Reliability analysis of numerical simulation of compression of coarse-grained soil foundation

Xiyu Bai*, Wenxian Zhang
Tibet Agriculture & Animal Husbandry University, Nyingchi Tibet, China

*Corresponding author: xiyubai@xza.edu.cn

Abstract. In order to study the reliability of numerical analysis in the calculation of foundation compression, this paper takes the coarse-grained soil foundation of a project in Milin, Tibet as the research object, uses the mechanical parameters obtained from the large-scale triaxial test and the numerical analysis of ABAQUS software to analyze and compare the compression value and load. It is considered that the load and compression situation in the numerical analysis are relatively reliable, which comprehensively reflect the stress and strain of soil under overlying load. It can be further studied according to the change of boundary conditions in engineering application.

Keywords: Numerical analysis, Foundation, Compression, Reliability.

1. Preface
As a general elastic-plastic material, coarse-grained soil will deform under external force. For coarse-grained soil foundation, due to the compressibility of soil, the foundation soil is compressed and deformed under the vertical load of buildings or structures on it, which will affect the safety of building structure. The calculation of foundation compression is a classical problem in soil mechanics, and finite element numerical simulation is a popular geotechnical engineering analysis method at present. For the foundation engineering, the overload method is generally adopted. By increasing the load step by step until the soil is in the limit state, the whole process of the stress and deformation of the foundation soil is analyzed, and the foundation compression amount and the damaged form are obtained [1]. In view of the compression of coarse-grained soil foundation under overburden load in a project in Milin, Tibet, this paper uses ABAQUS software to simulate the development process of soil stress and strain through different load levels on two groups of samples, and obtains the final amount of compression. Finally, by comparing with the data in large-scale triaxial test, the reliability of this method is analyzed.

2. Overview of ABAQUS software
ABAQUS software is a finite element numerical analysis software. It has a rich element library, which can simulate any complex shape materials. It also has a wide range of yield criteria. It can simulate the linear and nonlinear behavior of most materials. Through the combination of material library and element library, it can solve the problems of different materials and complex boundary conditions. This software is very suitable for geotechnical engineering research.
3. Finite element equal incremental loading method

Equal increment loading method is to increase the load step by step, and the difference of each stage is the same. In the process of soil compression, the corresponding stress and deformation development are measured. Because the stress-strain relationship of soil in the process of compression is non-linear, in order to simplify the calculation, the equal increment is used for loading, and the smaller the amount of each loading, the more accurate the result is. When the material is in the final state, the load value plus the stress in the initial equilibrium state, the sum is the stress value, and the final state can be until the soil breaks through the critical value.

4. Selection of yield criterion

The ideal elastic-plastic model is used in this numerical simulation. As a general elastic material, coarse-grained soil yields under the combined action of shear stress and vertical stress. When the soil changes from elastic state to plastic state, this condition is not only the yield condition, but also the critical limit condition. Mohr Coulomb yield criterion is usually used in geotechnical analysis [2].

The yield surface of Mohr Coulomb criterion in the principal stress space is an irregular hexagonal cone, and its projection in the plane is an irregular hexagon, and the yield surface has sharp top and edge angle. The results under the criterion are safe, and the numerical value of the corner points of the yield curve is missing a singular point, which leads to the calculation complexity and slow convergence. The derivative value of the yield function along the normal direction of the surface is not easy to determine, which leads to the non-unique plastic flow direction. Therefore, it is assumed that the yield surface is smooth in ABAQUS calculation, and the elastic potential function under the criterion is smoothed to ensure the uniqueness of plastic flow direction.

5. Selection of flow law

In the finite element numerical analysis, the flow rules include non-associated flow rule and associated flow rule. According to the generalized plastic theory [3], the associated flow rule can be regarded as a special case of the uncorrelated plastic flow rule, which is only applicable to the material whose plastic potential surface coincides with the yield surface. In general, the associated flow rule is used for metal materials, and the non-associated flow rule is used for geotechnical materials.

In mechanical analysis, the dilatancy of coarse-grained soil is the result of the interaction of spherical tensor and deviator tensor, so the influence of dilatancy angle must be considered in geotechnical engineering analysis. According to the slip line field theory, the angle between the velocity vector direction of geotechnical materials and the failure surface is equal to half of the angle of internal friction [4].

6. Establishment of calculation model and selection of parameters

6.1. Model units and physical parameters

The foundation is mainly affected by the overlying vertical load. The foundation soil actually bears the shear failure effect, without tension or torsion, and is little affected by Poisson's ratio and elastic modulus of soil [5]. Therefore, the numerical simulation mainly focuses on different cohesion, internal friction angle and incremental load.

The test data are directly obtained or calculated from large-scale triaxial tests. It is shown in the table below:

| sample | $E$ (MPa) | $\mu$ | $\gamma$ (kN/m$^3$) | $\rho$ (kg/m$^3$) | $c$ (kPa) | $\phi$ (°) | $\psi$ (°) |
|--------|----------|-------|---------------------|------------------|----------|-----------|-----------|
| No.1   | 63       | 0.3   | 23                  | 2300             | 84.6     | 35.8      | 17.9      |
| No.2   | 30       | 0.3   | 25                  | 2500             | 71.5     | 39.49     | 19.75     |
6.2. Prescribing conditions

In order to simplify the simulation process and accurately reflect the influence of overlying load on the compression value, the following assumptions are specified before calculation:

(1) The foundation soil is defined as ideal isotropic material, the action depth of the foundation soil is set as 1m, and the size of the foundation soil is 1m * 1m * 1m;

(2) The material is only subjected to the vertical load of the upper part, and the surrounding of the soil model is constrained by the fixed support on the bottom surface, that is, the horizontal displacement is zero, and the vertical displacement is not limited;

(3) The equilibrium initial stress field is established by manual input;

(4) This simulation mainly checks the change of stress and displacement within 4% of axial deformation, taking the corresponding triaxial test load at 600kpa;

(5) The soil is located above the groundwater level, that is, the buoyancy effect of water is not considered.

