Cracking Analysis of the Cable Saddle in Extradosed Bridges Based on the Extended Finite Element Method

Shi-Feng LIN[1,a], Qiao HUANG[1,b*], Shi-Xiang HU[1,c], Xue-Feng HU[2], Yi-He LIU[2]

[1]School of Transportation, Southeast University, Nanjing, China

[2]Jilin Highway Planning and Design Institute, Changchun 130021, China

[a]linshifeng1919@hotmail.com, [b]qhuanghit@126.com, [c]93892336@qq.com

*Corresponding author

Keywords: Extradosed Bridge, Concrete Pylon, Extended Finite Element Method (XFEM), Nonlinear Analysis.

Abstract. Cable pylons are important components of extradosed bridges. However, cable pylons pose difficulties in design and analysis because of their complicated loading conditions in the anchorage zone. In the present paper, taking the super-large Ningjiang Songhua River Bridge as the reference, a theoretical study of the crack resistance of concrete pylons in serviceability stage, and a numerical simulation analysis of the crack condition and stress distribution below the cable saddle constructed of parallel steel pipe are conducted using extended finite element method (XFEM) and the material nonlinear finite element method. The results obtained could provide a reference for the design and reinforcement of the pylon of similar bridges.

Preface

The extradosed bridge, also called partial cable-stayed bridge, is a new-type of bridge intervenient the cable-stayed bridge and the continuous beam bridge. Given that its pylon is relatively low and the proportion of load borne by its stay cable is lower than that borne by the stay cable of the cable-stayed bridge, the mechanical properties of the extradosed cable-stayed bridge are closer to those of the continuous beam bridge and the continuous rigid frame bridge. In the course of design, the extradosed bridges are provided with relatively high-rigidity main beams and relatively small amount of stay cables, thus, both the cable force of stay cables and the increment of their cable force under the live load are relatively low. For the convenience of construction and to control the pylon size, layered cable saddle is usually adopt to anchor stay cables onto the pylon. Among more than 30 prestressed concrete extradosed bridges constructed nationwide, most of them adopt the anchoring scheme of setting a cable saddle on the pylon. In the case of the cable saddle-structured pylon due to its relatively small size, it is difficult to configure effective circumferential prestressed reinforcement in pylons, so the pylons are usually designed as reinforced concrete members. However, in contrast to the mature design methods of reinforced concrete beams and columns, the cable saddle has a relatively complicated stress-concentrated local response which is influenced by its size and shape. A mature design and computation method in the cable saddle have not been developed yet. Thus, it is necessary to conduct a detailed nonlinear finite element analysis of the pylon of extradosed bridges in order to study the rule of stress distribution of the concrete below the saddle, particularly the pattern of crack distribution around the saddle, thus providing a theoretic basis for its local reinforcement design.

The extended finite element method (XFEM) is a computational method which can simulate the patterns of cracks as well as their initiation and extension along any direction. Moreover, for reinforced concrete structures which are mainly subjected to local tensile failure, computation using XFEM can visually reflect their crack distribution and the extension direction of these cracks properly.

