Abaqus Technology Brief

TB-06-SLOPE-1

Revised: April 2007

Analysis of Reinforced and Un-reinforced Soil Slopes using Abaqus

Summary

Assessing the strength of soil slopes and investigating the means for increasing their safety against failure are crucial in construction projects involving large soil masses. Slope stability analyses have traditionally been performed using a limit state approach. However, any presence of reinforcement or local heterogeneity necessitates the use of numerical techniques such as finite element analysis.

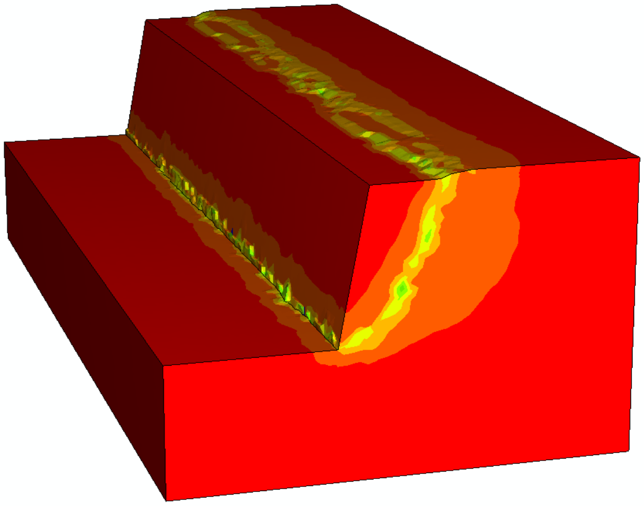
Abaqus/Standard can be used for modeling reinforced soils and can thus help geotechnical engineers in determining optimal reinforcement sizes and placement configurations. Abaqus/Explicit can be used for modeling the failure as well as the movement of dislodged soil masses.

Background

Analyses of soil slopes have traditionally been performed using a limit state approach. In this method, the soil is assumed to fail along a curved surface, which is usually a circular arc in two dimensional plane strain cases. The soil material at each point along this curved arc is assumed to have fully mobilized its maximum (or limit) strength in shear. The limit strength is derived from the maximum shear strength of the soil corresponding to the particular pressure value at that point, which is computed based on the point's vertical position. The maximum values of the resultant forces that can be generated at the limit state to resist the dislodged soil mass against sliding due to gravity are then used to assess the likelihood of failure along this chosen trial failure surface. The likelihood of failure is then expressed as a factor of safety (FOS), which is specific to the particular chosen trial failure surface. Several such likely failure surfaces are considered, and the one that produces the least FOS is chosen as the most likely failure surface.

Although this limit state method has already been computerized and results can now be obtained numerically for specified slope angles, soil densities, etc., the assumption of limit equilibrium and that of a contiguous failure surface curtails the general applicability of this method in the detailed modeling of reinforced soil slopes, nailed slopes, excavations with tied sheet piles, tunnels with tied linings, etc.

An alternative approach for assessing the safety and the likelihood of failure of slopes is to use the finite element method. In this approach the actual stress-strain characteristics of the soil material are directly taken into account and the precise distributions of stresses and strains within the soil are computed at the actual working load levels. Regions of soil that fail due to plastic shear strains can then be identified. Even though some particular regions of the soil mass may undergo plastic deformations at the working load levels, the soil mass as a whole may still be stable for the specified material properties, geometry, loading, and boundary conditions. Measures of FOS hence cannot be directly obtained for stable in-situ soil slopes using finite element analyses.



