Computational and Simulation Analysis of Pull-Out Fiber Reinforced Concrete

Xia Zhao,1,2 Xiong-Jun He,1 Sheng Yan,1 and Nguyen Phan Anh1

1 School of Transportation, Wuhan University of Technology, Wuhan 430070, China
2 Binzhou Polytechnic, Binzhou 256603, China

Correspondence should be addressed to Xia Zhao; yxzhaoxia@sina.com

Received 26 November 2013; Revised 15 January 2014; Accepted 16 January 2014 Published 3 April 2014

Copyright © 2014 Xia Zhao et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

The computational and simulation analysis of pull-out fiber reinforced concrete was investigated. The finite element analysis was used to make this modeling and analysis on this reinforced system and three parts (concrete matrix, the placed fiber reinforcement polymers (FRP), and resin layer) were studied. A constant load was directly applied on the free end of placed FRP and the deformation, von Mises stress, displacement, and strain of these three analyzed parts were obtained. Meanwhile, the specimen system of bonding strength and strain was calculated by the method of ABAQUS. The results showed that, with the constant load, the von Mises stress, deformation, and strain appeared in these three parts, and the maximum values in both FRP and resin layer were shown at the free end side, which provides an accurate description of the rupture mode.

1. Introduction

Concrete is the most widely used construction material in large quantities for its low cost and wide availability [1, 2]. However, it suffers from low tensile strength and limited strain capacity, which gives rise to formation of microcracks in a loading state. Microcracks have an enormous influence on the durability and the formed cracks accelerate the deterioration by increasing the permeability of the matrix through freezing-and-thawing damage, alkali silica reaction, chloride penetration, and other mechanisms [3, 4]. Nowadays, the researchers find that the fiber reinforcement polymers (FRP) have created an extensive field to control the matrix cracks. The structures achieve a good strengthening effect with FRP in mechanical and durability properties. Their high stiffness, chemical resistance, tensile strength, and fire resistance make them attractive for the next generation high performance reinforced composites materials of the 21st century [5–10].

Prior works on FRP reinforced concrete have focused on the finite element analysis modeling and simulation in the literature [11–30]. Wu et al. [31] studied the cracking behavior and interfacial debonding fracture in FRP-strengthened concrete beams and a finite element analysis was performed to obtain the different types of debonding propagation along FRP-concrete interface and crack distribution in matrix. Benzarti et al. [32] presented a coupled damage model to predict the durability of concrete elements strengthened by external bonding of FRP plates and their numerical results and experimental tests showed that the model captures well the debonding fracture initiation. An experimental investigation of the fatigue behavior of FRP-concrete was investigated by Carloni et al. [33] and they found that the length of stress transfer zone during fatigue loading was smaller than the stress transfer zone associated with the cohesive crack under quasistatic loading; also the postfatigue results suggested the possibility of a different debonding mechanism during fatigue loading.

In this paper, the finite element analysis [34, 35] was used to make this modeling and analysis on this pull-out FRP/concrete system. Each specimen of bonding strength and strain was calculated by ABAQUS method. The concrete matrix, the strengthening FRP, and resin layer were modeled individually as damageable materials with a constant loading. The von Mises stress, strain, deformation, and displacement were all given to analyze this rupture mode of selected composites.
Table 1: Young’s modulus and Poisson’s ratio of three analyzed parts.

<table>
<thead>
<tr>
<th>Materials</th>
<th>Young’s modulus/GPa</th>
<th>Poisson’s ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete matrix</td>
<td>$3.152 \times 10^4$</td>
<td>0.3</td>
</tr>
<tr>
<td>FRP</td>
<td>$1.618 \times 10^5$</td>
<td>0.2</td>
</tr>
<tr>
<td>Resin layer</td>
<td>$5.21 \times 10^3$</td>
<td>0.39</td>
</tr>
</tbody>
</table>

Table 2: Summary of testing results.

<table>
<thead>
<tr>
<th>Specimen</th>
<th>$L$ (mm)</th>
<th>$b_c$ (mm)</th>
<th>$t_c$ (mm)</th>
<th>$f_c$ (MPa)</th>
<th>$E_c$ (MPa)</th>
<th>$P_{exp}$ (kN)</th>
<th>Failure</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 MPa-250-10</td>
<td>250</td>
<td>10.29</td>
<td>1.22</td>
<td>30</td>
<td>31,520</td>
<td>26.6</td>
<td>D</td>
</tr>
</tbody>
</table>

Table 3: The detailed dimensions of each three parts.

