CONTENTS

Lecture 1
Introduction

Introduction ...................................................... L1.2
Classical and Modern Design Approaches .................. L1.3
Some Cases for Numerical (Finite Element) Analysis ...... L1.8
Experimental Testing and Numerical Analysis .......... L1.11
Requirements for Realistic Constitutive Theories ........ L1.13

Lecture 2
Physical Testing

Physical Testing ................................................. L2.2
Basic Experimental Observations ............................ L2.4
Testing Requirements and Calibration of Constitutive Models . L2.11

Lecture 3
Constitutive Models

Stress Invariants and Spaces .................................. L3.2
Overview of Constitutive Models ............................ L3.10
Elasticity ...................................................... L3.12
Mohr-Coulomb Model .......................................... L3.15
  Description .................................................. L3.17
  Usage and Calibration ...................................... L3.24
Modified Drucker-Prager Models ............................ L3.28
  Linear Drucker-Prager Model .............................. L3.31
  Hyperbolic Model ......................................... L3.35
  Exponent Model .......................................... L3.38
Flow in the Hyperbolic and Exponent Models
Usage
Matching Experimental Data
  Linear Drucker-Prager Model
  Hyperbolic Model
  Exponent Model
Matching Mohr-Coulomb Parameters
  Matching Plane Strain Response
  Matching Triaxial Test Response
Coupled Creep and Drucker-Prager Plasticity
  Basic Assumptions
  Creep Laws
    Time Hardening Creep Law
    Strain Hardening Creep Law
    Singh-Mitchell Creep Law
  Creep Flow Potential
Usage
Modified Cam-Clay Model
  Description
  Usage and Calibration
  Example
Modified Cap Model
  Description
  Usage and Calibration
  Example
Coupled Creep and Cap Plasticity
  Basic Assumptions
    Cohesion Creep
    Consolidation Creep
  Creep Laws
  Creep Flow Potentials
Usage
Jointed Material Model
Description ................................................................. L3.130
Usage and Calibration .................................................. L3.137
Example ................................................................. L3.142
Numerical Implementation ............................................. L3.148

Lecture 4
Analysis of Porous Media

Basic Assumptions and Effective Stress ......................... L4.3
Stress Equilibrium and Flow Continuity ....................... L4.8
Types of Analyses and Usage ........................................ L4.17
Examples ............................................................... L4.29
  Fully Saturated Example ...................................... L4.29
  Partially Saturated Example ................................. L4.40

Lecture 5
Modeling Aspects

Element Technology ................................................. L5.2
Infinite Domains ...................................................... L5.4
Pore Fluid Surface Interactions ................................ L5.26
Geostatic States of Stress .......................................... L5.28
Element Addition and Removal ................................... L5.34

Lecture 6
Example Problems

Dry Problems .......................................................... L6.3
  Limit Analysis of Foundation ............................... L6.3
  Slope Stability Problem .................................... L6.20
  A Dynamic Analysis .......................................... L6.33
Saturated Problems ................................................... L6.44
  Consolidation Problem (Transient) ....................... L6.44
### Appendix A

#### Stress Equilibrium and Fluid Continuity Equations

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fully Saturated Fluid Flow</td>
<td>A.2</td>
</tr>
<tr>
<td>Special Cases</td>
<td>A.5</td>
</tr>
<tr>
<td>Partially Saturated Fluid Flow</td>
<td>A.11</td>
</tr>
<tr>
<td>Special Cases</td>
<td>A.15</td>
</tr>
</tbody>
</table>

### Appendix B

#### Bibliography of Geotechnical Example Problems
Analysis of Geotechnical Problems with ABAQUS

ABAQUS, INC.
The information in this document is subject to change without notice and should not be construed as a commitment by ABAQUS, Inc.

ABAQUS, Inc., assumes no responsibility for any errors that may appear in this document.

The software described in this document is furnished under license and may be used or copied only in accordance with the terms of such license.

No part of this document may be reproduced in any form or distributed in any way without prior written agreement with ABAQUS, Inc.

Copyright © ABAQUS, Inc., 2003.

Printed in U.S.A.
All Rights Reserved.

ABAQUS is a registered trademark of ABAQUS, Inc.

The following are trademarks of ABAQUS, Inc.: ABAQUS/ADAMS; ABAQUS/Aqua; ABAQUS/CAE; ABAQUS/C-MOLD; ABAQUS/Design; ABAQUS/Explicit; ABAQUS/MOLDFLOW; ABAQUS/Standard; ABAQUS/Viewer; and the ABAQUS, Inc., logo.