Key Abaqus Features and Benefits

- Supports a variety of constitutive models for soils, including Mohr-Coulomb, Drucker-Prager, Cam-Clay, and Cap plasticity models
- Can model contact and frictional interaction between soil and structures
- Embedded elements can be used to model reinforcement structures such as nails and ties
- Ability to model transient distribution of pore fluid pressure in soils in Abaqus/Standard
- Ability to model consolidation, or the deformation in soils associated with the redistribution of pore water and pressure, in Abaqus/Standard
- In Abaqus/Explicit, ability to model the damage and removal of failed elements, as well as the movement of dislodged regions of soil relative to the intact regions

In order to obtain some measures of the likelihood of failure akin to the FOS in the limit state approach, engineers have proposed several modifications to the finite element approach, such as reducing the material strength values or increasing the gravity load in order to obtain a stress state closer to the limit state. However, the appropriateness of designating safety factors that are obtained from such methods has been questioned. Additionally, in the finite element codes that employ modified material properties or loading conditions, the inability of the implicit solution method to obtain convergence in a selected number of iterations is often chosen as the basis for determining failure. This choice is questionable, as any instability in localized regions of a heterogeneous reinforced soil mass may also lead to convergence difficulties in the implicit finite element analysis, and such convergence difficulties may not therefore be indicative of a complete failure of the model as in the limit state approach.

Finite element analysis of models wherein the actual material data, loading, and boundary conditions are used can nonetheless provide a wealth of data on the state of stress and strain within the soil mass. Such detailed knowledge of the stress-strain regime can be very useful to geotechnical engineers for assessing the behavior of heterogeneous soil masses, reinforced soils, nailed slopes, etc. Additionally, information on the proximity of particular regions in the soil to the onset of yielding can help in assessing the possibility of local failure in reinforced and inhomogeneous soils.

Finite Element Analysis Approach

In this Technology Brief, a case study on the analysis of a pre-stressed nailed soil slope using Abaqus/Standard is presented. An example demonstrating the use of Abaqus/Explicit in the modeling of failure of slopes subjected to gravity and an overburden pressure is also presented. The latter example involves the use of material damage models and the inclusion of contact interactions between the dislodged and intact soil regions. In contrast to the implicit solution process of Abaqus/Standard, which has advantages in predicting the onset of yielding or localized material failure, the explicit solution process of Abaqus/Explicit is robust in modeling post failure deformations and large displacements of dislodged soil masses.

Reinforced soil slope model

In this model a soil slope inclined at 55 degrees to the horizontal and rising to a vertical height of 20 m is considered. At the height of 20m, the inclination reduces to 10 degrees. The problem is defined as a two dimensional plane strain analysis. The slope is modeled as having been constructed by excavating from a horizontal ground level as shown in Figure 1. The soil in the regions colored yellow and green is removed in sequential stages, which exposes the soil contained in the orange colored region.

For this analysis we have assumed that the soil is dry. The soil material has a density of 1900 kg/m^3, and has an elastic modulus of 100 MPa. The soil behaves as a linear Drucker-Prager plastic material, with a friction angle of 49.8 degrees and a cohesion value of 12.7 kPa. The geometry of the soil slope and the plastic material properties are based on Ref. 1, and for these material properties the soil slope is just stable at the full gravity load level.

Reinforcements in the form of pre-stressed nails are then included in a series of models in order to assess their effects on the stress-strain distribution within the soil mass. The nails have a density of 7800 kg/m^3, and an elastic modulus of 200 GPa. These nails are considered as embedded, that is, fully bonded to the surrounding soil. The nails have a length of 20 m. As depicted in Figure 2, a series of models is considered: unreinforced, and reinforced with a single nail, three nails, and seven nails, placed starting from the lowest point of the slope and with a vertical separation of 2.5 m. The nails are inclined at 30 degrees from the horizontal. The nails terminate at nail heads tied to the sloped region of the soil.

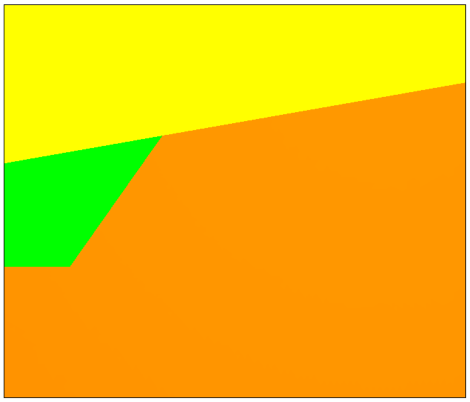


Figure 1: The soil slope (orange colored region) is obtained by excavating the yellow and green regions in sequential stages

