Analysis of crack propagation in longitudinal beam of high-pile wharf during winter temperature decrease period

Z L Huang\textsuperscript{1}, Z Y Huo\textsuperscript{1} and Z Y Huang\textsuperscript{1,2}
\textsuperscript{1}Zhejiang Ocean University, Zhoushan 316000, China

E-mail: 735539477@qq.com

Abstract. Ports play an increasingly important role in economic development and are the core strategic advantages that drive regional economic development. As hydraulic structure, the working environment of the wharf is relatively complicated. There are many factors affecting its normal use. Visible or invisible cracks are common in wharf components. Temperature stress change is the main factor for the formation of cracks on the surface of concrete beams. When affected by temperature changes, existing fine cracks will connect, expand, and even penetrate the entire component. It will not only affect the aesthetics of the wharf, but also affect its service life, and bring considerable economic losses to later maintenance. Therefore, it has certain reference value and significance to study on crack propagation for assessment of safety, reliability and durability of engineering structures. In view of this, this paper carries out numerical analysis on crack propagation of reinforced concrete longitudinal beams of a high-piled wharf in China with ABAQUS software and extended finite element method (XFEM). In the simulation process, only temperature drop factor is considered. At the same time, this paper introduces the research progress and some basic theories of XFEM at home and abroad. The results show that the numerical simulation results are similar to the crack opening degree measured by the seam gauge. XFEM can effectively describe the crack propagation in concrete, which is more accurate and feasible than the conventional finite element method (CFEM).

1. Introduction
Port construction in China develops rapidly and makes important contributions to the development of national economy. The high pile structure is a kind of structural form commonly used in wharf engineering. It can adapt to complex geological conditions and unfavorable natural environment, and is widely used in coastal and inland river port projects in our country. The superstructure of the high-pile wharf directly withstands the vertical and horizontal loads acting on the wharf and transmits it to the pile foundation. The pile foundation is used to support the superstructure and transfer the load of the superstructure and the wharf surface to the depth of the foundation, and also to stabilize the bank slope. It’s normal to work with cracks for the high-pile wharf. Concrete members are subjected to various loads during construction and actual use. The tensile strength is much smaller than compressive strength. Cracks are generated under the action of small tensile stress. The development of cracks will affect the durability and safety of concrete components. Under the action of load, the crack tip will produce great plastic deformation, which will promote the crack passivation. The open displacement of the crack tip is determined by the passivation of the crack tip. When the pre-crack tip is passivated to a certain extent, plastic crack will occur. The plastic crack further expands to a critical size and a fast cleavage fracture occurs. Obviously, the existence of cracks destroys the integrity of the
terminal. Over time, it will have a major impact on its structural form, and even serious damage to the safe operation of the wharf.

The conventional finite element method has defects in simulating cracked structures. The uncertainty of crack propagation makes the calculation more difficult and takes longer when meshing. The XFEM inherits all the advantages of CFEM. The idea of unit decomposition method guarantees the convergence of XFEM. It increases the discontinuous function reflecting the crack surface in the displacement field and the asymptotic displacement field function reflecting the local characteristics of the crack tip, and reflects the existence of cracks, so that the crack and the finite element model are independent of each other, overcoming the difficulties caused by high-density meshing at the high stress and deformation concentration areas at the crack tip. It’s often considered to be the most effective numerical analysis method for crack treatment. It has been rapidly developed and applied in just a few years.

In the finite element analysis of reinforced concrete structures, the crack treatment of components has been a hot and difficult point in the field. The cause analysis and Countermeasures for the Longitudinal Girder Crack of Piled Wharf [1] investigated and detected the longitudinal girder crack of piled wharf, and the investigation and detection results showed that temperature deformation was the main factor leading to the transverse crack of the longitudinal girder of wharf. Research on Monitoring Model of Crack Opening and Fitting of Piled Wharf [2] studied the monitoring model of crack opening and fitting of piled wharf, and drew the conclusion that crack formation and expansion are extremely sensitive to temperature change. According to the above literature analysis results, this paper mainly considers the temperature factor when analyzing the crack expansion problem in the winter cooling period of the wharf. Therefore, based on the analysis and summary of the basic theory of XFEM, the ABAQUS extended finite element method is used to take a finite element model of a high-pile wharf reinforced concrete longitudinal beam member [3] in China, and the finite element model is established to determine the model material parameters and the temperature. The field calculation results are applied as a load to the model to study the progressive cracking failure process of the longitudinal beam under the initial temperature, which is useful for the design and safety assessment of the concrete structure of the wharf. The calculation results verify that the XFEM has high accuracy and demonstrates its superiority, which provides a convenient way to solve practical complex problems. This paper also preliminarily prospects the XFEM this function still needs to be further solved.

