Nonlinear Finite Element Modeling of the Interior Steel-Concrete Composite Beam Joints

Pengfei Mao\(^1\) *, Donglin Wang\(^2\)

\(^1\) Department of Civil Engineering, Jiangsu University, Zhenjiang, Jiangsu, 212013, China
\(^2\) Urban Main Road Construction Office, Zhenjiang, 212001, China

(Received 30 December 2010, accepted 31 January 2011)

Abstract: Nonlinear 3D finite element models (FEM) of the two kinds of joints, namely steel bars headed through the pipe joint and steel bars welded with the upper strengthened ring joint, are established by using the ANSYS program. By choosing the suitable element type, boundary condition and loading regime, the author made intensive study on the stress distribution of the steel tubular, the concrete, the strengthened ring and the bar. The result shows that the finite element model accords with the actual stress situation of the joint. Holes on the steel tubular have little effect on the bearing capacity of the axial load of the joint, but the greater impact on the load transfer. The specimen \( \sigma \) with the strengthened ring is a better structural form.

Keywords: nonlinear finite element analysis; joint; steel-concrete composite beam; concrete-filled steel tubular column; joint; stress

1 Introduction

The steel-concrete composite beam is a new kind of composite beam which works well under the negative bending moment. And this also makes it possible to design a rigid connection between the steel-concrete composite beam and the concrete-filled steel tubular column [1,2]. The concrete-filled steel tubular structure as a new form of structure has been widely used Engineering Practice. This kind of structure has some notable features, like high strength, good seismic performance, and easy construction. But how to deal with the steel-concrete composite beam to concrete-filled steel tubular column joints has always been one of the key technical issues that affects the application of this kind of structure [3,4]. The joints between the steel-concrete composite beam and concrete-filled steel tubular column are the key parts. The design principle of "strong column and weak beam, joints even stronger" in the seismic design shows us the importance of joints in the structure. So, the author proposes application of a composite system composed of the steel-concrete composite beam to concrete-filled steel tubular column joints. Compared with the normal concrete structure system, this composite system has not only better function of use and faster construction. It also has advantages of lower cost than the steel frame system [5-7]. But the steel-concrete composite beam to concrete-filled steel tubular column joint is a new form of composite joint of which the mechanical behavior is complicated. The structural form needs to be reasonably designed and verified by experiment. By using the nonlinear method, we can save the expenditure and make more changes to the parameters to get more results that we can’t get in the experiment. So it’s necessary to do some further research on the nonlinear mechanical behavior of joints, in order to establish the nonlinear model which could accurately reflect the deformation feature of joint under force. It would also help us to have a better understanding of the loading mechanism of the steel tubular, the concrete, the longitudinal reinforced bars and the strengthened ring of the core area in theory.

2 Element model

The SOLID65 element which is usually used in the 3-D solid models with (or without) bar is chosen to simulate concrete [8]. This finite element model can simulate tension cracking, crush, plastic deformation and creep of concrete in three

\* Corresponding author.  E-mail address: housemaker0422@163.com
orthogonal directions. The author chose SHELL181 element to simulate the wrapped steel, strengthened ring and steel tubular. This finite element model can simulate stress stiffening and large deformation of these materials. The LINK8 element which is the tension-compression element along axis is chosen to simulate the bar. This finite element model has 3 Degrees of freedom of each joint. Meanwhile, two rigid pads are added to both the top and bottom of the column to avoid the external loads applied on the solid element directly. This also reduces the stress concentration and makes the solving process much easier. The rigid pad is simulated by the SOLID45 element which is capable of plasticity, creep, expansion, stress hardening, large deformation and large strains. There are two methods to deal with the bond-slip relationship between the bar, steel tubular and concrete: one ordinary method is to consider them complete bond without slip; the other method is to add the bonding element to simulate the bond-slip behavior between steel and concrete, such as the COMBIN39 element, TARGE170 element and CONTAC173 element. Some research [9-11] shows that it’s not significant for the reinforced concrete by adding the bonding element. So in this paper, the author assumes complete bond between the bar and concrete while using the COMBIN39 element to simulate the contact and slip between the wrapped steel, the steel tubular and the concrete.

3 Cell mesh

When dividing the element, the joints of the bars and the concrete, the wrapped steel and the concrete, the steel tubular of the core area and the concrete should be the same. That is to say the division of the bar element and the concrete element, wrapped steel element and concrete element, steel tubular element of the core area and concrete element should keep the same. The joint coincides with the connecting concrete element and has the same coordinate values but different element number. The finite element model is shown as Figs. 1 and 2.

![Figure 1: The finite element model of G − 1](image1)

(a) The whole model  
(b) The reinforced bar through the tubular

![Figure 2: Finite element model of G − 2](image2)

(a) The whole model  
(b) Reinforced bar and strengthened ring

The material parameters of the nonlinear model are the values determined by mechanical experiment, material description and empirical formula of the materials used in the joint experiment. The values are shown in Table 1.