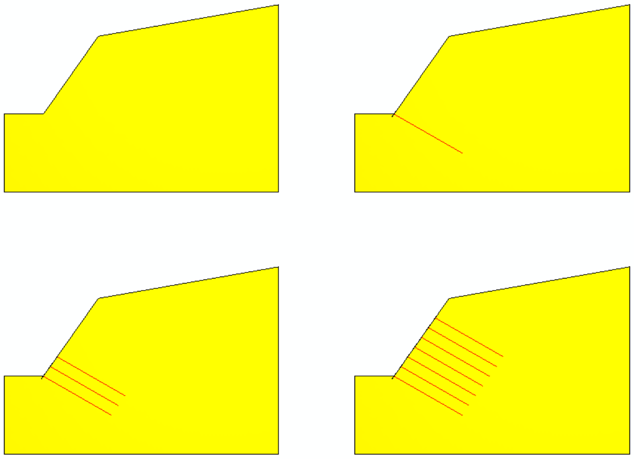


Figure 2: Nail configurations used in the reinforced soil slope model. The top left configuration has no reinforcement. The top right, bottom left, and bottom right figures show the one, three, and seven nail cases, respectively.

Overburden pressure model

An un-reinforced soil slope inclined at 79 degrees and with vertical height of 25 m is considered. The horizontal distance between the top and the bottom of the slope is 5 m. The soil is considered to be dry, and the soil density is 1800 kg/m3. A linear Drucker-Prager plasticity model is employed with friction and dilation angles of 30 degrees. Material cohesion is 100 kPa. Damage initiates in the soil at an equivalent fracture strain of 0.01 at a stress triaxiality of 0.01. Damage is considered to evolve through a displacement-based damage evolution criterion.

The model is subjected to gravitational forces. Additionally, the top portion of the sloped soil mass is loaded by an overburden pressure load of 1 MPa distributed over a rectangular patch as shown in Figure 3.

Analysis

The reinforced soil analysis is performed using Abaqus/Standard through a series of steps. In the first step geostatic equilibrium is obtained by applying the gravity load. In the second step the region overlaying the top of the slope (the yellow region in Figure 1) is removed so that a ground slope of 10 degrees is obtained. In the third step a second excavation is performed and the region depicted in green in Figure 1 is removed. The 55-degree slope is then exposed.

The reinforcing nails in a nailed soil slope contribute to soil stability only after the surrounding soil has failed and appreciably displaced, which results in the generation of tensile forces in the nails. In order for the nails to significantly contribute towards resisting the failure of the soil without any associated appreciable displacement in the soil, the nails need to be pre-stressed. In these analyses a tensile pre-stress is applied to the reinforcing nails.

As the soil material properties are not temperature dependent, the pre-stress is induced in the nails through the use of thermal contractions. A maximum tensile force of 4 MN is assumed in each nail when the displacements at their ends are restricted.

Abaqus/Explicit has been used to analyze the model involving the failure of a slope due to gravity and overburden pressure. This analysis is run in two steps. In the first step the gravity load is applied through a smooth step amplitude definition. The overburden pressure is applied in the second step. As the pressure load is applied, the soil material in the interior of the slope domain fails in shear, and is considered to be damaged. The damaged elements are automatically deleted; a contact interaction zone is newly created between the dislodged soil and the intact soil regions, and the analysis progresses further.

Results

Figures 4, 5, and 6 show the deformed configurations during the excavation of the slope analyzed using Abaqus/Standard. In Figure 4, the model is shown at the end of the first step. Figure 5 shows the deformed configuration after the second step, that is, after the removal of the soil in the yellow colored region. Figure 6 shows the final deformed configuration of the soil mass, that is, after the removal of the soil in the green colored region.