2. Research progress of XFEM

2.1. Research progress of foreign XFEM

The theory of XFEM was first proposed by Belyschko [4] in 1999, a professor at Northwestern University in the United States. This is a new finite element method that can effectively solve the problem of discontinuous mechanics. Due to its unique advantages in analyzing discontinuities, it has developed rapidly and is used in various industries and fields. Since the advent of the world, many scholars have improved and perfected the method, especially in the fracture problem, specifically the following aspects: (1) Improved accuracy of the crack tip field. Xiao and Karihaloo [5] combine XFEM and SAR to analyze the influence of branch function and enhancement range on the accuracy of crack tip field, and improve the accuracy of crack tip field. (2) Stability and convergence of numerical results. Laborde and Chahine [6] studied the effect of crack region on convergence; In order to avoid the stiffness of the stiffness matrix, Peter and Hack [7] cut some parts to strengthen the degree of freedom of nodes to avoid singularity of matrix. Ventura [8] successfully applied the standard Gaussian integral to the non-continuous cells with cracks, and these cells do not need to be divided into subunits. (3) It is an improvement of the calculation method of the stress intensity factor. Xiao and Karihaloo [9] proposed a method for calculating the stress intensity factor directly by applying extended finite element without post-processing. This method not only considers the cracked progressive displacement field, but also considers the high-order term. However this method of
directly calculating the stress intensity factor is inconvenient to use in simulating crack propagation, only apply in the stress intensity factor calculation of static crack. (4) Description of the crack interface and improvement of the tracking method. Sethia [10] applies the level set method to the description of the crack interface and combines it with the extended finite element idea to analyze the problem of more sophisticated crack interfaces. Chopp et al. [11] used the fast marching method and the XFEM to simulate the extended path of multiple cracks. The results show that works well.

2.2. Research progress of domestic XFEM

The XFEM has developed rapidly and is widely used abroad, but compared with domestic research, there are few studies on this issue. Dong [12] of Hohai University, used XFEM to conduct numerical simulation and research on cracking and propagation of cracks in concrete structures, aiming at the shortcomings and shortcomings of existing numerical methods in the current analysis of concrete cracking and failure, and with the unique advantages of discontinuity problem in XFEM analysis. Liu [13] uses a four-node isoperimetric element to establish a numerical solution format for the viscoelastic displacement field of I-type cracks, which provides a new way for the displacement field analysis of viscoelastic fracture problems. Yu [14] established an extended finite element model of closed friction cracks. The friction contact problem between crack faces was considered in the model. At the same time, the XFEM and linear complementary method were combined to open up a new way to solve the contact problem. Dong [15] and other researchers have given an extended finite element method for directly calculating the stress intensity factor. This method can directly calculate the stress intensity factor without post-processing, which greatly simplifies the pre- and post-processing.

3. Research progress of XFEM

The XFEM mainly adopts the idea of independent mesh to solve the crack propagation problem in finite element, which overcomes the shortcomings of CFEM requiring the crack surface and unit boundary to be consistent, and the mesh to be re-divided after the crack is expanded.

3.1. Characteristic

The features of the extended finite element method are as follows: (1) XFEM is an extension of the conventional finite element method based on the concept of unit decomposition; (2) Realizing the existence of discontinuities in the unit by increasing the degree of freedom of the displacement equation; (3) XFEM does not require re-meshing of brittle materials and is a positive and effective method for simulating the initiation and propagation of discrete cracks in any path dependent on a feasible solution; (4) Allow material and geometric nonlinearities; (5) XFEM can also be used to encircle the line of any static surface crack; (6) Allowing the crack surface to have a contact effect based on a small-scale slip equation; (7) XFEM is only universal for first-order stress or displacement solid continuous elements; (8) Discontinuation of mesh matching geometry is not required.

