Analysis of Embankment Slope Stability Based on the Strength Reduction Finite Element Method of Abaqus

Jian Mei

School of civil engineering and architecture, Southwest Petroleum University, Sichuan, 610500, China

1432558386@qq.com

Abstract

In this article, abaqus is combined with strength reduction finite element method to analyse the practical problems in embankment slope engineering.Cohesive force and angle of internal friction of soil is changed by alterring the strength reduction coefficient Ft during the process of simulation calculation.The reduction coefficient of embankment slope which is in a critical state is thinked as the minimum stability coefficient.

Keywords

Abaqus, slope, finite element, reduction coefficient.

1. Introduction

Geological disasters bring huge economic losses every year in China,so the analysis on the stability of the slope is of great significance.Slope stability research is a classic topic in geotechnical engineering, Limit equilibrium analysis method and finite element method[1] are two dazzling Stars in all of analysis methods.The present numerical analysis methods can get stress, displacement and plastic zone of slope normally, but it can`t get dangerous sliding surface of slope and corresponding safety coefficient.With the development of computer technology, especially the development of elastic-plastic finite element calculation of rock and soil materials technology, the application of finite element method to analyse the slope stability is becoming more and more commoniy.

Finite element method overcomes shortcomings which assumes soil as rigid body in limit equilibrium analysis method,Considering the soil elastoplastic constitutive relation, and the influence of deformation on the stress,it can simulate instability process of slope,stress distribution, plastic zone and displacement of slope,It can be applied to any complex boundary conditions.

In this article, using abaqus to gain cloud pictures which can show the development of plastic zone under the condition of a given strength reduction coefficient.The plastic zone did`t reach a breakthrough or a local yield failure,it shows that the slope is stable under the reduction coefficient;Increasing the reduction coefficient,and the coefficient can be thinked as the minimum stability coefficient when plastic zone reaches the top of the slope.

2. Principle of strength reduction finite element method

Strength reduction finite element method[2-4] is an extensive numerical analysis method in slope stability analysis.It combines strength reduction technology with the finite element method to gain the minimum safety coefficient of slope by adjusting the reduction factor to analyses the stability of slope under the given evaluation indicators.Safety coefficient,used in engineering commonly,is under the condition that material strength is certain to increase the load ratio,but strength reduction coefficient is to decrease of strength of the material.They have the same purpose essentially.

The basic principle [5] of strength reduction finite element method is to get a new set of rock mass strength index which is the resultant that c , φ is divided by a reduction coefficient (F_t). Then put the new material parameters into finite element to calculate many times until the slope soil in accordance with the given critical damage state judgement conditions, and the reduction coefficient (F_t) is the minimum stability coefficient.

3. The relationship between abaqus and embankment slope instability

3.1 Brief introduction of abaqus[6]

Abaqus is one of the famous nonlinear finite element analysis software in the world.The software has excellent nonlinear solving ability and powerful simulation performance, it has a variety of unit model library, it can reflect the character of soil constitutive model really,and it can solve the process of the specific problem of geotechnical engineering, so it`s widely used in the field of geotechnical engineering.Abaqus offers a wide range of user subroutine as a second development platform, users can define a specific model according to their own need.These user subroutine covers the basic model of the whole category basically.Such as,user-defined load, user-defined material and unit,etc.

A powerful post-processing function of abaqus software provides widely choices for the description of the calculation results and interpretation.in addition to the usual cloud contour and animation display, contour, curve and other common tools are ways to complete the project.For plastic strain of slope calculation results, using cloud can accurately show the size of generalized plastic strain value, position and development condition of the plastic zone clearly.

3.2 Embankment slope instability criterion

Criterion of the slope unstability affects the correction of strength reduction directly,and the accuracy of the existing instability criterion can be divided into three kinds generally[7].

Strength parameters after reducting makes calculation can't convergence:Nonlinear finite element equations of the slope in the iteration process is in a state of limit equilibrium will not convergence, but this method is restricted in the geotechnical engineering problems,it has poor applicability.

Characteristics of displacement or strain of slope mutated and develope unlimitedly:With a characteristic part of displacement inflection point to judge the slope instability,this method has clear physical significance.

Plastic zone reachs the top of the slope:When the plastic zone in the domain connected, then judge the slope damages.For plastic strain,the size of the plastic strain values, the location of the plastic zone and the plastic zone development can be showed in cloud way clearly.

4. Analysis of projects

Using abaqus to analyze embankment slope under self-gravity simulation, and verify the rationality and feasibility of the method.