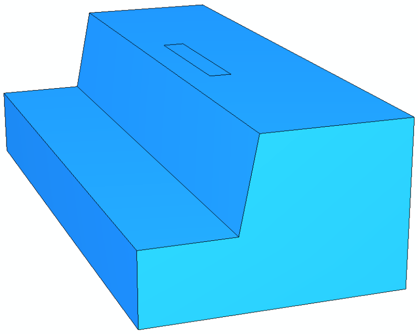


Figure 3: Abaqus/Explicit slope failure analysis model

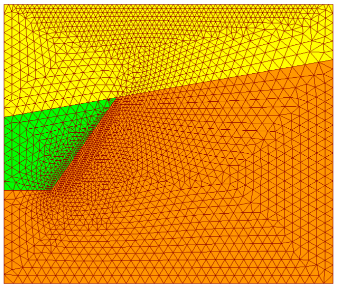


Figure 4: The finite element mesh at the end of the geostatic step and showing the three soil regions

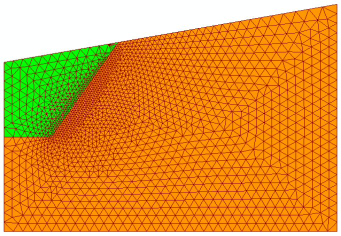


Figure 5: Deformed soil mesh after the first excavation

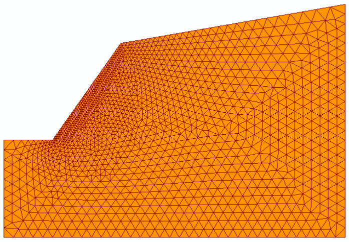


Figure 6: Deformed soil mesh after the second excavation, exposing the 55-degree slope

As the overlaying soil is removed, the underlying soil displaces vertically upwards due to the removal of the associated weight of the overlay. In Figure 7, the displacements at the end of the third step, relative to those at the end of the second step, are shown in a symbol plot. The arrows indicate the magnitude and direction of the displacements. The vertical displacements at the leftmost region of the lower flat portion of the soil are larger than those along the inclined region due to a larger reduction in the overburden weight.

Slope failures can occur when contiguous regions of the soil mass yield simultaneously in shear. The equivalent plastic strain measure available as an output variable (PEEQ) in Abaqus can be used to assess the accumulation of plastic strain in the soil region. Figures 8, 9, 10, and 11 show contours of the equivalent plastic strain for the un-reinforced and, one, three, and seven nail cases, respectively. It is seen that as the number of nails increases, the size of the region over which the equivalent plastic strain is large decreases.

