Optimization Design of Double Jacked-in Spun Pile Tip

Zhiren Yuan¹, Yanqi Yang¹ *, Lijun Dou¹ and Yane Sui¹

Changchun Institute of Technology, Jilin Province Key Laboratory for Earthquake Resistance&Hazard Mitigation of Civil Engineering, Jilin Province Research Center of Construction Technology&Engineering for Housing in Cold Regions, 130012, Changchun, China

*Corresponding author: 2834559005@qq.com

Abstract. The design method of the pile end plate of double jacked-in spun pile tip is described. The finite element analysis is used to simulate the stress state of pile sinking and loading test. Two models to stimulate the actual stress of the pile end plate. According to the analysis results, the existing design method of pile end plate is optimized.

Keywords: Double Jacked-In Pile, Finite Element Analysis, Spun Pile Tip

1. Introduction of Double Jacked-in Spun Pile

In order to improve the mechanical performance of the ordinary prestressed concrete pile pile during the pile pressing process and improve the construction efficiency, the double Jacked-in Spun Pile is proposed, which applies the pressure at the pile top and the core of the pile at the same time, and the schematic diagram of double Jacked-in Spun Pile pressing is shown in Figure 1. It can be seen from Figure 1 that the end plate of the pile is subject to the pressure of the internal pressure bar, the resultant force exerted by the pile body and the steel strand and the reaction force of the soil. Because the stress of the end plate is complex, which has a great influence on the safety and construction efficiency of the pile, it is necessary to optimize it.

2. Finite Element Analysis

2.1. Create Parts

Prestressed high-strength concrete (PHC) pile with an outer diameter of 400mm and thickness of 95mm is used as the pile body of the double Jacked-in Spun Pile. The pile length is 12m, and the concrete strength grade is C80. The pile end plate is made of steel plate with diameter of 400mm and thickness of 20mm, and the material is Q235 steel. The diameter of steel strand is 9mm (1 × 7). The internal pressure bar is a steel column with a diameter of 180mm and a yield strength of 400MPa. The part module of ABAQUS software is used to create the parts of the 8-meter double Jacked-in Spun Pile in prototype scale. According to the design size create concrete pile body, steel strand, anchor plate, pile end plate, internal pressure bar and other components.
2.2. Define Material Properties

The material property is the constitutive relation of the material. Choosing the appropriate constitutive relation can improve the accuracy of the finite element simulation results. The constitutive relation of concrete is relatively complex. The concrete constitutive relation models provided by ABAQUS are as follows: [1] smeared crack model, brittle cracking model and plastic damage model.

Considering the damage effect, the plastic damage model is suitable for the static loading simulation with small surrounding pressure and good convergence.

The material properties of each component are defined by the property module, the material parameters of each component are shown in Table 1.

| parts             | Modulus of elasticity (MPa) | Poisson's ratio v | Yield stress (MPa) | Expansion coefficient |
|-------------------|----------------------------|-------------------|---------------------|-----------------------|
| Concrete C80      | 3.8×10^4                   | 0.2               | --                  | --                    |
| Steel strand      | 1.9×10^5                   | 0.3               | --                  | 1.2×10^5              |
| Internal bar      | 2.1×10^5                   | 0.25              | 420                 | --                    |
| Pile end plate    | 2.1×10^5                   | 0.25              | 235                 | --                    |

2.3. Define Constraints

The assembly module is used to assemble the parts of the model. Each component is assembled and positioned to form an assembly. The steel strand is embedded into the concrete component by the interaction module, and tie constraints are set between the anchor plate surface and the concrete surface.

MPC binding constraints shall be set for steel strand and anchor plate. The definition of MPC (multipoint constraints) multipoint constraints in ABAQUS software is a coupling relationship of node degrees of freedom. By defining one or several degrees of freedom of a point, and then specifying the degrees of freedom of other nodes and defining the degrees of freedom of the point to establish a motion coupling relationship, the displacement of the two points is no longer independent of each other, so as to connect different parts [2]. There is no relative displacement between prestressed steel strand and anchor plate, so rigid binding MPC constraint is used to connect the two parts [3].
2.4. Define Contact
The contact relationship of the whole model includes the contact between the internal pressure bar and the pile end plate, and the contact between the concrete pipe pile and the pile end plate. Contact tangential behavior follows Coulomb friction formula [4].

In the model, the internal pressure bar and the pile end plate are in contact with steel tube and steel plate, the friction coefficient is 0.15, the pile body and the pile end plate are in contact with concrete and steel plate, and the friction coefficient is 0.6°.