<table>
<thead>
<tr>
<th>Material Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Compressive strength of the concrete cube</td>
<td>28MPa</td>
</tr>
<tr>
<td>Axial compressive strength</td>
<td>21.3MPa</td>
</tr>
<tr>
<td>Thickness of the steel plate</td>
<td>t mm</td>
</tr>
</tbody>
</table>

**Annotation:** The value of the compressive strength of the concrete cube is 28MPa. The value of the axial compressive strength is 21.3MPa. The t in the Tab means the thickness of the steel plate and the dimension of t is mm.

**IJNS email for contribution:** editor@nonlinearscience.org.uk
Table 1: Mechanical property of materials

<table>
<thead>
<tr>
<th>Material</th>
<th>Type</th>
<th>Yield strength(MPa)</th>
<th>Tensile strength(MPa)</th>
<th>Modulus of elasticity(MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete</td>
<td>C30</td>
<td>30</td>
<td>43</td>
<td>$3 \times 10^4$</td>
</tr>
<tr>
<td>Steel tubular</td>
<td>Q235, $t=6$</td>
<td>411</td>
<td>496</td>
<td>$2.1 \times 10^5$</td>
</tr>
<tr>
<td>Side plate</td>
<td>Q235, $t=2$</td>
<td>328</td>
<td>400</td>
<td>$2.1 \times 10^5$</td>
</tr>
<tr>
<td>Bottom plate</td>
<td>Q345, $t=8$</td>
<td>443</td>
<td>543</td>
<td>$2.1 \times 10^5$</td>
</tr>
<tr>
<td>Reinforced bar</td>
<td>$\phi 8$</td>
<td>294</td>
<td>416</td>
<td>$2.1 \times 10^5$</td>
</tr>
<tr>
<td></td>
<td>$\phi 25$</td>
<td>457</td>
<td>553</td>
<td>$2.1 \times 10^5$</td>
</tr>
</tbody>
</table>

Table 2: Axial ultimate bearing capacity of joints

<table>
<thead>
<tr>
<th>Specimen number</th>
<th>Bearing capacity (kN)</th>
<th>Compare with $G-2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$G-1$</td>
<td>3008.61</td>
<td>1.13%</td>
</tr>
<tr>
<td>$G-2$</td>
<td>3043</td>
<td></td>
</tr>
</tbody>
</table>

4 Analysis of the FEA results

4.1 Analysis of the loading capacity of specimens

The boundary conditions of the finite element model should keep the same with the device of the experiment. Constrained conditions: even articulations are added to both the top and bottom of the steel tubular in the experiment to simulate hinge support. In this way, the top and bottom of steel tubular could only spin but no translation in the plane formed by the beam and column. In order to simulate this constraint in finite element analysis, the bottom of column is constrained in $X, Y, Z$ axis to limit the horizontal and vertical displacement but release the rotational displacement. Meanwhile, the top of column is constrained in $X, Z$ axis to limit the horizontal displacement but release the rotational and vertical displacement. The stress-strain relationship of the joints is shown in Fig.3 and the values of the axial ultimate bearing capacity of the joints are shown in Table 2.

Figure 3: Load-displacement curves of joints under axial load

We can see from Fig.3 that the bearing capacity of both joints decrease significantly when they reached the maximum compression. And the curve of $G-1$ is relatively gentle before the max value while the curve decreases more greatly than $G-2$ after that. We can draw a conclusion that the ductility of $G-2$ is better than $G-1$. We can see from the table that the values of the loading capacity of the both joints are nearly the same. The values of the axial loading capacity of the joints are 1.13% apart. It suggests that the axial loading capacity of the column with hole in the core area doesn’t decrease much. This is mainly because that although the big hole exists in the core area, the concrete inside and outside of the hole works together. The concrete outside and the bonded steel provided a kind of lateral restraint to the concrete inside. And meanwhile part of the longitudinal bars go through the hole also provide a kind of vertical bracing.
4.2 Stress analysis of the joints under axial load

The \( Y \)-axis axial normal stress nephograms of the wall of the steel pipe and the concrete under the axial ultimate bearing capacity of the column are shown in Figs. 4 and 5. The stress nephograms show that the phenomenon of stress concentration of the specimen \( G - 1 \) is more serious than \( G - 2 \). This is because the longitudinal bars go through the hole provide a kind of vertical bracing to \( G - 1 \), while the strengthened ring of \( G - 2 \) can avoid the stress concentration caused by the hole and the stress on the column is relatively uniform. However this has no great effect on the axial ultimate bearing capacity of the column.

The SOLID65 element which is usually used in the 3-D solid models with (or without) bar is chosen to simulate concrete. This finite element model can simulate tension cracking, crush, plastic deformation and creep of concrete in three orthogonal directions. The author chose SHELL181 element to simulate the wrapped steel, strengthened ring and steel tubular. This finite element model can simulate stress stiffening and large deformation of these materials. The LINK8 element which is the tension-compression element along axis is chosen to simulate the bar. This finite element model has 3 Degrees of freedom of each joint. Meanwhile, two rigid pads are added to both the top and bottom of the column to avoid the external loads applied on the solid element directly. This also reduces the stress concentration and makes the solving process much easier. The rigid pad is simulated by the SOLID45 element which is capable of plasticity, creep, expansion, stress hardening, large deformation, large strains.