<table>
<thead>
<tr>
<th>Sample</th>
<th>$L_c$ (mm)</th>
<th>$b_c$ (mm)</th>
<th>$t_c$ (mm)</th>
<th>$L_f$ (mm)</th>
<th>$b_f$ (mm)</th>
<th>$t_f$ (mm)</th>
<th>$L_r$ (mm)</th>
<th>$b_r$ (mm)</th>
<th>$t_r$ (mm)</th>
<th>$P$ (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 MPa-250-10</td>
<td>350</td>
<td>300</td>
<td>180</td>
<td>350</td>
<td>1.22</td>
<td>10.29</td>
<td>250</td>
<td>1</td>
<td>10.29</td>
<td>26600</td>
</tr>
</tbody>
</table>

For each component of this modeled system, the stress-strain curve of concrete follows the mathematical model investigated by Todeschini et al. [36]. The size of the concrete block was $350 \times 300 \times 180$ mm, which was shown in Figure 2. Concrete cylinders were used to define material properties including the compress strength and Young’s modulus in Table 2. FRP is assumed as behaving a linear-elastic condition which showed the failure stress and strain in longitudinal tension. At the failure point, FRP loses its tensile strength. The type of constitutive behavior of resin is simulated by an elastic-plastic model with strain hardening for quasistatic response during pull-out testing. In order to define this behavior, the properties of resin were given below, such as Young’s modulus of 5210 MPa, the tensile strength of 16 MPa, and Poisson’s ratio of 0.39. All components were followed the American Standard ASTM D638. A finite element analysis is used to make this modeling and analysis on this reinforced system. The details of this modeling procedure are as follows.

1. Define a 3D concrete block with a sized groove placed in the middle of the matrix. The size of groove is $(1 + 1.22 + 1) \times 10.29$ mm.

2. Define a 3D deformable FRP plate $(1.22 \times 10.29 \times L_{FRP}$ mm) which lays in the groove. A constant load is applied at the free size of FRP: $L_{FRP} > L = 250$, where $L_{FRP}$ is length of FRP strip and $L$ stands for bond length of FRP as experimental test.

3. Define two layers of the deformable coated resin $(1 \times 10.29 \times 250$ mm). Resin layer is coated between the matrix and FRP. The detailed dimensions of concrete matrix, FRP plate and resin layer are shown in Table 3, where $b_f$, $t_f$, $b_p$, $t_p$, and $P$ are the width and length of concrete matrix, the placed grooves, and FRP, respectively. $\tau_f$ represents the maximum value of interfacial shear stress and $L$ shows the placed length of FRP in matrix. A constant load (26600 N) was applied on the free end of placed FRP.

4. Define interfacial bond between FRP plate and concrete and FRP plate and resin layer by tying constraint.
of two adjacent surfaces. In this modeling, the tie constraint surfaces include concrete-first resin layer, first resin layer-FRP, FRP-second resin layer, and second resin layer-concrete. The defined model is shown in Figure 2.

2.3. Mesh, Loading, and Boundary Condition. As one important step in this modeling, a detailed meshing [37] can ensure a relatively accurate modeling result. Structural mesh generation technique was applied in this modeling. Mesh of concrete matrix, FRP plate, and resin layer are all shown in Figure 3. A constant load (26600 N) is applied at the free end of FRP plate and the corresponding boundary condition was followed as the experimental test, which was done by Seracino et al. [38]. The loading and boundary condition were shown in Figure 4.

2.4. Analysis Algorithm and Control Solutions. In this simulation, a directly nonlinear analysis technique was employed and this technique followed the method of Newton-Rapson. In the modeling procedures, the system stayed in a static loading condition. Automatic time step was applied with set 1. The maximum number was 100 and the increment size included the initial value 1, the minimum value 1E-05, and the maximum value 1.

3. Analysis

3.1. Calculation. In order to satisfy the accuracy of the model, the bond interface characteristics of the analytical model were calculated by using the method of finite element analysis. In the Seracino et al. [39] model, the predicted IC debonding failures of FPR strengthening system with a constant loading can be calculated by

$$P_{IC} = \alpha_p \varphi_f^{0.25} \sigma_y^{0.3} \sqrt{L_p E_p A_p} < f_{rup} A_p.$$  (1)

As calculated in (1) and previous design, results of each specimen are shown in Table 4. This thesis uses ABAQUS software to simulate finite element analysis method. All specimen systems were broken down finally.