Nonlinear Analysis Methods of Reinforced Concrete Structures

The current methods used in crack analysis of reinforced concrete structures mainly adopt a fracture mechanics or damage mechanics-based theoretical method. The numerical models of concrete cracks based on fracture mechanics theory can be divided mainly into the following two types, i.e., discrete crack model and smeared crack model.[1] The discrete crack model is the earliest crack model which simulates the cracking of concrete. The main limitation of this model is that the finite element mesh must be rezoned with crack propagation, which results in a considerable workload and produce some problems between the rezoned mesh and the original mesh. Thus, the applications of the discrete crack model are very limited. However, in the case of the smeared crack model, rezoning the mesh is unnecessary, and the influence of cracks is taken into consideration by adjusting the rigidity matrix of the elements. Presently, the model has already been integrated in some large commercial software like ANSYS, ABAQUS, etc. However, the smeared crack model lacks a discontinuous displacement pattern, which may easily cause shear locking if the mesh boundary is not parallel to the principal stress. Therefore, converge problems often occur in the use smeared crack model. Concrete damage mechanics, the subject which applies damage mechanics theory to the analysis of concrete structures, started in 1980s. The studies of damage theory on both micro scale and macro scale have made some progresses till date. In the current stage, the macro damage theory based on continuum damage mechanics is still one of the means of computation as concrete is assumed as an anisotropic material of poor homogeneity and high discreteness. This theory assumes that the material is homogeneous and continuous and the cracks are uniformly distributed, which also introduces a damage variable to construct the constitutive model and applies the damage evolution equation to describe the mechanical behavior of the material. Nonetheless, similar to the case of the smeared crack model based on the assumptions of continuum mechanics, the damage model cannot yet take the directionality of the damage variable or some mechanical parameters of the crack plane into account when the simulated concrete reaches complete damage and forms macrocracks. Thus, applying only the damage model to the failure analysis of concrete structures still possesses certain problems.

The extended finite element method (hereinafter for short as XFEM) was put forward by Belytschko et al[2], in 1999, based on the idea of partition of unity. By virtue of concept of partition of unity, the displacement function of the finite element method has been improved, in XFEM. Moreover, by introducing an additional function representing local discontinuity, the displacement field of XFEM may be constituted by a displacement function which represents the discontinuity after crack connection and the stress singularity near the crack tip, as illustrated below:

$$u=\sum_{i=1}^{N}N_i(x)[u_i+H\ x\ a_i+\sum_{a=1}^{4}F_a(x)b_I^a] \tag{1}$$

Where $N_i(x)$ represents the shape function of the element; u_i represents the nodal displacement vectors of the elements; a_i represents the nodal additional degrees of freedom (DOF) vector; $H(x)$ is the Heaviside function, as shown below:

$$H(x)=\begin{cases}1 & (x-x^*)\cdot n\geq 0\\ -1 & (x-x^*)\cdot n<0\end{cases} \tag{2}$$

Where x represents Gaussian integral point; x^* represents the point closest to x on the crack; n represents the element outer normal vector of the crack at x^*, b_I^a represents the nodal additional degree of freedom (DOF) vector near the crack tip; $F_a(x)$ is an approximate expression of the displacement field function of the crack tip in the (r,θ) polar coordinate system which takes the crack tip as the origin of coordinates. It is obtained from the approximate displacement field of the crack tip by the linear elastic fracture mechanics, and consists of four basic functions, as shown below:

$$F_a(x) = \sqrt{r}[\sin\frac{\theta}{2},\ \cos\frac{\theta}{2},\ \sin\theta\sin\frac{\theta}{2},\ \sin\theta\cos\frac{\theta}{2}] \tag{3}$$

The XFEM does not need to take into account the consistency of the boundary of the finite element mesh with the crack plane when analyzing a fracture problem compared to the finite element methods. And there is no need to rezonize the mesh in the case of crack propagation. So it is one of the most effective methods for solving discontinuity problems. In this paper, XFEM and the concrete plastic damage constitutive model-based finite element method were respectively employed to conduct a nonlinear crack resistance analysis of the pylon of extradosed bridges.

Computation and Analysis

Engineering Background

Ningjiang Songhua River Bridge, located in the northeast of Jilin Province, is an extradosed cable-stayed bridge (also called partial cable-stayed bridge) which adopts a continuous structure with 4 pylons and 5 spans. The span arrangement is 95m+3×150m+95m; the ratio of side span to main span is 0.63; the prestressed concrete box girder is provided with variable cross-section beam. Its depth ranges from 3.0m to 5.5m, and the depth-span ratio ranges from 1/50 to 1/27. The pylon height is 21.5m, and the ratio of pylon height to span is 1/7. It adopts the single cable plane-double row cable form of stay cable arrangement, and one stay cable consists of 37 steel strands of 15.24mm in size. The cable saddle adopts the structure type of parallel steel pipe, as shown in Fig.1.

