Finite element analysis of typical fastener type full hall steel tubular scaffolds

Fan Chen¹, Wendong Wan¹, Yibo Fan²*, Xiaohai Qi²

¹ East China Power Transmission and Transformation Engineering Company of State Grid, 518124, P.R.China
² China Nuclear Power Technology Research Institute Limited, Shenzhen, Guangdong, 200050, P.R.China
*Corresponding author’s e-mail: fanyibo@cgnpc.com.cn

Abstract. Based on fastener type full steel tubular scaffolds of power transmission and transformation project, this paper discusses in detail the establishment process of three-dimensional finite element model, including steel pipes, semi-rigid connections, mass of fasteners and boundary conditions. The stability of scaffold is analyzed and the failure mode of instability and ultimate bearing capacity are calculated. Then the static analysis is carried out to analyze the displacement and axial force response of the scaffold during construction, and finally the measures to strengthen the stability and safety of the scaffolds are given.

1. Introduction
Fastener type full hall steel tubular scaffolds are one of the most commonly used formwork support systems in substation construction. Since the steel pipes are connected by fasteners, and the method of erection is a structural unstable system. There has not been an appropriate theoretical calculation model to simulate and analyze the stress of the formwork support frame accurately under construction [1]. In addition, it is difficult to guarantee the quality of the components that make up the formwork support systems [2]. Therefore, there are hidden dangers in the stability of fastener type full hall steel tubular scaffolds. Scaffold collapses and casualties occur frequently in China.

Based on the science and technology innovation project of East China Power Transmission and Transformation Engineering Company of State Grid, aiming to demand of monitoring of fastener type full hall steel tubular scaffolds, the finite element analysis of the fastener type full steel tubular scaffold is carried out for the according to the requirement of design.

This project selected fastener type full hall steel tubular scaffolds between the axes 6-7 and E-F for analysis and calculation. Figure 4 is a brief profile of the scaffolds. The fifth steel pipe supports the beam formwork, which is shorter than others, and other steel pipes support the floor formwork.
Figure 1. The brief profile of the scaffolds

2. 3D model establishment

As shown in Figure 2, the three-dimensional finite element model of the scaffolds is established with ANSYS software. The height of the scaffolds is up to 11m, and the longitudinal and transverse length are 13.2m and 10.97m respectively in the horizontal direction. Simulations of steel pipe components, diagonal bracings, semi-rigid connections, fasteners, and boundary conditions are described in detail below. The total number of nodes and elements in the finite element model is 117,789 and 77,531, respectively.

2.1. Steel pipes simulation

BEAM189 element is used to simulate steel pipe components. BEAM189 is suitable for slender beam structures with 6 degrees of freedom at both ends. It is suitable for linear, large rotation angles and nonlinear large strains.

2.2. Diagonal bracings simulation

LINK180 element is used to simulate diagonal bracings. The function of diagonal bracings is to increase the overall stability of the support systems. The mechanical character of the diagonal bracings
is similar to the two-force rods, and LINK180 element is usually used to simulate trusses, ropes, connecting rods, springs, etc. The element does not bear bending moments, and has the functions of plasticity, creep, rotation, large deformation, large strain and so on.

2.3. Semi-rigid connections simulation
In traditional steel frame analysis and design, it is generally assumed that the connections are either completely rigid (the shear force and the moment can be both transferred) or ideally hinged (it can only transfer shear force). A large number of tests have shown that all connections are between rigid and hinge connection, which can be referred to as semi-rigid connection. The stiffness of the fastener is closely related to the mass and tightening degree of the fastener under load. In this paper, COMBIN14 spring element is used to simulate the fastener of the scaffolds. COMBIN14 has one-dimensional, two-dimensional or three-dimensional axial or torsion performance.

The specific simulation method is as follows: First, three nodes with the same coordinates are set at each node of the model. These nodes are used to generate vertical elements, horizontal elements and horizontal elements. Then, the three translational degrees of freedom (UX, UY, UZ) of the three nodes are coupled. Finally: the three nodes are connected in pairs with rotational springs respectively. According to the technical specification for construction fastener type full hall steel tubular scaffolds safety regulations, fastener tightening torque is generally between 40N.m and 50N.m, when tightening torque is 40N.m, moment-Angle tangent stiffness of springs is 19kN.m/rad.

2.4. Mass of fasteners Simulation
MASS21 element is used to simulate mass of fastener.

2.5. Boundary conditions
The bottom of the scaffolds is hinged.

3. Stability analysis
ANSYS software has a stability analysis function - buckling analysis. Buckling analysis is a technique used to determine the critical load and buckling mode shape of a structure when it begins to become unstable. Eigenvalue analysis is a common method of analysis, which is similar to the elastic buckling analysis method based on Euler's formula. The characteristic of calculation is that the eigenvalue of buckling analysis can be obtained quickly, and then the critical buckling force can be obtained.