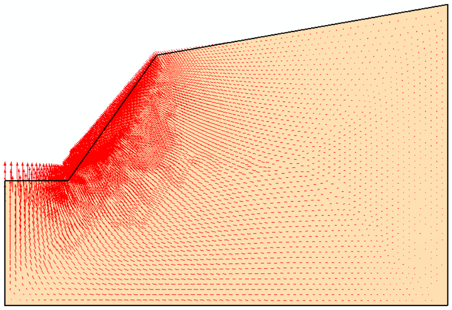


Figure 7: Deformed soil mesh after the second excavation

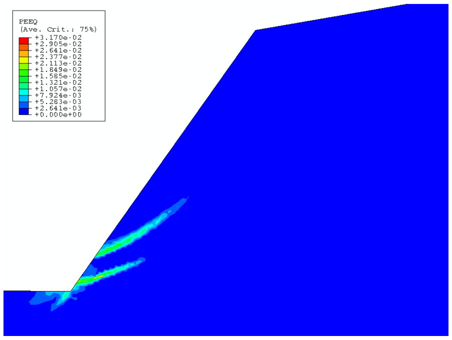


Figure 8: Equivalent plastic strain for the un-reinforced slope

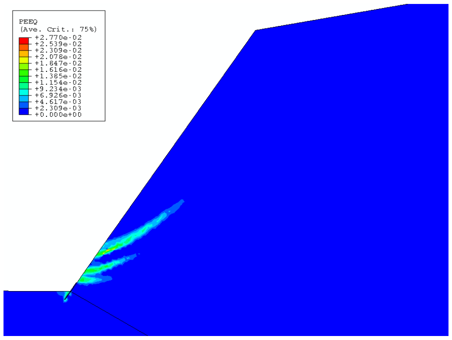


Figure 9: Equivalent plastic strain for the one nail case

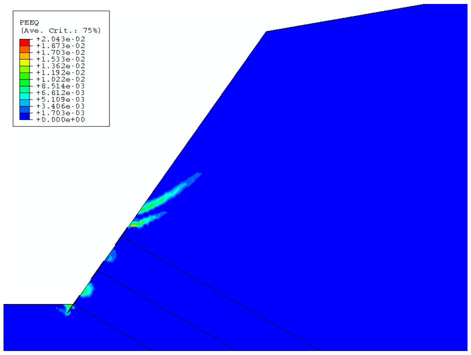


Figure 10: Equivalent plastic strain for the three nail case

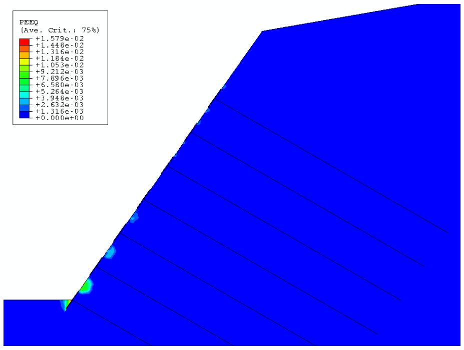


Figure 11: Equivalent plastic strain for the seven nail case

For the un-reinforced case, continuous bands of high equivalent plastic strain extend to about half the slope height. Such large bands are expected, as the slope configuration is just stable. However, with increasing number of reinforcing nails the extent of these bands reduces, and for the case with seven nails no such contiguous bands are seen, indicating a much higher degree of stability.

The propensity for instability at a particular point in a soil mass can also be assessed by examining the proximity of the deviatoric or shear stress values at that point to the maximum shear strength that can be generated at that point, which in turn depends on the pressure existing at that point. In order to express this propensity quantitatively, we consider the ratio of the Mises stress at that point to the maximum Mises stress that can be generated at the existing pressure. A Python script has been written to obtain values of such ratios, which we term as the DP Yield Factors at the soil material points. The DP Yield Factor will be 1 when the material point undergoes yielding, and is zero when the Mises stress is zero. Figures 12, 13, 14, and 15 depict contour plots of the DP Yield Factor for the un-reinforced and, one, three, and seven nail cases, respectively, at the end of the analyses.

When the soil mass is un-reinforced, the DP Yield Factor is close to 1 over a larger portion of the interior region. However, in the presence of an increasing number of reinforcing nails the extent of the interior region experiencing a high DP Yield Factor gets reduced, indicating that the soil is becoming more and more stable as the number of reinforcing nails increases. The free surfaces of the soil however show a high DP Yield Factor because of the presence of very low pressure stress values. However, this does not indicate a propensity for instability as the equivalent plastic strain values are very low at those locations.

Figure 16 shows a snapshot of the deformed shape of the model analyzed using Abaqus/Explicit at an intermediate post-failure instant. In contrast to the analysis using Abaqus/Standard, Abaqus/Explicit is able to simulate the movement of the dislodged soil mass after the slope has failed.