3.2. Unit decomposition method [3]

The unit decomposition method is the basis of the extended finite element. The principle is to cover all the solution regions with some sub-domains \( \Omega_i \) centered on node \( x_i \) that is the whole \( \Omega = \sum_{i=1}^{k} \Omega_i \).

Define a function \( \varphi_i(x) \) on each \( \Omega_i \) that is not zero only in this subregion. And meet the unit decomposition conditions: \( \sum_{i=1}^{k} \varphi_i(x) = 1 \). Then, function set \( \{ \varphi_i(x) \}_{i=1}^{k} \) can be called to solve the unit decomposition function in domain \( \{ \Omega_i \}_{i=1}^{k} \). For any function \( P(x) \), there is \( \sum_{i=1}^{k} j_i(x) P(x) = P(x) \).

Assuming that the function \( P_i(x) \) is an approximation function of the function \( f(x) \) in the sub-region \( \Omega_i \), then the global approximation of the function \( f(x) \) over the entire solution domain can be
taken as $f^h(x)$:

$$f^h(x) = \sum_{i=1}^{k} \phi_i(x)P_i(x)$$  \hspace{1cm} (1)

In the formula, $\phi_i(x)$ is a function that satisfies the unit decomposition. Any function that can approximate $f(x)$ in subdomain $\Omega$ can be used as approximation function $P_i(x)$, so that the approximation function in each subregion $\Omega$ approximates the real function as accurately as possible. Finally, the functions of each subdomain are integrated to obtain a global approximation.

3.3. Fracture criterion [3]

In the case of ABAQUS platform simulation degradation and final failure, the damage model is needed to specify the material properties of the crack propagation region. The failure mechanism includes the damage initiation criterion and the damage evolution criterion. The damage evolution of the material can only respond if the damage initiation criteria are satisfied. There are a total of six crack initiation criteria in ABAQUS, three based on stress and three based on strain, specifically maximum nominal stress (MAXS) and maximum nominal strain (MAXE), secondary nominal stress (QUADS) and secondary nominal strain (QUADE), maximum principal stress (MAXPS) and maximum principal strain (MAXPE). The initial judgment of the crack mainly includes two criteria: the maximum principal stress criterion and the maximum principal strain criterion.

The maximum principal stress cracking criterion can be expressed as $f = \left\{ \frac{\sigma_{\text{max}}}{\sigma_{\text{max}}^0} \right\}$, $\sigma_{\text{max}}^0$ represents the maximum tensile stress allowed by the material, that is, the maximum allowable principal stress, $\sigma_{\text{max}}$ is the maximum principal stress of the calculated material, $\{ \}$ is the Macaulay bracket to indicate that the pure compressive stress cannot cause damage. When the maximum principal stress reaches a critical value, the damage initiates.

The maximum principal strain cracking criterion can be expressed as $f = \left\{ \frac{\varepsilon_{\text{max}}}{\varepsilon_{\text{max}}^0} \right\}$, $\varepsilon_{\text{max}}^0$ represents the maximum tensile strain allowed by the material, that is, the maximum allowable main strain, $\varepsilon_{\text{max}}$ is the maximum principal strain of the calculated material, $\{ \}$ is the Macaulay bracket to indicate that the pure compressive strain cannot cause damage. When the maximum principal strain reaches a critical value, the damage initiates.

In the calculation process, if the formula of the above two cracking criteria is used, the direction of crack propagation is perpendicular to the direction of the maximum principal stress or the maximum principal strain. If the other four cracking criteria are used, you need to define the axis perpendicular to the crack surface. When the computation $f$ is satisfied $1.0 \leq f \leq 1.0 + f_{\text{tol}}$, it is considered that the initial damage criterion is satisfied. At this time, the damage evolution response of the material enters the softening stage, that is, additional cracks begin to expand or the length of existing cracks increase. $f_{\text{tol}}$ represents the allowable error. The default value in ABAQUS is 0.05.

The maximum nominal stress criterion is proposed in this paper.

4. Material constitution

4.1. Steel constitution [16]

The constitutive relationship of the steel bars is measured by a tensile test, and untreated steel bars or round steel bars are generally used. At present, the most commonly used constitutive models of steel bars include four types: ideal elastoplastic model, linear enhanced elastoplastic model (two-fold line model), double-fold elastic plastic model and ideal plastic-hardening plastic model, as shown in
figures 1 and 2. It is shown that these two typical types of steel constitutive models are adopted more. In the numerical simulation of this paper, the ideal elastoplastic model is preliminarily drawn up.