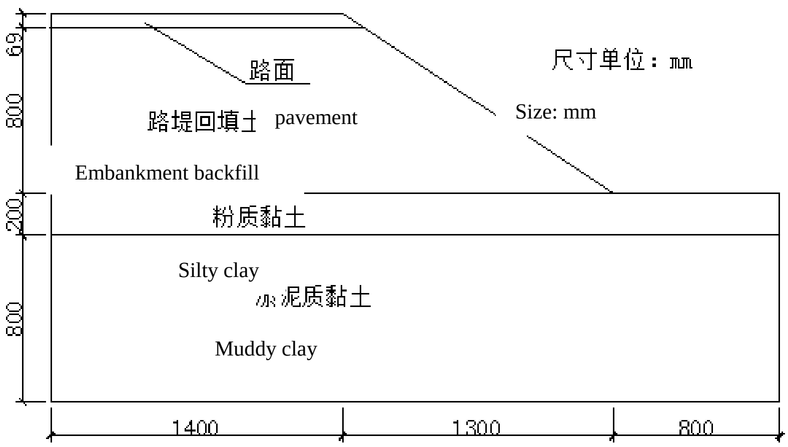

Figure 1 a model for the slope of embankment

Pavement uses linear elastic model, the remaining materials use Mohr Coulobm elastoplastic model,the slope model parameters is shown in table 1.

4.1 An engineering example problem description

As shown in figure1, thick of pavement is 69cm,embankment is 8m high, thick of silty clay is 2m , silty clay is 8m , slope ratio is 1: 1.5.

Table 1 Embankment slope model calculation parameters

Material type	γ_d (KN/m^3)	c (KPa)	φ (°)	E(kPa)	μ
Pavement materials	23	\	\	1.2×10^6	0.3
Embankment backfill	18.2	15	25	50000	0.3
Silty clay	18.2	15	25	50000	0.3
Muddy clay	18.2	15	25	50000	0.3

4.2 Description of model building process

Establishing a two-dimensional plane strain model, eight-node plane strain element (CPE8) is used on slope mesh. Soil obeys Mohr-Coulomb constitutive criteria. Building model (slope) to solve the initial stress balance calculations, and then constantly adjusting reduction coefficient and observeding displacement and plastic zone of the slope by cloud. the coefficient can be thinked as the minimum stability coefficient when plastic zone reaches the top of the slope. The process of establishing model of eembankment slope is as follows.

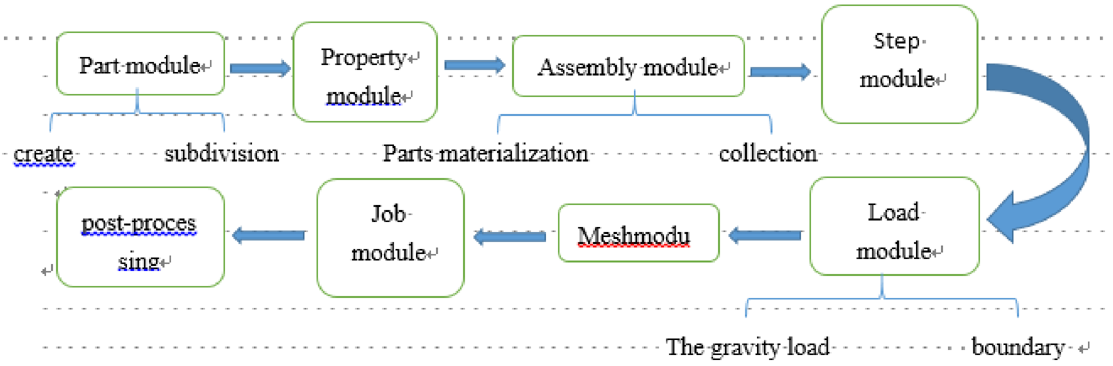

Figure 2 Flow Chart of Model

In the process of post-processing, c , φ of soil are reducted three times,and reduction parameters are as follows.