6.3. Grid division

Mesh generation is an important part of finite element simulation. The size of the model is determined by the form and quantity of mesh, which affects the calculation accuracy of the model. This simulation is mainly static analysis, uniform load. After setting the boundary conditions, take 0.1M as the node and divide the 3D mesh according to the hexahedral reduced integral element, as shown in the following figure:

![Figure 1. grid division view](image)

6.4. ABAQUS post processing

ABAQUS has a powerful post-processing module, which can directly generate model and visualization module by calculating data. In this simulation, isotropic soil is used, and uniform load is applied on the upper part, so the displacement and stress can be observed in two-dimensional plane, as shown in figures 2 to 13:

![Figure 2. stress nephogram and displacement nephogram of No.1 sample at 100KPA](image)
Figure 3. stress nephogram and displacement nephogram of No.1 sample at 200KPA

Figure 4. stress nephogram and displacement nephogram of No.1 sample at 300KPA

Figure 5. stress nephogram and displacement nephogram of No.1 sample at 400KPA

Figure 6. stress nephogram and displacement nephogram of No.1 sample at 500KPA
Figure 7. Stress nephogram and displacement nephogram of No.1 sample at 600KPA

Figure 8. Stress nephogram and displacement nephogram of No.2 sample at 100KPA

Figure 9. Stress nephogram and displacement nephogram of No.2 sample at 200KPA

Figure 10. Stress nephogram and displacement nephogram of No.2 sample at 300KPA
According to the settlement nephogram of the two groups of soil samples, it can be seen that there is no stress concentration phenomenon in the soil under the vertical uniformly distributed graded load, the compression trend is stable and uniform, the compression amount increases with the increase of the upper load, and the compression diffusion range develops from top to bottom, which conforms to the characteristics that the additional stress of the foundation soil decreases with the increase of depth. The results of numerical analysis are as follows:

Table 2. sorting values of calculation results

| sample | Convergence value (kPa) | Initial ground stress (kPa) | Bearing capacity (kPa) | Compression (mm) |
|--------|------------------------|----------------------------|------------------------|-----------------|
| No.1   | 353.3                  | 23                         | 376.3                  | 27.41           |
| No.2   | 355.1                  | 25                         | 380.1                  | 19.13           |
7. Verification of validity of numerical Analysis

The stress and strain curves are derived from ABAQUS and compared with the data in large-scale triaxial test, as shown in the following figure:

![Figure 14. P-S curve of sample No.1](image1)

![Figure 15. P-S curve of sample No.2](image2)

After comparing the results of triaxial test and numerical analysis, it can be seen that the overall trend of the two methods is basically fitted and conforms to the law of strain hardening. Considering that the pressure loading process in static analysis is instantaneous, and the coarse-grained soil material in the model is an ideal isotropic soil constrained around, and in the triaxial test, the specimen bulges around after being subjected to axial stress, so the compression value of numerical analysis under the same load must be smaller than that of large-scale triaxial test. The comparative analysis between them proves that the load and settlement in the numerical analysis is more reliable.
8. Summary

(1) In this paper, the incremental loading method is used to simulate the bearing capacity of foundation soil. Different from the traditional limit analysis method, the development of soil stress and deformation is obtained. The vertical compression and foundation bearing capacity are obtained by using Mohr Coulomb yield criterion, non-associated flow rule and surrounding constraint conditions;

(2) For the same soil sample, the change trend of soil mass is consistent with the direction of stress loading. The compression deformation increases with the increase of load, and there is no obvious stress concentration phenomenon. The compression condition is uniform. The upper part of the soil is obviously compressed, and the deformation decreases with the increase of depth. The compression mainly occurs in the upper shallow soil area;

(3) Due to the different settings of test conditions, the results of numerical analysis are different from those of large-scale three-week test, but the trend and law of numerical analysis results are similar to those of triaxial test, which shows that the results of finite element numerical analysis are relatively reliable and can be simulated by changing boundary conditions or load conditions according to the actual situation of the project.

References

[1] Li Gang. Calculation of foundation bearing capacity based on finite element limit equilibrium method [J]. Water conservancy planning and design, 2018, (1): 75 - 78.
[2] Liu Ying, Yu Lihong. Application of Mohr Coulomb yield criterion in geotechnical engineering [J]. World geology, 2010, (1): 633 - 639.
[3] Zheng Yingren. Theory of generalized plastic mechanics [J]. Geotechnical mechanics, 2000, (1): 188 - 192.
[4] Kong Weixue, Rui Yongqin, Dong Baodi. Discussion on the selection of dilatancy angle of geotechnical materials under the non associated flow rule [J]. Geotechnical mechanics, 2009, (1): 3278 - 3282.
[5] Lian Zhenying, Han Guocheng, Kong Xianjing. Study on stability of excavated slope by strength reduction finite element method [J]. Acta geotechnical engineering, 2001, 23 (4): 407 - 411.