefore, it is far from enough to use only eight-node hexahedral elements, and it is necessary to compile the calculation classes of other element types. Also consider the relationship between the model components. The second is to further optimize the analysis algorithm. In order to cope with the complex large structure in the future, the algorithm should be further optimized to reduce the time and space complexity of the algorithm.

References
[1] Yue, J.H., Li, R., Zhang, W.B., et al. (2020) Stress Calculation Method of Prestressed Bending Simply Supported Steel - Concrete Composite Beam Bridge. Journal of Civil Engineering and Management., 37(05): 164-168.
[2] Wang, Y.F. (2015) Calculation Method of Measured Stress and Realization of MALAB in Bridge Construction Monitoring. Transportation Technology and Economy., 17(06):117-121.
[3] Chen, G., Yin, X.F., Liu, J. (2018) Stress Analysis of Bridge Structure under Static Load. Highway and motor transport., (02):144-148.
[4] Zeng, P. (2009) Basic Course of Finite Element. Higher Education Press, Beijing.
[5] Li, Y.Z., Zhao, M.Y., Wang, X.P. (2004) Foundation and Program Design of Finite Element Method. Science Press, Beijing.
[6] Guo, J.T., Xue, Q.W. (2020) Finite Element Method and MATLAB Programming. Mechanical Industry Press, Beijing.
[7] Meng, K.J., Yu, Y. (2008) 3D-FEM Programming Based on MATLAB—Take 8-node Hexahedral Elements as an Example. In: Sichuan Institute of Mechanics 2008 Academic Conference. Sichuan. pp. 99-102.