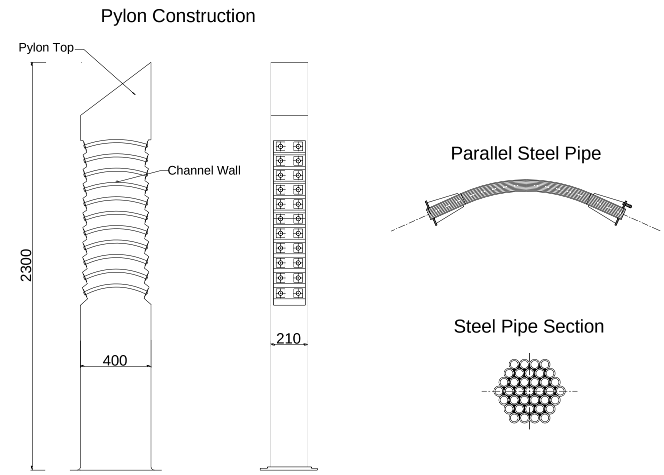


Fig. 1 Pylon and Parallel Steel Pipe Construction Layout

Introduction to the Computational Models

In order to accurately determine the boundary conditions of pylons, firstly the general software MIDAS/CIVIL was adopted to build the member model of the full bridge (see Fig.2) to determine the cable force of each stay cable under the designed load combinations. The results of cable force computation are presented in Tab.1. In this table, the tensioning cable force was taken from the design drawings, while the working cable force was derived from the computation results obtained by using MIDAS. Secondly, the general software ABAQUS was adopted to build the overall spatial solid model of pylons and the local model of cable saddles in order to reduce the computation time. Then linear static analysis and nonlinear analysis were conducted for pylons and cable saddles, respectively. A submodel was employed between the overall spatial finite element model and the local model for variable mapping.

Introduction to MIDAS Model. A full bridge is modeled with spatial beam elements and cable elements, consisting of 923 elements and 752 nodes in total. Concretely, the girder was divided into 560 elements; pylons were divided into 164 elements, and main piers were divided into 23 elements. In order to take into account construction process and relevant loads, 78 construction stages were simulated. Considering various designed loading conditions in service stage, the possible cable forces under service states were computed, and the cable force obtained under the most unfavorable loading condition combination was adopted as the computed cable force of the pylon model.

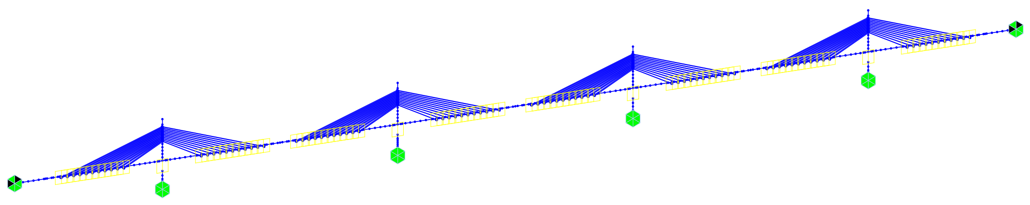
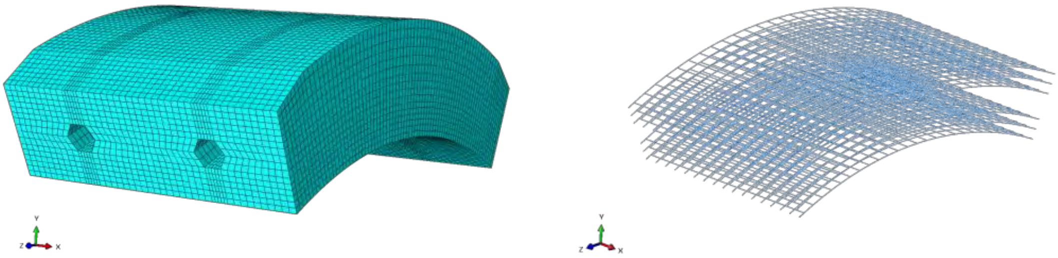