3.2. Deformation. With the constant loading, a deformation was obtained in FRP plate. No deformation appeared in the area of XY plane and YZ plane; but a remarkable deformation occurred in XZ plane and it was bended inwards. According to the symmetric principle, the deformation was cancelled in XY plane and YZ plane. However, a bending moment was obtained for the constant load in XZ plane. The deformation of this modeling was shown in Figure 5 and an evident deformation occurred at the edge between matrix and FRP plate.
Figure 3: Mesh of concrete matrix, FRP plate, resin layer, and the reinforced concrete system.

Figure 4: The loading and boundary condition of reinforced concrete system.

Table 4: Calculation for each specimen test by Abaqus software.

<table>
<thead>
<tr>
<th>Specimen</th>
<th>Bond strength (kN)</th>
<th>$\varepsilon_{\text{max}}$ (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>30 MPa-100-10</td>
<td>20.4</td>
<td>0.009133</td>
</tr>
<tr>
<td>30 MPa-150-10</td>
<td>23.2</td>
<td>0.010213</td>
</tr>
<tr>
<td>30 MPa-200-10</td>
<td>27.9</td>
<td>0.012238</td>
</tr>
<tr>
<td>30 MPa-250-10</td>
<td>26.6</td>
<td>0.0118</td>
</tr>
<tr>
<td>30 MPa-300-10</td>
<td>26.0</td>
<td>0.011452</td>
</tr>
<tr>
<td>30 MPa-350-10</td>
<td>23.0</td>
<td>0.010106</td>
</tr>
</tbody>
</table>
3.3. von Mises Stress. von Mises stress [40, 41] was always used to describe the distribution of stress, and the color in each mesh showed the stress value. The stress means to be increased when the color turns from blue to red, and the von Mises stress values can be obtained. The von Mises stress of concrete matrix, FRP plate, and resin layer were all shown in Figure 6. The maximum von Mises stress (2.661 × 10^3 MPa) value was obtained at the edge between matrix and FRP plate. Meanwhile, the maximum von Mises stress of FRP plate 2.661 × 10^3 MPa and resin layer 2.661 × 10^2 MPa appeared at the board edge of FRP plate.

3.4. Displacement. The displacement showed the degree of deformation of concrete matrix, FRP plate, and resin layer, which indirectly reflected the bonding strength. The maximum displacement of tested three parts (Figure 7) appeared at the edge of contact point, and the maximum values were 0.2056 mm, 2.011 mm, and 0.2752 mm, respectively. The displacement decreased with the deeper groove of concrete, which showed that the bonding strength was much higher and a greater durability property was obtained.

3.5. Strain. The simulated strain showed the deformation resistance ability, which indirectly reflected the displacement and bonding strength. The same distribution trends were obtained in Figure 8, and the maximum value all occurred at the contact place. All the maximum strain was 2.188 × 10^{-3}, 1.644 × 10^{-2}, and 4.637 × 10^{-2}, respectively.
4. Conclusions

In this paper, the computational and modeling analysis of the pull-out FRP/concrete system was studied systematically. A finite element analysis was used in this modeling procedure. Each specimen system of bonding strength and strain was calculated by the method of ABAQUS. All the three parts, concrete matrix, FRP, and resin layer, were studied in this analysis individually. A constant load (26600 N) was applied on the free side of the placed FRP. Deformation, von Mises stress, displacement, and strain of each individual part were obtained and the maximum values all occurred at the edge of the contact point. Meanwhile, the values decreased with the deeper groove of concrete, which showed that the higher bonding strength was gained in the deeper groove, and the contact point was the weakest zone in this pull-out FRP/concrete system.

Further researches are needed to obtain a deeper analysis of pull-out FRP reinforced concrete. Also, the detailed pulling-out process (elastic stage, elastic softening stage, debonding stage, and softening-debonding stage) and slip and shear stress at the interfacial bond shall be explored. We believe that our results at least in the trend are helpful for the research of FRP reinforced concrete system.

Conflict of Interests

The authors declare that there is no conflict of interests regarding the publication of this paper.

Acknowledgment

The authors would like to express appreciation for the financial support by the Natural Science Foundation of China (51178361).

References


Submit your manuscripts at http://www.hindawi.com