![Figure 1. Ideal elastoplastic model.](image1)

![Figure 2. Two-fold line model.](image2)

The ideal elastoplastic model can be used when the material flow phase is long and the structural strain is small, and the strengthening phase is negligible. The relationship between stress and strain is as follows:

\[ |\sigma| < \sigma_y \quad \varepsilon = \frac{\sigma}{E} \]

\[ |\sigma| \geq \sigma_y \quad \begin{cases} \varepsilon = \frac{\sigma}{E} + \lambda \text{Sign} \sigma, & \sigma d\sigma \geq 0; \\ d\varepsilon = \frac{d\sigma}{E}, & \sigma d\sigma < 0. \end{cases} \]  

\[ \text{Sign} \sigma = \begin{cases} 1 & \sigma > 0, \\ 0 & \sigma = 0, \\ -1 & \sigma < 0. \end{cases} \]  

In the formula, \( E \) represents the elastic modulus, \( \lambda \) is a parameter, and \( \text{Sign} \) is called a symbol function in mathematics.

4.2. Concrete constitution [17]

In this paper, according to the Concrete Structure Design Code (GB 50010-2010) to determine the uniaxial tension constitutive relationship of concrete, see the following formula:

\[ \sigma = (1-d) E \varepsilon, \]

\[ d_i = \begin{cases} 1 - \rho_i (1.2 - 0.2 \varepsilon^3), & x \leq 1; \\ 1 - \frac{\rho_i x}{\alpha_i (x-1)^{\frac{1}{3}} + x}, & x > 1. \end{cases} \]

\[ \rho_i = \frac{f_{\varepsilon,i}}{E \varepsilon}, \]

\[ x = \frac{\varepsilon}{\varepsilon_{\varepsilon,i}}. \]

In the formula, \( d_i \) represents the uniaxial tensile damage evolution parameter of concrete, \( \alpha_i \) represents the parameter value of the concrete uniaxial tensile stress-strain curve falling section, \( f_{\varepsilon,i} \) represents the uniaxial tensile strength value of concrete, and the actual value needs to be analyzed according to the structure, \( \varepsilon_{\varepsilon,i} \) represents the peak tensile strain of the concrete corresponding to the
uniaxial tensile strength representative value \( f_{tr} \).

5. Example analysis

5.1. Project overview
A high-pile wharf in China has a total length of 162 m and a width of 14 m. The total length of the approach bridge is 66 m and the width is 10 m. The wharf adopts high-pile beam-plate structure, and the pile foundation adopts 550 mm×550 mm hollow square column, which is divided into two types: oblique pile and straight pile. The cast-in-place beam is embedded in the pile cap, and the prefabricated stringer, prefabricated panel and paving layer are arranged on the beam. The pile foundation of the approach bridge adopts a prestressed hollow square column, the pile top adopts a 1000 mm×1000 mm pile cap, and a longitudinal and transverse beam is used between the piles. The top of the pile cap adopts a column, the cast-in-place cap beam is used on the column, the panel adopts a prefabricated panel, and a paving layer is laid on the panel. The longitudinal beam and the front beam are located in the water level change zone. Since the wharf has been used, cracks of different sizes have appeared. Therefore, a seam gauge is arranged to monitor the crack. Considering the limitations of conventional methods to establish crack models, this paper uses the XFEM to simulate crack propagation.

5.2. Extended finite element model of longitudinal beams
In this paper, the longitudinal beam of the front leg of the high-pile wharf is selected as the model. The longitudinal beams are arranged parallel to the shoreline of the wharf. The total length of the wharf platform is 162 m, and the longitudinal beam is divided into 36 spans. After considering the shelving width, the longitudinal beam length between the adjacent transverse truss is counted as 4 m. The geometric dimensions of the longitudinal beam are shown in figures 3 and 4, the top surface of the cross-section is 0.75 m wide, the bottom is 0.45 m wide and the height is 1.30 m.

