Simulation Analysis of Strength of Double Layer Structure of Prefabricated Substation

Hailong Li  
State Grid Liaoning Electric Power Company, Shenyang, Liaoning, China

Yun Fu  
Shenyang Power Supply Company, State Grid Liaoning Electrical Power Company, Shenyang, Liaoning, China

Chundong Li  
Liaoning Electric Energy Development Co. Ltd., Shenyang, Liaoning, China

Jing Gao  
State Grid Liaoning Electric Power Company, Shenyang, Liaoning, China

Shishen Guan  
Liaoning Electric Energy Development Co. Ltd., Shenyang, Liaoning, China

Jun Zhao  
State Grid Liaoning Electric Power Company, Shenyang, Liaoning, China

ABSTRACT: The construction of the substation in the urban center has become a common phenomenon. There is a need to consider the cost of land, and make the substation covering a small area, safe and economical. This paper presents a new model of substation built with assembled upper and lower double layer structure of module blocks in the urban center, and proposes to install 110kV of prefabricated cabin on the top of 10kV and 35kV of prefabricated cabin, with a small area and easy construction, but the strength of double layer structure becomes the key to design. Based on the finite element theory, this paper establishes a finite element model of double layer structure of prefabricated substation in the finite element software ABAQUS, and carries out strength simulation analysis of the most unfavorable conditions, of which the inclined degree of the entire structure is 5°, and the seismic intensity is 8°. The research results show that: when the inclined degree of the entire structure is 5°, the maximum stress of the structure is 92.49MPa, and the safety factor of strength is 2.5; when the seismic intensity is 8°, the horizontal load excitation has a greater impact on the double layer structure, the maximum stress is 160.5MPa, and the safety factor is 1.5; the vertical load excitation has a less impact, and the maximum stress is only 30.3MPa. In the early period of design of the strength simulation analysis method, this paper can obtain the safety factor of strength of structure under the most unfavorable conditions, which has reference significance on the design and optimization of the structure of substation.

Keywords: prefabricated cabin; distribution type; substation in the urban center; double layer structure; yield strength

1 INTRODUCTION

With the accelerating process of urbanization, in order to meet the ever-increasing requirements on the electricity utilization in the urban center, and ensure the electric power supply in the whole city, the construction of the substation in the urban center has become an increasingly common phenomenon. If the substation is to be built in the urban center area, there is a need to not only consider the extent of dense pop-
ulation in urban areas, urban planning and architectural style, but also consider the cost of land, and make the built-up substation covering the minimal area.

The program of compact and hermetic double layer structure of prefabricated substation not only solves puzzling and difficult problems of the electromagnetic and noise interference, but also solves the problems of covering a large area and a long construction period, which creates a new type of fast and efficient, compact and safe and environmental model of substation building [1]. In order to compress the covering space, the design scheme uses the double layer structure (As shown in Figure 1): to install 110kV of prefabricated GIS composite apparatus on the top of 10kV of prefabricated switching station and the prefabricated cabin of the secondary combined device as an upside down T-shaped layout, and to symmetrically install the prefabricated transformer on the other side of the fire-fighting access. Such an arrangement form can control the total area of substation area within 1 acre of land (667 square meters), saving land and taking full advantage of the space.

![Figure 1. Double layer structure of prefabricated substation.](image)

For the new model of substation built with assembled upper and double layer structure of module blocks in the urban center developed by the company, to install 110kV of prefabricated cabin on the top of 10kV and 35kV of prefabricated cabin, with a small area and easy construction, but the strength of double layer structure becomes the key to design. In order to ensure that the designed and developed double layer structure meets the requirements of strength, this paper uses a large nonlinear finite element software ABAQUS [2-3] in the research and development process, in order to carry out simulation analysis of strength under two kinds of most unfavorable conditions (in the static state, the inclined degree of the entire structure is 5°, and the seismic intensity is 8°) [4-6], and obtain the corresponding safety factor.

2 BASIC THEORY OF FINITE ELEMENT METHOD

The finite element method is a kind of numerical computation method for engineering developed in the 1960s [7-8]. With the development of electronic computer technology, this method has become a powerful tool for engineering numerical analysis. Especially in the field of solid mechanics and structural analysis, the use of it has successfully solved a large number of problems with important significance.