Fig. 2 Computation Model of MIDAS

Introduction to ABAQUS Overall Pylon Model. In the ABAQUS model, concrete members were modeled using the spatial eight-node element C3D8, and the bottom boundary of pylons was fixed. The parallel steel pipe structure was adopted so that the cable saddles could uniformly transmit cable force to channel walls. Thus, cable force could be converted into uniform area load applied on the lower half of each channel. The C50 concrete with an elasticity modulus of 34.5GPa and a Poisson's ratio of 0.2 was used. The overall pylon model was divided into 192,263 elements and 211,247 nodes.

Introduction to ABAQUS Local Cable Saddle Model. The cable saddle was modeled using commonly-used reinforcement and concrete models. Coupling constraints were used to simulate the interactions on the interfaces. Concrete members were modeled using the spatial eight-node element C3D8, while the reinforcement was modeled using truss element T3D2. The concrete adopted the spatial eight-node element C3D8 simulation, and the reinforcement adopted the truss element T3D2 simulation. The upper and lower boundaries of the cable saddle model were defined as the boundaries of the submodel to present the results of the overall pylon model. The local model was divided into 59,088 C3D8 elements and 6,672 T3D2 elements. XFEM and the concrete plastic damage constitutive model-based finite element method were adopted for the computation purpose.

Fig. 3 Pylon Finite Element Model Using ABAQUS (section)



1) Local Concrete Model   2) Local Reinforcement Model

Fig. 4 Local Finite Element Model of the Cable Saddle

While analyzing the cable saddle model using XFEM, it was assumed that the concrete was a quasi-brittle material, or the fracture behavior of the concrete could still be described by the linear elastic fracture criteria of brittle materials after appropriate modification. The concrete material would crack when reaching the maximum principal tensile stress as given by the following formula:

$$f = \left\{ \frac{\langle \sigma_{\max} \rangle}{\sigma^{0}_{\max}} \right\} \tag{4}$$

Where $\sigma_{\max}$ represent the maximum principal stress; $\sigma^{0}_{\max}$ is the maximum allowable principal tensile stress, and the present paper sets the standard value of the tensile strength of the C50 concrete at f_{tk}=2.64MPa. When the material reached its $\sigma^{0}_{\max}$, cracks began to emerge. Computation and analysis results indicated that the principal compressive stress of the model reached the standard value of the tensile strength of the C50 concrete, and the compressed concrete still lay within its elastic working range. In the softening stage of cracking, the cohesive crack model was adopted to simulate the mechanical behavior of the crack plane. The softening curve attributed the exponential type curve which employed the fracture energy as the complete separation criterion of the crack plane. The fracture energy G^{I}_{f} was set at 140N/m.[3]

While using the concrete plastic damage constitutive model-based finite element method for analysis, the concrete damage evolution equation selected the concrete uniaxial tensile/compressive damage evolution equation recommended by literature [？], as given below:

$$d_t = \begin{cases} 1-\rho_t(1.2-0.2x^5) & x \le 1 \\ 1-\dfrac{\rho_t}{\alpha_t(x-1)^{1.7}+x} & x > 1 \end{cases} \tag{5}$$

$$x = \frac{\varepsilon}{\varepsilon_t} \tag{6}$$

$$\rho_t = \frac{f_{tk}}{E_c \varepsilon_t} \tag{7}$$

Where α_t represents the parameter value of the descending portion of the tensile curve, and was set to 2.2; ε_t represents the tensile strain of the concrete peak value which was set to 110$\mu\varepsilon$.