All other brand or product names are trademarks or registered trademarks of their respective companies or organizations.

Revision Status
Lecture 1 2/03 Minor Changes
Lecture 2 2/03 Minor Changes
Lecture 3 2/03 Minor Changes
Lecture 4 2/03 Minor Changes
Lecture 5 2/03 Minor Changes
Lecture 6 2/03 Minor Changes
Appendix A 2/03 Minor Changes
Appendix B 2/03 Minor Changes
Course Schedule

Analysis of Geotechnical Problems with ABAQUS

Day 1
• Lecture 1: Introduction
• Lecture 2: Physical Testing
• Lecture 3: Constitutive Models
• Lecture 4: Analysis of Porous Media

Day 2
• Lecture 5: Modeling Aspects
• Lecture 6: Example Problems
• Appendix A: Stress Equilibrium and Fluid Continuity Equations
• Appendix B: Bibliography of Geotechnical Example Problems
Lecture 1
Introduction

Overview

• Introduction
• Classical and Modern Design Approaches
• Some Cases for Numerical (Finite Element) Analysis
• Experimental Testing and Numerical Analysis
• Requirements for Realistic Constitutive Theories
Introduction

In this lecture we discuss the philosophy on which the usage of numerical (finite element) analysis for geotechnical problems is based.

Lecture 2 deals with experimental testing and how it relates to the calibration of constitutive models for geotechnical materials.

The different ABAQUS constitutive models applicable to geotechnical materials are presented in Lecture 3. Their usage, calibration, implementation, and limitations are discussed.

In Lecture 4 we outline the treatment of porous media in ABAQUS and discuss the coupling between fluid flow and stress/deformation.

Several modeling issues relating to geotechnical situations are discussed in Lecture 5.

In Lecture 6 typical problems are used as illustrative examples.
In the classical approach two basic types of calculations are done: failure estimates and deformation estimates.

Failure estimates are based on rigid perfectly plastic stress-strain assumptions:

Examples:

\[ \tau \]
\[ \gamma \]

constant shear strength at failure

Zero elastic strains before failure

Slope stability
Examples (cont.):

The result is a factor of safety, which is evaluated based on experience (design code).
Deformation estimates assume linear elastic behavior with average elastic properties:

\[
\sigma = \frac{E}{\nu} \cdot \varepsilon
\]

Foundation settlement example:

\[
w = p \cdot b \left( \frac{1 - \nu^2}{E} \right) \cdot f
\]

- \( p \) is bearing pressure
- \( b \) is width of foundation
- \( E, \nu \) are average elastic properties
- \( f \) is shape factor (based on small scale tests)
In the modern approach, failure and deformation characteristics are obtained from the same analysis. The analysis requires a complete constitutive model and the numerical solution of a boundary value problem.
Numerical (finite element) analysis can handle arbitrary geometries.
Some Cases for Numerical (Finite Element) Analysis

Cases when self weight of soil plays an important role, such as slope stability:

Classical limit failure calculations can predict ultimate stability of a slope accurately because collapse stresses on the failure plane are proportional to the weight of the soil and independent of the detailed soil behavior. Modern numerical analysis is necessary for calculation of deformations.
Cases when detailed soil behavior plays an important role, such as building of earth dam and subsequent filling of reservoir:

Local soil failure (stress or wetting driven) may trigger overall collapse or hamper functionality of structure. The sequence of events (construction, filling of reservoir, and long term consolidation) must be considered using numerical analysis.
Some Cases for Numerical (Finite Element) Analysis

Cases when the initial state of stress of the soil or rock mass is important, for example, tunneling:

\[ \sigma_v = \rho gh \]
\[ \sigma_h = K_o \sigma_v \]

The virgin state of stress caused by the weight of the soil and tectonic effects must be taken into account at the start of the analysis. The stability of the excavation depends on the virgin stress state as well as the sequence of the excavation process. It is possible to control the stability of the excavation by designing the excavation sequence (and perhaps using aids such as liners and rock bolts).
Experimental Testing and Numerical Analysis

- Laboratory testing
- Constitutive model
- Calibration
- Finite element model
- Small or large scale testing
- Comparison of model predictions with test results
- Design prototype
The measurements required in the simple laboratory tests depend on the proposed constitutive model. The constitutive model must be proposed based on simple experimental observations. Laboratory testing and constitutive model development are closely tied.