The finite element method is essentially a kind of approximating numerical computation method in the mechanical model, and its basic idea is to continuously solve a set of assembly with finite units connected together in accordance with a certain way in discrete zones. The unit can be combined in different ways, and the unit itself has different shapes, so the solution domain with a complex geometrical shape can be modeled. Another important feature of the finite element method is: to solve the unknown field functions to be solved in the whole solution domain in fragmentation by the use of approximate function given in each unit, while the approximate function in the unit is generally represented by the unknown field function and numerical values and interpolation functions of its derivatives in each node of the unit. Thus, in the finite element analysis of an issue, the unknown field function and numerical values of its derivatives in each node of the unit become a new unknown quantity (generally referred to as the degree of freedom), so that the issue of a continuous infinite degree of freedom is turned into an issue of discrete finite degree of freedom. Once these unknown quantities are obtained, the approximate value of the field function in each unit can be calculated through the interpolation function, thus obtaining the approximate solutions of the entire solution domain. Obviously, with the increase of the number of unit, or with the increase of the number of degree of freedom of unit and the improvement of the precision of interpolation function, the degree of approximation of solutions will be improved continuously. If the unit meets the requirements of convergence, with the increase of number of unit, the approximate solution will eventually converge to the exact solution [9].

For the displacement method in the elastic mechanics, the steps of solving the finite element method are as follows:

1) To discretize the non-individual body, namely: to discretize continuous solution domain as a set of assembly with finite “units” constructed by virtual lines or planes. Such an assembly can analytically simulate or approximate the solution domain;
2) Assuming that the above “units” are connected with nodes on the border of units, to view these nodal displacements as the basic unknown quantity;
3) To use unknown quantities of the node proposed in (2), and select a set of interpolation function to uniquely define distribution of corresponding physical field (displacement, stress and strain) in each unit, that is, to select the mode or column type of the unit;
4) To convert various types of loads into equivalent load just acting on the node, and establish the basic
equations of basic unknown quantity and equivalent nodal loads, namely, balance equation;
5) To solve the basic equation, and obtain answers of the basic unknown quantity.
Thus, the unknown field functions in the entire solution domain can be approximately represented by the numerical values and interpolation functions on the nodes of each unit, so that an issue of infinite degree of freedom in theory can be converted into an issue of finite degree of freedom of finite nodes of the unknown field function, and complex issues unable to be achieved in the past can be easily realized by the use of current efficient computer.

3 FINITE ELEMENT MODEL

ABAQUS software is a kind of nonlinear finite element analysis software developed by Dassault SIMULIA Company, which is one of the most advanced large-scale finite element analysis software in the world [2-3]. It can not only solve the majority of linear and nonlinear problems, but also can research complex solid mechanics and structural mechanics. In view of this case, the paper uses such software for static engineering analysis and seismic load effect analysis of the compact and hermetic double layer structure of the prefabricated substation.

Table 1. Mechanical parameters of Q235 steel.

<table>
<thead>
<tr>
<th>Material No.</th>
<th>Density (kg/m³)</th>
<th>Young’s modulus (GPa)</th>
<th>Poisson’s ratio</th>
<th>Yield strength (MPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Q235</td>
<td>7850</td>
<td>210</td>
<td>0.3</td>
<td>235</td>
</tr>
</tbody>
</table>

4 SIMULATION CALCULATION AND ANALYSIS

For the double layer structure of prefabricated substation, this paper carries out simulation calculation under two kinds of the most unfavorable conditions, which are respectively the static condition of uneven foundation settlement and dynamic condition of (horizontal, vertical) seismic load effect on the supporting system (box foundation). In the process of analysis, it is assumed that the inclined degree of the entire double layer structure is 5°, and the seismic fortification intensity [7] is 8°.

4.1 Simulation calculation and analysis of static conditions

For static conditions, the most unfavorable conditions are selected to be analyzed, of which the inclined degree of the entire supporting system is 5°, and the load is 100t. The forces of entire structure are shown in Figure 4.

The vertical gravity load is applied to large-scale nonlinear finite element simulation software ABAQUS, and Static analysis steps are selected, and the initial increment is set as 0.1. The solver automatic control incremental method can be used for iterative analysis and calculation. After completion of the simulation analysis, the distribution of Mises stress and
distribution of displacement of double layer structure are respectively shown in Figure 5 and Figure 6.

Through observation of analysis results of Figure 5 and Figure 6, we can find that:

1) In the static state of the supporting system, with the inclined degree of 5°, and the load of 100t, the maximum stress of double layer structure is 92.49 MPa, appearing at the bottom of the column in the middle of the second row, and the maximum stress is far less than the yield strength of 235 MPa of Q235 steel, the strength meets the safety requirements, and the safety factor is 2.5 (235 / 92.49).