The compressive damage evolution equation is given below:

$$d_t = \begin{cases} 1 - \dfrac{\rho_c n}{n-1+x^n} & x \le 1 \\ 1 - \dfrac{\rho_t}{\alpha_c (x-1)^2 + x} & x > 1 \end{cases} \tag{8}$$

$$x = \frac{\varepsilon}{\varepsilon_c} \tag{9}$$

$$\rho_t = \frac{f_{ck}}{E_c \varepsilon_c} \tag{10}$$

$$n = \frac{E_c \varepsilon_c}{E_c \varepsilon_c - f_{ck}} \tag{11}$$

where αt represents the parameter value of the descending portion of the compressive curve, and was set to 2.48; εt represents the compressive strain of the concrete peak value, and was set to εc=1920με (C50 concrete) in the case of uniaxial compression.

Analysis of Results

The model was sliced into thin layer in order to monitor precisely. It can be seen from the principal stress nephogram (Fig.5) that the principal tensile stress of the cable saddles was mainly distributed below the channel, and the maximum principal tensile stress occurred on the arc roof of the cable saddle channel of cable A1 with the numerical value of 5.78 MPa, which already exceeded the tensile strength of the C50 concrete, incorporating the cracks in concrete. It can be observed from the principal tensile stress vectogram (Fig.6) that the direction of the principal tensile stress was basically perpendicular to the longitudinal axis of the channel, so it could be concluded that the concrete below the channel wall would form cracks along the pipe. The maximum principal compressive stress of the cable saddle zone occurred in the corner of the cable saddle reached a numerical value of 23.79 MPa, caused by stress concentration in the corner. However, since the value did not reach the standard value of the compressive strength of the concrete (fck=32.4MPa), the concrete would not reach compression failure.   .

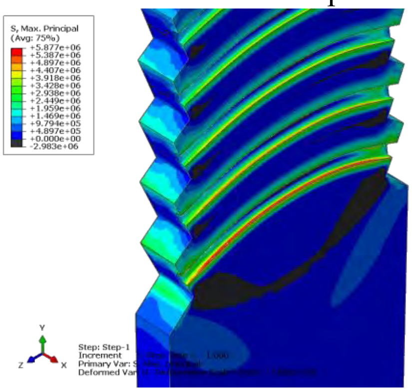


Fig. 5 Principal Tensile Stress of the Saddle

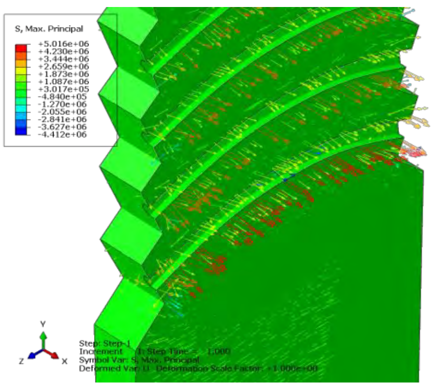


Fig. 6 Principal Tensile Stress Vector of the Saddle

Fig.7 shows the crack propagation process computed by XFEM. It can be seen from Fig.7 that a crack first occurred on the arc roof where the principal tensile stress reached its maximum value under the load. With the further progressive increase in load to about 58% of the maximum cable force, gradual crack propagation occurred at the bottom along the pipe until the entire channel got connected. When the load was increased to the maximum designed cable force, the radial crack depth along the pipe reached up to 3~5 elements. It is noted that there would be a crack on the top surface of the channel along the pipe which would propagate upward if the load was continuously increased. If the radial penetrating crack on the bottom surface of the channel of the upper-layer cable saddle occurred, the durability of the pylon structure would be influenced.