Table 2 Reduction Parameters

F_t	1.0	1.5	1.7	1.75
c (KPa)	15	10	8.8	8.57
φ (°)	25	17.2	15.3	14.9

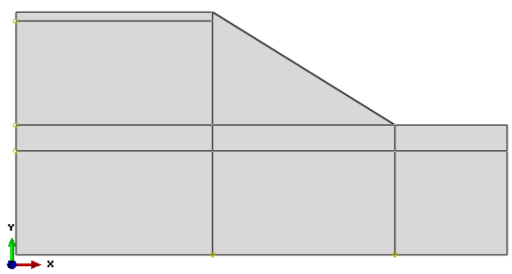

Figure 3 Component model after spliting

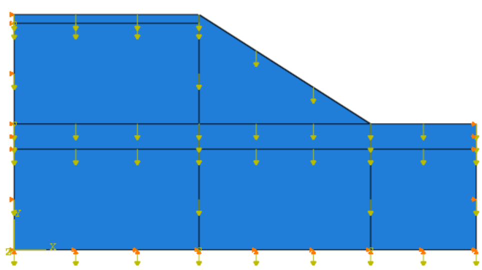

Figure 4 Model after applying gravity load

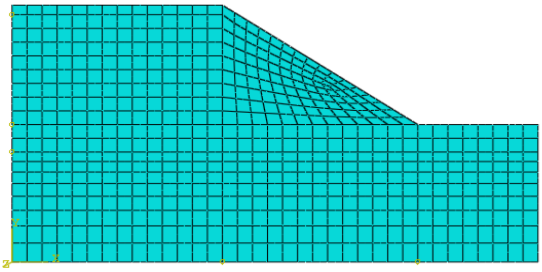

Figure 5 Complete meshed model diagram

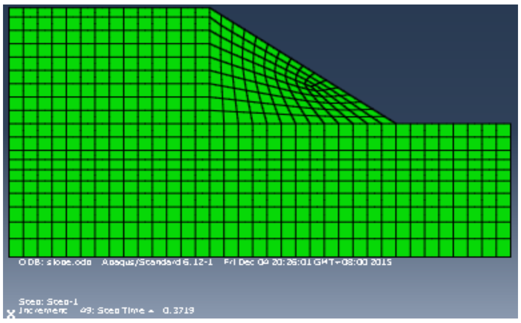

Figure 6 Post-processing cloud image before deformation

4.3 Analysis of the Results

Reduction coefficient F_t=1: Embankment slope behaves compression deformation Under the action of gravity, in another words it behaves consolidation settlement.As shown in figure 7 and figure 8.

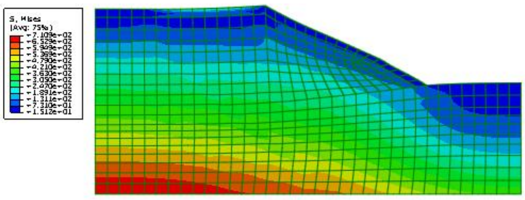

Figure 7 Cloud picture after deformation (Ft = 1)

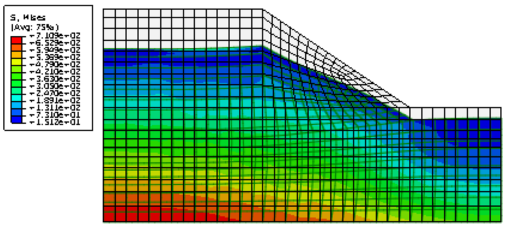

Figure 8 Compared before and after deformation (Ft = 1)

Reduction coefficient F_t=1.5:plastic area is becoming on the foot of slope obviously(curved in figures),slope slide shift phenomenon is becoming on the surface of the embankment and subgrade border(circled in figures), But plastic region is not obvious.as shown in Figure 9 and 10.

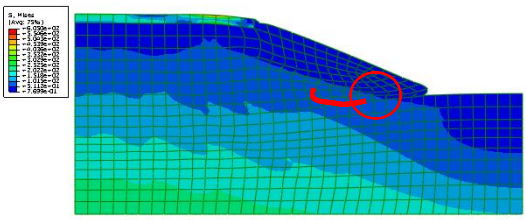

Figure 9 Comparison chart before and after deformation (Ft = 1.5)

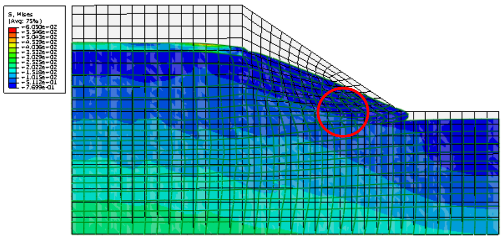

Figure 10 Cloud picture after deformation (Ft = 1.5)

Reduction coefficient F_t=1.7:The plastic area of Slope on the foot is becoming bigger than before.Slope slide shift phenomenon is becoming bigger than reduction coefficient Ft=1.5,on the surface of the embankment and subgrade border(circled in figures),The top of the slope has the trend that the pavement ruptures from slop.As shown in Figure 11 and 12.