![Figure 3. Cross-section view.](image)

![Figure 4. Front view.](image)

The material parameters of the concrete used in this analysis are: Young's modulus is \( E=3000 MPa \), Poisson's ratio is \( \nu=0.167 \), \( f_{tr}=3.19 MPa \), viscous zone is linear softening constitutive, fracture energy is \( G_p=50 N/m \), and linear expansion coefficient is \( \alpha=10^{-5} \). A steel bar is pre-buried at a quarter of the bottom of the longitudinal beam, which is HRB400 hot rolled ribbed, diameter \( d=20 \) mm, Elastic modulus \( E_s=200 GPa \), Poisson's ratio \( \nu_s=0.3 \), Reinforcement density \( \rho=7850 kg/m^3 \). Binding constraints are used between steel and concrete. Since in the actual wharf structure the initial crack is generated at 1.65 m away from the end of the longeron tensile zone on one side, the crack is preset here. The initial crack runs through the bottom of the longitudinal beam, with a width of 0.45 m and a shallow depth of 2 mm. In the model, the longitudinal beam is regarded as an elastic support continuous beam, the support elastic support is selected to perform elastic foundation and boundary setting, and the stiffness coefficient is set to \( K=400000 KN/m \). As shown in figure 5, the longitudinal beam model of the existing crack is established. The meshing size: encryption place is 25, the unencrypted is 50, divided into 30,576 units and 34,850 nodes. Figure 6 shows the results of the pre-embedded rebar, the meshing size of the rebar is 10. Figure 7 shows the results of the XFEM
extended pre-crack setting.

Figure 5. XFEM model of the longitudinal beam.  
Figure 6. Pre-embedded steel bars.  
Figure 7. Prefabricated crack.

In the use date, the self-weight of the beam has less influence. Based on the analysis of relevant literature [1,2], it is concluded that the main factor of crack expansion in the winter cooling period of the wharf is temperature plunge. Combining with the measured temperature data, as shown in figure 8 selected temperature drop section of from October to January of the following year, the initial temperature of 21°C, minimum temperature of 0°C. The cooling process is set as multiple load steps, and the temperature fields of each step are calculated respectively. Taking each temperature field as load, the temperature fields are applied on the longitudinal beam step by step, so as to study the cracking process of wharf structure crack in the whole cooling period.

Figure 8. Daily average temperature.

5.3. Calculation results
The calculation is submitted and the results are shown in figures 9-11. The maximum principal stress, strain and deformation contours of the reinforced concrete longitudinal beam. It can be seen that the main crack of the longitudinal beam expands along the direction of the initial crack while the maximum principal tensile stress is located at the tip of the crack, and obvious stress concentration phenomenon appears. From the surface of the beam, the expansion path of the crack is approximately horizontal to the beam height direction, the expansion direction of the crack is approximately perpendicular to the direction of the main tensile stress, and the initial crack is placed inside the element to better simulate the singularity of the stress field at the crack tip.

Figure 9. Maximum principal stress contour.  
Figure 10. Strain contour.  
Figure 11. Deformation diagram.
Comparing the two discounts in figure 12, we can find that the variation of the crack opening value caused by the XFEM simulation of the longitudinal beam cracking during the period from October to January of the following year is compared with the actual measured crack opening. The law of convergence is generally the same. This verifies the accuracy and convenience of the XFEM module in calculating crack problems. At the same time, we can see that the measured values and the calculated results are not completely consistent. This is because this paper only considers the effect of temperature on the cracking of the pier stringer, and there is only one steel bar is embedded in the model. In the actual construction and using procedure of the high-pile wharf, it is affected by various complex internal and external factors, and no further research is carried out in this paper. For example, loads such as stacking can also lead to the formation of cracks.

Figure 12. Curve opening and closing line chart.

6. Summary and outlook

6.1. Summary of this article
Since the official introduction of the XFEM terminology in 2005, the method has been widely used in discontinuities in many fields such as solids, complex fluids, and complex heat transfer. At present, the research and application of the XFEM in fracture mechanics mostly focus on the crack problem.

In this paper, the reinforced concrete longitudinal beam of a high-pile wharf in China is taken as the research object. Based on the extended finite element method, the ABAQUS software is used as the calculation platform to analyze the cracking characteristics of the longitudinal beam. Practice has proved that the XFEM can avoid the mesh re-division work and obtain higher efficiency. It utilizes the idea of unit decomposition and uses a discontinuous function to enhance the conventional displacement mode near the crack. In the process of simulating crack propagation, there is no need to preset the crack path and adjust the mesh. The evolution of the discontinuous interface is characterized by increasing the discontinuous displacement mode.