2) The maximum displacement U is 6.71 mm under this condition, appearing in the central region of two long beams, and the maximum displacement value can meet the requirements of stiffness in normal use, and have a large safety stock.

4.2 Simulation calculation and analysis of seismic conditions

Simulation calculation process takes into account the simulation of seismic conditions, of which the seismic fortification intensity is 8° [10-11], and they are divided into horizontal and vertical seismic load spectrum excitation for analysis and calculation, in order to examine whether the supporting system of the entire double layer structure meets the requirements of seismic fortification.

For the analyzed and selected characteristic curves of the seismic wave and structural resilience, the stepwise integration method can be used to directly integrate dynamic equation, in order to obtain each instantaneous displacement, velocity and acceleration response of the structure during an earthquake, and facilitate to observe the internal force changes from elastic to inelastic stage, and the extent of component cracking, damage and even the structure collapsed under the effect of strong earthquake.

4.2.1 Horizontal load spectrum excitation

For the simulation calculation of the supporting system of double layer structure under the horizontal load spectrum excitation, the function of Amplitude magnitude curve in ABAQUS software can be used to apply the horizontal seismic load into the structure. Static General analysis steps are selected to simulate the stress changes of structure in the whole applied process of seismic response spectrum, and the analysis shows that,

1) The maximum stress of the supporting system substantially appears in the same region at different moments. When the peak stress occurs at 5.4 s, the stress distribution of supporting system at this moment is shown in Figure 7-8, and the maximum stress is 160.5 MPa (appearing on the end edge of the column), which is also far less than 235 MPa, and the safety factor is 1.46, meeting the strength requirements.

2) Figure 9 shows Mises stress time history curve of the maximum stress of 160.5 MPa. Before 3 s of applying the horizontal seismic load, the stress value is relatively small, and then the stress value gradually increases, and becomes the maximum at 5.4 s.
4.2.2 Vertical load spectrum excitation

For the simulation analysis of the supporting system of double layer structure under the vertical load excitation, the analysis methods and operational process are similar to that of the horizontal load excitation. The analysis results show that:

1) The maximum stress of the supporting system at each moment appears in the same region.

2) When the maximum stress appears at 4.55s under the vertical load excitation, the stress distribution of supporting system at that time is shown in Figure 10-11. As can be seen from the diagram, the stress distribution of the entire supporting system is as follows: the stress at the end is larger, stress in the middle area is smaller, and the maximum stress appears at the end of the shorter supporting system.

3) The maximum stress of the entire supporting system is 30.3MPa under the vertical load excitation, which is smaller than the maximum stress of 160.5MPa under the horizontal load excitation, far less than the yield strength of the material of 235MPa, and the safety factor is 7.8, indicating that the structure is safe and reliable.

Figure 12 shows the stress time history curve of the maximum stress location under the vertical load excitation. As can be seen from the diagram, at 0-1s, the stress increases linearly with the time, and increase rapidly, and then enters the shock area, and becomes maximum at 4.55s.

4.2.3 Simulation analysis results under seismic load effect

Though the introduction of the simulation analysis process in Section 4.2.1 and Section 4.2.2, the following conclusions can be drawn: when the seismic intensity is 8°, the horizontal load excitation has a greater impact on double layer structure, the maximum stress of the entire supporting system is 160.5 MPa, and the safety factor is 1.5; the vertical load excitation has a less impact, and the maximum stress is only 30.3MPa. Therefore, the prefabricated substation adopts double layer to build substation, and its supporting system can meet the seismic effect, with the seismic fortification intensity of 8°.

5 CONCLUSION

Based on the finite element software ABAQUS, this paper establishes the finite element model of double layer structure of prefabricated substation, and carries out strength simulation analysis of the most unfavorable conditions, of which the inclined degree of the entire structure is 5°, and the seismic intensity is 8°:

1) In the static state of the supporting system, with the inclined degree of 5°, and the load of 100t, the maximum stress of double layer structure is 92.49MPa, appearing at the bottom of the column in the middle of the second row, and the strength meets the safety requirements, and the safety factor is 2.5.

2) When the seismic intensity is 8°, the horizontal load excitation has a greater impact on the double layer structure, the maximum stress is 160.5MPa, and the safety factor is 1.5; the vertical load excitation has
a less impact, and the maximum stress is only 30.3MPa.

3) Based on the finite element software ABAQUS, this paper carries out simulation analysis of the strength, stiffness, stability and other aspects of the structure. In the early period of design, this paper can obtain the safety factor of strength of structure under the most unfavorable conditions, which provide reference for improving design and optimizing structure.

REFERENCES