In the ideal state, the contact surface has no shear deformation before the sliding state, but it is difficult to converge in the finite element numerical calculation. Therefore, ABAQUS software introduces the concept of "elastic sliding deformation". Elastic slip deformation means that a small amount of relative slip deformation is allowed when the surfaces are bonded together, as shown in Figure 2. According to the normal element length on the contact surface, the elastic sliding deformation is determined. In this model, the software default elastic sliding deformation is used, that is, the deformation is 0.5% of the element length.

![Figure 2 Elastic sliding deformation](image)

The contact normal behavior adopts "hard contact", that is, the contact pressure that can be transmitted between the contact surfaces is not limited. When the contact pressure becomes zero or negative, the two contact surfaces are separated and the contact constraints of the corresponding nodes are removed. This normal behavior may have penetration phenomenon in the calculation limit. The relationship between pressure and clearance is shown in Fig.3.

![Figure 3 Model contact normal behavior](image)

There are two common ways to apply "hard" contact: 1) Direct method; 2) "Penalty" method. Using direct method to apply "hard" contact, it is considered that the contact pressure is zero before the contact pair contacts, and the contact pressure reaches the maximum at the moment of contact. The contact of direct method is an ideal contact relation, so it is easy to cause convergence difficulty in the simulation of finite element software. When the "penalty" method is applied to contact, it is considered that the contact pressure does not increase to the maximum value instantaneously, but gradually increases to the maximum along a oblique line, and the slope of the line is the penalty stiffness. Generally, the default penalty stiffness can be approximate to the direct method in the accuracy of the results, and the calculation cost is small. In this paper, the method of "penalty" is applied to contact, and the penalty stiffness is the default penalty stiffness of the program [5].
2.5. Prestress Application

Through the analysis step module, multiple analysis steps are created to calculate the prestress and external load. The type of analysis step is set as static universal. Then load module is used to set the load condition of the model. The prestressing force of the steel strand is exerted by the cooling method. The specific implementation method is to restrain the displacement of both ends of the steel strand by the anchor plate, and then apply the temperature load (cooling) on the prestressed steel strand to shrink the steel strand, so as to achieve the effect of prestressing force.

According to the specification, 933MPa\(^8\) of prestress shall be applied to each steel strand, and the temperature shall be reduced by 410 °C after calculation. After the prestress condition is applied, the prestress Mises stress reaches 912MPa.

2.6. Load Exertion

Apply the pile top load of 3000kN and the center load of 1500KN to the pile body and internal pressure bar respectively, and the loading position is shown in Figure 4. At the same time, in order to simplify the calculation model, the soil around the tip of pile and the pile body is not modeled, and the influence of the side friction resistance of the soil around the pile on the stress of the pile body is ignored. The effect of the soil on the pipe pile is replaced by the uniform reaction force on the pile end plate, and the load size is shown in table 2.

Table 2. Applied surface load of components

| Parts              | Area (mm\(^2\)) | Total load (N) | Surface load (MPa) |
|--------------------|-----------------|----------------|--------------------|
| Internal pressure bar | 25434           | 1.5×10\(^6\)   | 58.976             |
| Pipe pile          | 90981.5         | 3.0×10\(^6\)   | 32.974             |
| Pile end plate     | 1256000         | 4.50×10\(^6\)  | 35.828             |

![Internal pressure bar load](image1)

![Pile load](image2)

Figure 4 Load on parts of model

2.7. Boundary Condition

In the process of pile sinking, there is no relative displacement between the pile body and the outer casing, so the fixed end constraint is applied on the top of the pile; the internal pressure bar is in close contact with the piston, and the piston can have vertical displacement in the oil cylinder, so the end of the internal pressure bar needs to release the vertical displacement, only constraining the horizontal displacement.

2.8. Mesh Generation

According to the interpolation order of node displacement, ABAQUS can be divided into linear element, quadratic element and modified quadratic element. The linear element is also called the first-order element. Its shape function is determined only by the angular displacement of the element, which is determined by the linear interpolation method, so the calculation cost of the first-order element is small5.

In the finite element solver, the element stiffness matrix and the equivalent node load are determined by numerical integration. According to the different integration methods, the element can
be divided into linear complete integration element, quadratic complete integration element, linear reduced integration element, quadratic reduced integration element and incompatible mode element. When the shape of element is regular, Gauss integral is used to integrate the stiffness matrix of element accurately. Compared with the complete integral, the reduced integral uses one less integral point in each direction. The linear reduced integral has a high accuracy in the solution of displacement, and is not sensitive to the distortion of the grid, and will not occur shear self-locking under the bending load [6]. Therefore, the concrete, end plate and anchor plate need 8-node 6-face linear reduced integral element-c3d8r, and the steel strand adopts three-dimensional truss two node complete integral element-t3d2.