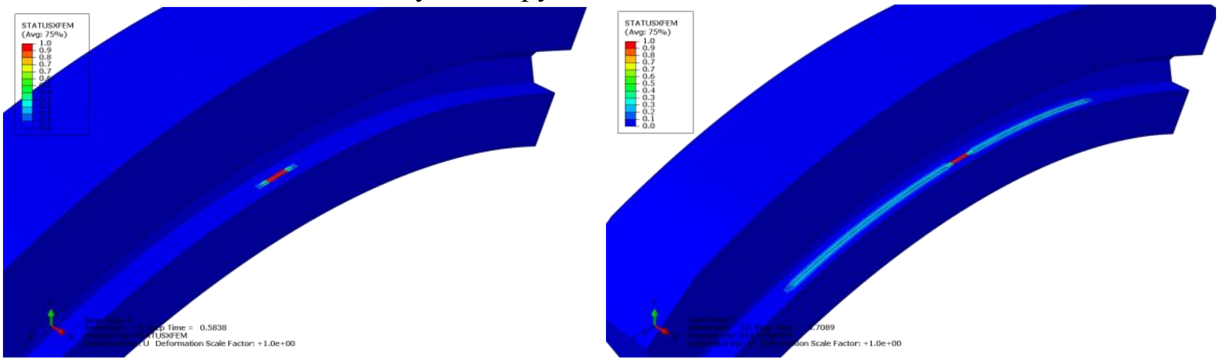
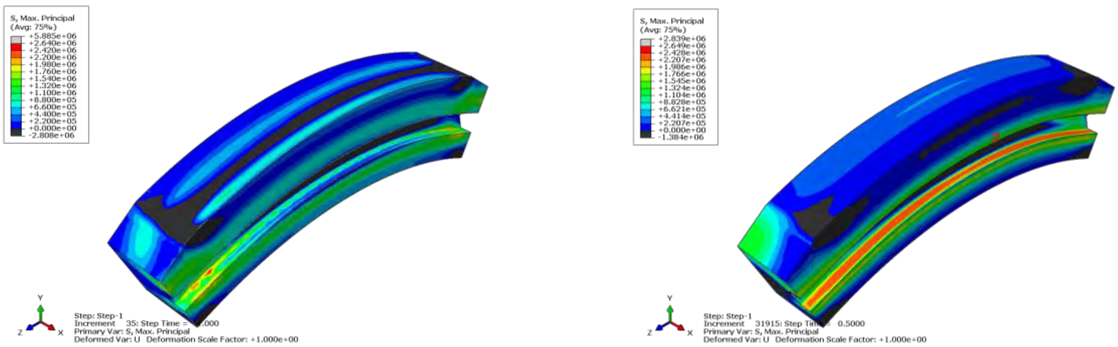


Fig. 7 Longitudinal Crack Propagation along the Pipe

Fig.8 presents the principal tensile stress diagrams computed by XFEM and FEM, respectively. From the principal tensile stress distribution diagrams, it is indicated that there are certain differences in the distribution of the principal tensile stress on the channel surface of the cable saddle. The stress computed by XFEM was less than that computed by FEM. This might be due to following reasons: 1) after the formation of cracks, the stress in the concrete is released. According to the adhesive crack theory of concrete,[5] the numerical value of the residual stress is determined by the ratio of the fracture energy of the concrete to the virtual width of the crack. The stress value of concrete is determined by the concrete constitutive model's softening segment curve and the damage evolution equation after the occurrence of tensile damage. These values are usually determined through experiments. 2) The damage constitutive model is still based on the assumptions of continuum solid mechanics, and can conduct the simulation of microcracks in concrete in a relatively accurate manner. However, if there are distinct macrocracks or displacement field discontinuity, it will be very difficult to evaluate the accuracy of the computed results. According to literature,[3] the XFEM which can be incorporated into the discontinuous displacement function can more reliably solve such problems. But as far as non-cracked areas are concerned, the linear elastic material constitutive model is still applied, which may cause errors to the stress field computation of non-crack-connected areas. Thus, if XFEM and the damage model are combined, the results of computation will be more accurate. Nevertheless, given that both of them are highly nonlinear processes, employing an implicit solver to solve such problems may make it difficult to converge, and the computation time consumed by the model will be very long.