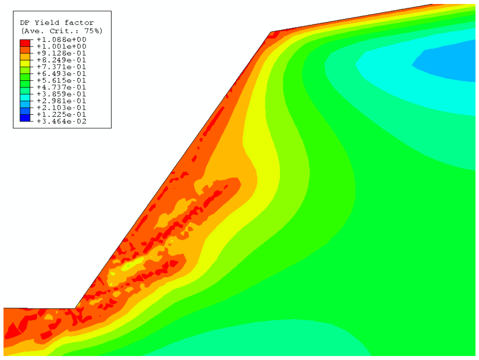


Figure 12: DP Yield Factor for the un-reinforced case

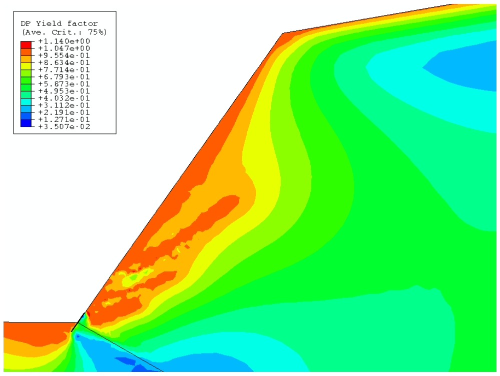


Figure 13: DP Yield Factor for the one nail case

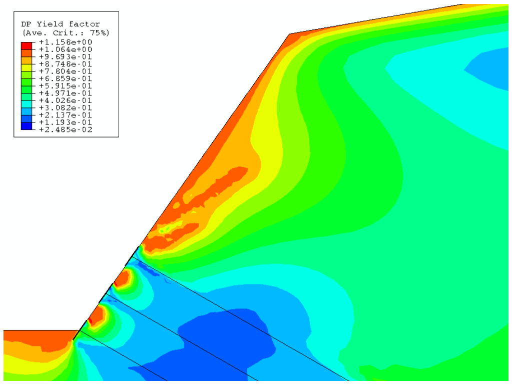


Figure 14: DP Yield Factor for the three nail case

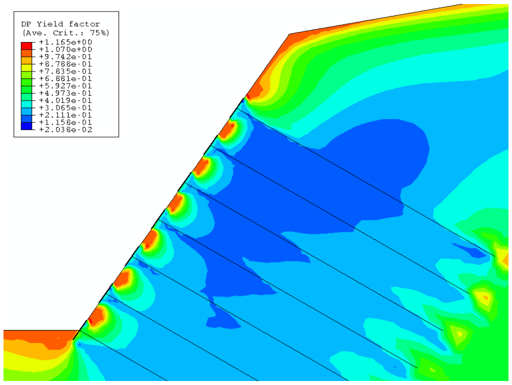


Figure 15: DP Yield Factor for the seven nail case

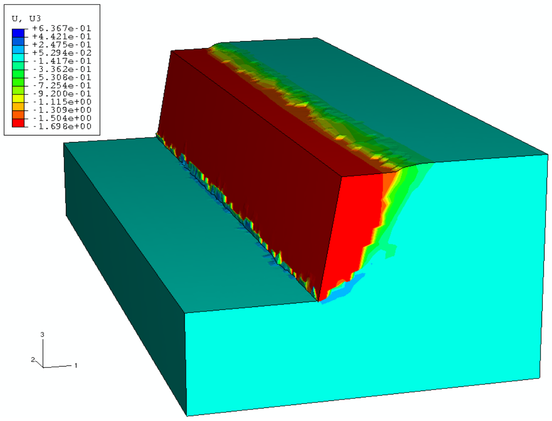


Figure 16: Vertical displacement at an intermediate post-failure instant for the model analyzed using Abaqus/Explicit

Conclusion

The available capabilities of Abaqus/Standard, including plasticity material models, contact, and constraint modeling, can be used for assessing the stress-strain distribution in reinforced soils.

Additionally, the scripting interface in Abaqus can be used for post processing the results and appropriate output variables helpful in gauging the likelihood of soil slope instability can be obtained.

Knowledge of the plastic strain values in different regions in the soil and additional information on the proximity of the existing deviatoric or shear stress to the maximum shear strength that can be generated in those regions can be helpful in improving designs for reinforced soil slopes. The capabilities of Abaqus/Explicit in the modeling of damage in materials and in modeling contact between newly created surfaces in the interior regions can be used to investigate the failure of slopes and the movement of dislodged soil masses.

References

1. Shiu, Y.K. and Chang, G.W.K., *Numerical Study on Soil Nail Heads and Nail Inclinations*, Proceedings of the Eleventh International Conference on Computer Methods and Advances in Geomechanics (IACMAG), Torino, Italy, 19-24 June, 2005, vol. 2, pp 373-380.

Abaqus References

For additional information on the Abaqus capabilities referred to in this brief please see the following Abaqus 6.11 documentation references:

- Analysis User's Manual
    - "Embedded elements," Section 33.4.1
    - " Extended Drucker-Prager models," Section 22.3.1
- Example Problems Manual
    - "Rigid projectile impacting eroding plate," Section 2.1.3