3.1. The eigenvalue buckling analysis steps
The eigenvalue buckling analysis consist of four steps: (1) Establish model; (2) Obtain static solutions; (3) Obtain eigenvalue distortion; (4) Extend the solution.

3.2. Modal analysis
The failure mode of instability of the scaffolds is shown in Figure 3. As is shown in Figure 3, it can be seen that the instability is in the form of overall instability, and the main deformation position is the middle and lower part, forming a wavy drum. The critical buckling force of the scaffolds is 25.91 kN.
4. Static analysis
The finite element static analysis is carried out in this paper, focusing on the response of displacement and axial force.

4.1. Applied load
Dead load of reinforced concrete beam: 25.1kN/m³ (including reinforcement);
Formwork dead load: 0.3kN/m³;
Live load (load of construction and vibrating concrete): 2.5kN/m³;
The dead load of steel pipes and fasteners is applied by means of applying gravity.

Wind load and seismic load are not considered in this paper. Fastener type full steel tubular scaffolds are usually set up in a relatively closed space, so the wind load can be ignored. In addition, the occurrence of an earthquake during the construction is an event with very low probability, so the earthquake effect is not considered [3].

4.2. Displacement response
Figure 4 shows the displacement nephogram in x-direction of the scaffolds. The maximum displacement in the x-direction is 1.75mm. The position of maximum displacement is at the top of the vertical steel pipes adjacent to the beam formwork. And displacement at the top of the scaffolds is large, and the displacement at the top of the vertical steel pipes adjacent to the beam scaffolds is the largest.
Figure 4. The displacement nephogram in x-direction

Figure 5 shows the displacement nephogram in y-direction of the scaffolds (the settlement of the formwork above). The maximum displacement in the y-direction is 0.0881mm. The displacement in the y-direction of the scaffolds is small, and the top displacement of the vertical steel pipes adjacent to the beam formwork is the largest.

Figure 5. The displacement nephogram in y-direction

Figure 6 shows the displacement nephogram in z-direction of the scaffolds. The maximum displacement in z-direction is 1.37mm, which is close to the top of the vertical steel pipes.
Figure 6. The displacement nephogram in z-direction

Figure 7 shows the total displacement nephogram of the scaffolds. The maximum total displacement of the scaffolds is 2.12mm, which is at the top of the vertical steel pipes adjacent to the beam formwork. And the displacement at the top of scaffolds is large, while the displacement at the bottom of scaffolds is small.

Figure 7. The total displacement nephogram of the scaffolds

4.3. Axial force response

Figure 8 shows the axial force nephogram of the scaffolds. The maximum axial force is 9.011kN, and is located at the bottom of the corner of the scaffolds.
4.4. Calculation of inclination angle

According to the analysis of displacement response in Section 4.2, the maximum horizontal displacement calculated in this paper is 1.75 mm. The inclination angle $\theta$ is calculated by the following formula:

$$\theta = \arctan\left(\frac{1.75}{11000}\right) = 9.11 \times 10^{-3} (\degree)$$  (1)

In general, the inclination angle of the whole scaffolds is very tiny.

5. Conclusion

Based on Songze power transmission and transformation project, this paper discusses in detail the establishment process of three-dimensional finite element model, including steel pipes, semi-rigid connections, mass of fasteners and boundary conditions. The stability analysis and static analysis of fastener type full steel tubular scaffolds are carried out in this paper, and it can conclude as follows:

- The buckling instability of scaffolds is the overall instability;
- The top of the scaffolds has a large horizontal displacement, and the position near the formwork has the maximum displacement;
- The axial force of steel pipes has little change in axial direction, and the maximum axial force is located at the bottom of the corner of the scaffolds;
- The inclination angle of the whole scaffolds is very tiny.

Acknowledgments

This paper was supported by the science and technology innovation project of East China Power Transmission and Transformation Engineering Company of State Grid.

References

[1] Jianmin Liu, Huimin Li. (2005) Review and consideration on the calculation model of fastener type steel pipe formwork support frame. Architecture Technology, 036(011):860-862.
[2] Weiguo Su, Jian Liu. (2013) Finite element analysis of the full house brackets of cast-in-situ box girder. Journal of South China University of Technology, (02):88-93.
[3] Hailang Chen, Huang Wang, Zengfeng Zhang. (2009) Calculation and analysis of fastener type steel pipe high support frame based on ANSYS. Zhejiang Architecture, 026(003):38-43.