(1) Principal Tensile Stress by XFEM (2) Principal Tensile Stress by FEM

Fig. 8 Principal Tensile Stress Distribution on the Cable Saddle by Different Methods

Conclusions

The present paper takes into account the cable saddle zone of the pylon of the super-large Ningjiang Songhua River Bridge as the object of the study. The linear elastic finite element method, the material nonlinear finite element method, and the extended finite element method were adopted to conduct an analysis and comparison of a cable saddles and to study the crack distribution and stress field of cable saddles. Following conclusions were drawn from the study.

1) Under the action of the cable force of the parallel steel pipe, the cable saddles would generate a splitting crack which would run through the entire channel, and the crack had a depth of 3~5 elements. It is suggested that the diameter of the reinforcement could be increased or the distance between horizontal reinforcement layers could be reduced in this range in order to improve the crack resistance of the cable saddles.

2) In the overall design of extradosed bridges, it was suggested that the height of the pylon and the length of the cable-free zone should be comprehensively taken into consideration, thus to increase the anchorage radius of the stay cables in the cable saddles and reduce the radial force applied on cable saddles. There would be a stress concentration in the corner of the cable saddles, where the stress value had exceeded the design value of the tensile strength of the C50 concrete. Therefore, it is suggested that the amount of ordinary reinforcement should be increased and that chamfers and fillets should be added, thus to reduce stress concentration in the corner.

3) Extended finite element method was adopted in this paper to conduct a detailed analysis of the reinforced concrete cable saddle structure. It could predict the positions of possible cracks on the structure and the directions of crack propagation without presupposing the cracking path, and thus overcoming the deficiency of the classical finite element method.

4) The results of the computation of crack distribution and propagation by extended finite element method and the damage constitutive model-based nonlinear finite element method were basically consistent. However, differences were observed in the results of the computation of stress fields near crack boundaries, which should be improved in future studies.

Reference

[1]Hillerborg A. Modéer M. Peterson P E. Analysis of crack propagation and crack growth in concrete by means of fracture mechanics and finite elements [J]. Cem. Concr.Res, 1976, 6: 773-782.

[2]Belytschko T. Black T. Elastic crack growth in finite element with minimal remeshing [J]. International Journal for Numerical Methods in Engineering, 1999, 45: 601—620.

[3]Fang Xiujun. Jin Feng. Wang Jinting. Simulation of mixed-mode fracture of concrete using extended finite element method [J]. Engineering Mechanics, 2007, 24(supplement 1):46-52.

[4]Fang Xiujun. Jin Feng. Extended finite element method based on abaqus [J]. Engineering Mechanics, 2007, 24(7):6-10.

[5]Jiang Jianjing. Lu Xinzheng. Ye Lieping. Finite element analysis of concrete structures [M]. Beijing: Tsinghua University Press, 2005.

[6]GB50010-2010, Code for concrete structure design [S]. Beijing: China Architecture & Building Press, 2011:15-18.

[7]ABAQUS (Vesion 6.9) Analysis User's manual [Z]. ABAQUS Inc, 2004.

[8]Chen Congchun. Study on Major Problems for Design Theory of Extradosed Cable-stay Bridges [D]. Shanghai: Tongji University, 2005:20-30.

[9]Zheng Yifeng. Huang Qiao. Zhang Lianzhen. Parameter Analysis for Partial Cable Stayed Bridges [J]. Journal of Highway and Transportation Research and Development, 2006, 23(6): 60-61.

[10]Zheng Yifeng. Huang Qiao. Sun Yongming. Research on rational state of accomplishment of partial cable-stayed bridge [J]. Journal of Highway and Transportation Research and Development, 2006, 23(11): 60-61.