The constitutive model must first be chosen qualitatively: it is important to capture the major features of material behavior while minor features may be ignored in the model. Calibration (or quantitative choice) of the model parameters follows. Calibration should not be attempted beyond available (and repeatable) experimental results.

The finite element model must capture important features of the physical situation, without irrelevant detail. Use of an adequate constitutive model is critical although simplifications are often justifiable.

Small or large scale testing usually requires some knowledge of the physical behavior being modeled. Details of the physical tests and finite element models must be compatible for meaningful comparisons.

Ultimately, design requires engineering judgment and a good deal of experience. Testing and numerical simulation are only useful tools to aid the design.
Requirements for Realistic Constitutive Theories

Realistic constitutive models should help us to better understand the mechanical behavior of the material. Their development must be based on the understanding of the micromechanical behavior of the material, translated to a macro model simple enough to use in numerical calculations, thus creating a tool for rational design.

Realistic constitutive models must be general, in that they must be capable of representing material behavior in any relevant spatial situation (one-dimensional, plane strain, axial symmetry, and full three-dimensional analyses).

Realistic models must be based on experimental data that are relatively easy to obtain. They must then be able to extrapolate to conditions that cannot be reproduced with laboratory testing equipment.
Lecture 2
Physical Testing

Overview

• Physical Testing
• Basic Experimental Observations
• Testing Requirements and Calibration of Constitutive Models
Physical Testing

Geotechnical materials are generally voided and, thus, sensitive to volume changes. These volume changes are closely tied to the magnitude of the hydrostatic pressure stress, so it is important to test the materials over the range of hydrostatic pressure of interest.

Most laboratory testing facilities are capable of performing standard tests:

- Hydrostatic (or isotropic) compression tests
- Oedometer (or uniaxial strain) tests
- Triaxial compression and extension tests
- Uniaxial compression tests (a special case of triaxial compression)
- Shear tests
A practical constitutive model should require for calibration only information generated by these standard tests.

More sophisticated laboratory tests can be performed only at a very limited number of testing facilities. Truly triaxial tests require cubical devices that are very expensive and not easy to operate. Any kind of direct tensile test is difficult to perform since it requires a very “stiff” machine (the same applies to compression tests in brittle materials that soften significantly in compression). In most practical constitutive models, assumptions are made regarding the tensile as well as the true triaxial behavior of the material because the tests required for calibration are generally not available.

The diversity of geotechnical materials means that a wide range of behaviors is possible. What follows are some very general observations for frictional materials.
Isotropic compression test:

\( \sigma_1 = \sigma_2 = \sigma_3 \)

\( \varepsilon_1 = \varepsilon_2 = \varepsilon_3 \)
Oedometer (uniaxial strain) test:

\[ \sigma_1, \varepsilon_1 \]
\[ \sigma_3 = \sigma_2 \]
\[ \varepsilon_3 = 0 \]
\[ \varepsilon_2 = 0 \]
Basic Experimental Observations

Triaxial compression tests:

\[ \sigma_1, \varepsilon_1 \]
\[ \sigma_2, \varepsilon_2 \]

\[ \sigma_3 = \sigma_2 \]
\[ \varepsilon_3 = \varepsilon_2 \]

\[ (\sigma_1 - \sigma_3) \]
\[ (\sigma_1 + \sigma_3) \]

Failure envelope

\[ \varepsilon_{\text{vol}} \]

+ 

- 

Dense material

Loose material
The “critical state” concept:

Casagrande defined “critical state” as the state (for monotonic loading) at which continued shear deformation can occur without further change in effective stress and volume (void ratio) of the material.
Cyclic tests:

- Isotropic compression or oedometer tests

- Triaxial or uniaxial compression tests
Truly triaxial (cubical) tests:
Essential aspects of behavior of voided frictional materials:

+ Nonlinear stress-strain behavior
+ Irreversible deformations
+ Influence of hydrostatic pressure stress on “strength”
+ Influence of hydrostatic pressure on stress-strain behavior
+ Influence of intermediate principal stress on “strength”
+ Shear stressing-dilatancy coupling
+ Influence of hydrostatic pressure stress on volume changes
+ Hardening/softening related to volume changes
+ Stress path dependency
+ Effects of small stress reversals
  – Effects of large stress reversals (hysteresis)
  – Degradation of elastic stiffness after large stress reversals

+ Included in most constitutive models
  – Not included in most models
Basic requirements for laboratory testing of geotechnical materials:

- Specimens must be tested under the assumption that they represent an average material behavior (in the ground there will be some variation within the same soil or rock mass and the spatial scale of such variations may be large compared to laboratory test specimens).