In the simulation process, I also found that the XFEM affects little on the calculation results in the mesh density. If the conventional finite element method is used, it is necessary to arrange a dense mesh around the crack tip to achieve good results. However, the size of the mesh should not be set too small. When the mesh size is close to 0.1mm, the crack will not expand and occur convergence error. When the mesh size is close to 0.5mm, the crack can expand, that is, the size of the mesh will influence the result. Therefore, for the expansion of plastic cracks, the XFEM also has some defects, which need to be improved by researchers.

6.2. Research outlook
Because in the process of simulating the longitudinal beam cracking, only temperature drop factor is considered, which obviously does not match the normal working environment of the wharf. Therefore, the calculated crack opening value and the measured data are different. Actually, there are still many problems to be further studied. I think that the following issues needed to be improved and perfected:
Considering the numerical value and deficiencies of the constitutive model itself, the process should use the appropriate modified steel and concrete constitutive model.

To use XFEM module for numerically analyzing different concrete members, considering the influence of various environmental factors and steel reinforcement ratio on the cracking of the longitudinal beam of high-pile wharf.

References

[1] Chen Z H, Fu X, Wang W P and Song R X 2005 The cause analysis and countermeasures for the longitudinal girder crack of piled wharf Water Transp. Eng. 2005 46-50

[2] Chen X D, Li J J and Huo Z Y 2017 Research on monitoring model of crack opening and fitting of piled wharf J. Water Conservancy Transp. Eng. 2017 53-9

[3] Lei Z P 2017 Research on cracking characteristics of concrete beams based on extended finite element method (Guangzhou, China: South China University of Technology)

[4] Belytschko T and Black T 1999 Elastic crack growth in finite elements with minimal remeshing Int. J. Numer. Methods Eng. 45 601-20

[5] Xiao Q Z and Karihaloo B L 2006 Improving the accuracy of XFEM crack tip field using higher order quadrature and statically admissible stress recovery Int. J. Numer. Methods Eng. 66 1378-410

[6] Chahine E, Laborde P and Renard Y 2006 A quasi-optimal convergence result for fracture mechanics with XFEM Acad. Sci., Paris, Ser. I 342 527-32

[7] Markus P 2005 Numerical aspects of the extended finite element method Proc. Appl. Math. Mech. 5 355-6

[8] Ventura G 2006 On the elimination of quadrature subcells for discontinuous functions in the extended finite-element method Int. J. Numer. methods Eng. 66 761-95

[9] Xiao Q Z and Karihaloo B L 2003 Direct evaluation of accurate coefficients of the linear elastic crack tip asymptotic field Fatigue Eng. Mater. Struct. 26 719-29

[10] Sethian J A 1999 Level set methods and fast marching methods: Evolving interfaces in computational geometry, fluid mechanics Comput. Vis. Mater. Sci. (Cambridge, UK: Cambridge University Press)

[11] Chopp D L and Sukumar N 2003 Fatigue crack propagation of multiple coplanar cracks with the coupled extended finite element and fast marching method Int. J. Eng. Sci. 41 845-69

[12] Dong Y W 2008 Numerical analysis of cracking of concrete structures based on extended finite element method (Nanjing, China: Hohai University)

[13] Liu B B 2009 Analysis of viscoelastic displacement field of extended finite element technology (Dalian, China: Dalian University of Technology)

[14] Yu T T 2010 Expansion finite element method for friction contact crack problem Eng. Mech. 27 84-9

[15] Dong Y W, Yu T T and Ren Q W 2008 XFEM for directly calculating stress intensity factor Chin. J. Comput. Mech. 25 72-7

[16] Chen L, Zhang W, Liu J and Yan L J 2017 Secondary development of ABAQUS software constitutively capable of automatically quantifying component damage grade Bldg. Struct. 47 60-7

[17] Zhang W, Cao G W, Li H N, Wang D B and Li M 2018 Secondary development of reinforced concrete material model based on ABAQUS fiber beam J. Shenyang Jianzhu University: Nat. Sci. Ed. 34 645-54