2.9. Establishment of Contrast Model
At present, the method of "equivalent load substitution" is adopted in the design of pile end plate, that is to say, the effect of internal pressure bar on pile end plate is considered to be equivalent load substitution. In practical engineering, the stress redistribution is caused by the deformation of the pile end plate, which makes the load on the pile end plate different from the theoretical design [7]. In order to analyze the force difference between the theoretical design and the actual situation, a comparative model is established. Under the same conditions of interaction, boundary condition and load size, the comparison model only changes the loading mode and cancels the internal pressure bar. The effect of the internal pressure bar on the pile end plate is replaced by the equivalent uniform load, as shown in Figure 5. Based on this simulation, the stress of the pile end plate using the equivalent load substitution method is analyzed, and the following results are represented by the equivalent load [8].

![Figure 5 Load acting directly on pile end plate](image)

3. Comparison of Analysis Results
The comparison results show that the stress magnitude and distribution of the two loading methods are totally different. The maximum stress of pile end plate loaded by internal pressure bar is 145MPa, which is quite different from 388MPa of equivalent load. At the same time, the distribution of the maximum stress of the pile end plate is not the same in the two cases. It can be seen from the figure that the maximum stress of the pile end plate loaded by the internal pressure bar is mainly distributed in the position contacting with the edge of the internal pressure bar, while the maximum stress of the equivalent loading case is distributed in the center of the pile end plate [9]. This is due to the deformation of the pile end plate during the working process of the double static pressure pipe pile, which results in the partial separation from the internal pressure bar. The comparison results are shown in fig.6 and fig.7.

As shown in Fig. 8, the stress nephogram of internal pressure bar is shown. It can be seen from the figure that the stress at the edge of the internal pressure bar is 119MPa, which is greater than 49MPa in the central area. Therefore, it is determined that after the center area of the internal compression bar is separated from the pile end plate, it only contacts the pile end plate around. After the central area is separated, the action of internal pressure bar on the pile end plate is only transferred through the surrounding contact part, the stress changes, and the stress redistributes [10].
Figure 6 Stress nephogram of pile end plate on the side contacting with pile

Figure 7 Stress nephogram of pile end plate on the side contacting with soil

Figure 8 Stress nephogram of internal pressure bar on the side contacting with pile

From the results of finite element analysis, it can be seen that there are two different loading methods; the stress and deformation of pile end plate are quite different. Due to the existence of local separation, the maximum stress on the pile end plate is reduced by nearly 60%.

4. Optimization of Pile End Plate Design

In conclusion, the design method of using equivalent load instead is conservative. In fact, due to the redistribution of stress, the load on the pile end plate from the internal compression bar will be reduced. Therefore, for the design optimization of pile end plate, the reduction factor can be introduced to reduce the load, as follows:

\[ F' = F \gamma \]  

(1)

in formula: \( F' \) is design value of pile end plate load, \( F \) is actual load, \( \gamma \) is reduction coefficient of stress redistribution.

The reduction coefficient is selected according to the soil condition and reliability around the pile, and the approximate value is 0.6 from the finite element results.

Reference

[1] Zhang Nan. Experimental Research on Seismic Behavior of Assembled Combined Shear Wall [D]. Changchun Institute of Engineering

[2] Song Chenchen, Liu Jiming, AI tengtengteng, et al. Research on damage factors in ABAQUS concrete plastic damage model [J]. Engineering construction, 2017 (7)
[3] Shi Yong, Wang Meng, Wang Yuanqing. Study on constitutive model of structural steel under cyclic load [J]. Engineering mechanics, 2012, 29 (9): 92-98

[4] Shi Yiping. Explanation of ABAQUS finite element analysis example [M]. Beijing: Mechanical Industry Press, 2006.140-141

[5] Zhu Yiwen. Application of ABAQUS in geotechnical engineering [M]. Beijing: China Water Conservancy and Hydropower Press, 2010.94-96

[6] Cao Jinfeng. FAQ of finite element analysis of ABAQUS [M]. Beijing: China Machine Press, 2016.130-133

[7] Cheng Daxian Mechanical design manual [M]. Beijing Chemical Industry Press, 2004.1.8-10

[8] Prepared by China Architectural Standard Design & Research Institute Prestressed concrete pipe pile Atlas [M]. Beijing China Planning Press, 2010.

[9] Wang Yuzhuo ABAQUS structural engineering analysis and case explanation [M]. Beijing: China Construction Industry Press, 2010.

[10] Zhuang Zhuo Finite element analysis and application based on ABAQUS [M]. Beijing Tsinghua university Press 2009.