There are two methods to deal with the bond-slip relationship between the bar, steel tubular and concrete: one ordinary method is to consider them complete bond without slip; the other method is to add the bonding element to simulate the bond-slip behavior between steel and concrete, such as the COMBIN39 element, TARGE170 element and CONTAC173 element. Some research [12] shows that it’s not significant for the reinforced concrete by adding the bonding element. So in this paper, the author assumes complete bond between the bar and concrete while using the COMBIN39 element to simulate the contact and slip between wrapped steel, steel tubular and concrete.

5 Cell mesh

When dividing the element, the joints of the bar and concrete, wrapped steel and concrete, steel tubular of the core area and concrete should be the same. That is to say the division of the bar element and concrete element, wrapped steel element and concrete element, steel tubular element of the core area and concrete element should keep the same. The joint coincides with the connecting concrete element and has the same coordinate values but different element number. The finite element model is shown as Figs 5 and 6.

![Stress nephogram of the steel tube](image1)

(a) Stress nephogram of the steel tube

![Stress nephogram of the concrete](image2)

(b) Stress nephogram of the concrete

Figure 4: \( \sigma_y \) stress nephogram of the steel tube and concrete of \( G - 1 \) Unit: MPa

5.1 Stress distribution of the longitudinal bars of the beam

The stress nephograms of the longitudinal bar of the beam under the monotonic load are shown in Fig.6. We can see from the figure that the farther away from the loading end, the greater the stress of the bar is. The max value of the stress of the specimen \( G - 1 \) is up to 295.34MPa in the core area, while the max value of the stress of the specimen \( G - 2 \) is up to 397.37MPa in the area where the bars split. Neither the reinforced bars of the joints reached the yield strength. That is probably because that the yield strength of the bonded steel is smaller than the yield strength of the reinforced bars. When
the beam fails, the bonded steel reaches the yield strength while the reinforced bars didn’t. We apply the displacement load in the FEM on the free end of the beam. The displacement applied transformed into the bending moment and shearing force which gradually pass along the beam to the joint. The stress distribution of the reinforced bars of the both joints is almost the same along the beam (except the core area). This shows that there is no great effect on the stress distribution of the reinforced bars because of the different structure of the core area. We can not determine if the plastic damage has already occurred when the solid side bars reach the yield strength.

5.2 Stress distribution of concrete

The $\sigma_y$ stress nephograms of the concrete under the monotonic load are shown in Fig.7. We can see from the figure that the farther away from the loading end, the destroyed areas of both joints are near the beam end of the joint. The value of the stress of joint $G - 1$ is larger and up to 83.02MPa. The value of the stress of joint $G - 2$ is smaller and up to 57.82MPa. This means part of the shearing force and the moment of the joint $G - 2$ passed to the upper strengthened ring in the core area meanwhile the load of the joint $G - 1$ without the upper strengthened ring is born by the concrete of the beam. The stress on the beam end of the joint $G - 1$ is higher than $G - 2$.

IJNS homepage: http://www.nonlinearscience.org.uk/
5.3 Stress distribution of the steel pipe

Figure 8: \( \sigma_y \) stress nephogram of \( G-1 \) steel tube under the monotonic loadUnit: MPa

Figure 9: \( \sigma_y \) stress nephogram of \( G-2 \) steel tube under the monotonic loadUnit: MPa

The \( \sigma_y \) stress nephograms of the joints under the monotonic load are shown in Figs 8 and 9. We can see from the figures that the stress of \( G-1 \) concentrates around the hole and is up to 555.87MPa. The stress of \( G-2 \) mainly concentrates on the area where the strengthened ring and the steel pipe join together and is up to 511.51MPa. They both reached the yield strength. This means the load transfer mechanism of the joints is different. A large part of the shear of the beam end of \( G-1 \) transferred along the weld between the strengthened ring, the side plate and the steel pipe and the rest transferred directly along the concrete through the hole into the concrete of the column. However the shear of the beam end of \( G-1 \) transferred mainly along the weld between the strengthened rings, the side plate and the steel pipe.

6 Conclusions

In this paper, the author uses the FEA method to study the axial load and the beam end under monotonous load of two kinds of joints by considering part of the bond-slip. We learn the loading mechanism of the joints and draw the conclusions from the FEA results: (1) To drill the hole in the core area has little effect on the axial loading capacity of the column. (2) The force of both concrete of the core area is the largest. And along the wing plate, the stress reduces. (3) The load transfer mechanism of \( G-2 \) helps the materials to work together effectively and make full use of the strength which makes it a better structural form than \( G-1 \).

Acknowledgements

The work is supported by Six Human Resource Peak Hund(2010-jz-10).
Figure 10: $\sigma_x$ stress nephogram of the bonded steel and the strengthened ring. Unit: MPa

References


IJNS homepage: http://www.nonlinearscience.org.uk/