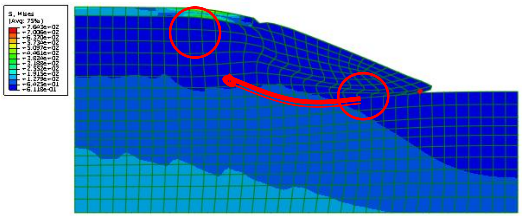

Figure11 Comparison chart before and after deformation (Ft = 1.7)

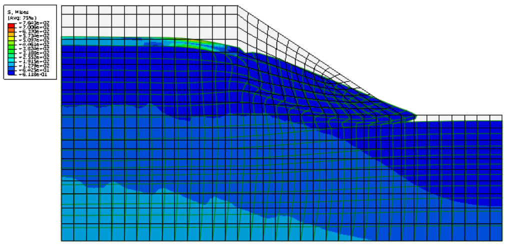

Figure12 Cloud picture after deformation (Ft = 1.7)

Reduction coefficient F_t=1.75: The plastic zone on the foot of the slope will reach the top of the slope (curved in figures), the slope shift phenomenon is more pronounced than before (circled in the figure). As shown in Figure 13 and 14.

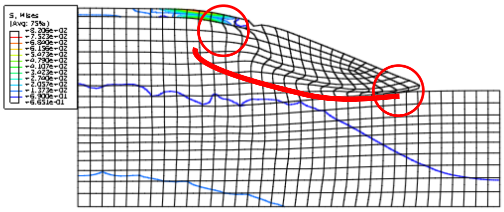

Figure 13 Cloud picture after deformation(line mode, Ft=1.75)

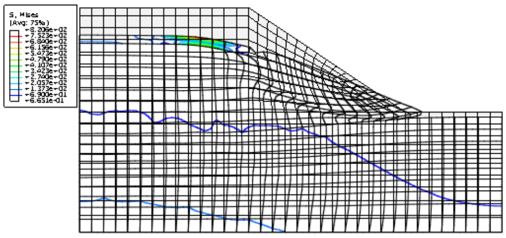

Figure 14 Comparison chart before and after deformation (line mode, Ft =1.75)

What can be seen from the figures, only under gravity loads, the development of the plastic zone which can reform the change of the slope appears at the catadrome side of the silty clay and the boundary of muddy clay.As the increasing of the reduction factor, the plastic zone in the backfill soil area expansion, Finally craze from the toe to the top of the slope. The answer to the question that appearing this kind of plastic zone development is that Backfill soil porosity is larger, has obvious shrinkage,so it`s easy to destroy.Plastic zone through model calculations (see Figure 13),and the plastic strain and displacement are showing rapid growth trend, indicating that slope instability has reached a critical state. At this time,the reduction factor given by abaqus is 1.75,so the reduction coefficient is 1.75,In other words, the minimum stability coefficient is 1.75.

5. Conclusion

In this article,combing the powerful features of the abaqus software which can solve the nonlinear problems and strength reduction finite element method.The development of generalized plastic strain and the plastic zone penetration and mutation of generalized plastic strain and displacement can be thinked as the evaluation to evaluate the unstability of slope.Using the excellent feature of abaqus which called dynamic graphical display technology to describe instability of slope vividly.So it can be used to determine the real stress state of slop,and can be thinked as a reliable basis to evaluate the stability of slope accurately. In the process of problem analysis, we can get conclusion as follows:

Cohesive force and angle of internal friction of soil are the most significant factors influence the stability of slope.Thus, the method that improving the soil strength ways can be used to improve the stability of slope,modulus of elasticity and poisson's ratio on the stability of the slope is smaller.

The density of grid computing in abaqus finite element calculation will affect the results, within a certain range,the more dense grid we make, the higher accuracy can we get.If the density of grid is too large,to improve the calculation accuracy it also can aggravate the computing burden,and it can make computing time becomes longer.

References

[1] Sun Shufang,CheYingHui. ABAQU application in slope stability analysis [J].Journal of shanxi building, 2008:313-314.

[2] Ying-ren zheng, shang-yi zhao,Zhang Luyu.To analyze the slope stability by finite element strength subtraction[J].Journal of China engineering science,2002,4(1):57-61,78.

[3] Zhang Luyu,ying-ren zheng,shang-yi zhao,etc. The finite element strength reduction factor method to calculate the accuracy of the slope stability safety factor research[J].Journal of water conservancy,2003(1):21-27.

[4] Xian-jing Kong.Strength reduction finite element method of slope excavation stability [J].Journal of geotechnical engineering,2001(4): 407-411.

[5] Zhong-chun Li, Xing-guo Cheng.Based on ABAQUS strength reduction fem slope stability analysis[J].Journal of disaster prevention and mitigation engineering,2006:89-94.

[6] Gong-yun liao,Huang xiao-ming.ABAQUS finite element software application in road engineering.Southeast university press.

[7] Shang-yi Zhao,ying-ren Zheng,Yu-fang Chang.The limit analysis of finite element method (fem) lecture: II finite element strength criterion of the slope unstability in the subtraction discuss [J]. rock and soil mechanics,2005, 26(2),332-336.