- Tests must simulate the in situ conditions as closely as possible: range of stresses, drainage conditions and density of the material (for deep mining cases this may be difficult).

- All stresses and strains must be measured throughout the stress-strain response to allow complete characterization of the constitutive behavior.
Laboratory testing should be guided by a previously proposed constitutive model. Understanding of this model is necessary for correct interpretation of laboratory tests. Model parameters should be physical and measurable in practicable experiments.

The following is a list of laboratory tests and the corresponding components of the constitutive model that they help calibrate:

- Isotropic compression test or oedometer test. One test is required to calibrate hydrostatic behavior. One unloading is necessary to calibrate the elastic part of this behavior.

- Triaxial compression tests. Two (preferably more) tests are required to calibrate the shear behavior and its hydrostatic pressure dependence. One unloading (in each test) is necessary to calibrate the elastic part of this behavior.

- Triaxial extension tests. Two (preferably more) tests are required to calibrate the intermediate principal stress dependence of the shear behavior.

- Direct tension test. One test is required to calibrate tension behavior of cohesive materials (rocks or soils with cohesion).
- Truly triaxial (cubical) tests. Many tests are required to calibrate the behavior of the material when subjected to different stresses in all directions.

- Shear box tests and indirect tension (Brazilian) tests. These may be useful to calibrate the cohesive properties of the material.

- Multiple unloading-reloading cycles in any of above tests. This is necessary to calibrate effects of large stress reversals such as hysteresis and elastic degradation.
Lecture 3
Constitutive Models

Overview
• Stress Invariants and Spaces
• Overview of Constitutive Models
• Elasticity
• Mohr-Coulomb Model
• Modified Drucker-Prager Models
• Coupled Creep and Drucker-Prager Plasticity
• Modified Cam-Clay Model
• Modified Cap Model
• Coupled Creep and Cap Plasticity
• Jointed Material Model
• Numerical Implementation
Stress Invariants and Spaces

Stress in three dimensions: three direct and three shear components.

Symmetry of stress tensor:

\[ \sigma_{12} = \sigma_{21}, \quad \sigma_{13} = \sigma_{31}, \quad \sigma_{23} = \sigma_{32}. \]

ABAQUS convention: tensile stress is positive.
Principal stresses: stresses normal to planes in which shear stresses are zero.
In two dimensions (Mohr’s circle):

\[
\sigma_{1,2} = \frac{\sigma_{11} + \sigma_{22}}{2} \pm \sqrt{\left(\frac{\sigma_{11} - \sigma_{22}}{2}\right)^2 + \tau_{12}^2}
\]

\[
\tan2\theta = \frac{2\tau_{12}}{\sigma_{11} - \sigma_{22}}
\]
Stress decomposition: deviatoric plus hydrostatic:

\[ \sigma = S - pI. \]

ABAQUS invariants:

- pressure stress, \( p = \frac{1}{3} \; \text{trace}(\sigma) \),

- Mises equivalent stress, \( q = \sqrt[3]{\frac{3}{2} (S : S)} \),

- third invariant, \( r = \left( \frac{9}{2} S \cdot S : S \right)^{\frac{1}{3}} \).
For all models except Mohr-Coulomb, also define deviatoric stress measure:

\[ t = \frac{q}{2} \left[ 1 + \frac{1}{K} - \left( 1 - \frac{1}{K} \right) \left( \frac{r}{q} \right)^3 \right], \]

so that \( t = q/K \) in triaxial tension \((r = q)\) and \( t = q \) in triaxial compression \((r = -q)\). If \( K = 1 \), \( t = q \). \( K \) is typically between 0.8 and 1.0.

\( t = \text{constant is a “rounded” surface in the deviatoric plane—not close to Coulomb behavior.} \)
Useful planes:

- deviatoric (Π) plane
- meridional plane
- triaxial plane
- hydrostatic axis \( \sigma_1 = \sigma_2 = \sigma_3 \)
Meridional plane:

$q, t$

$p$
Deviatoric (or $\Pi$) plane:
Overview of Constitutive Models

Elasticity models:

– Linear, isotropic
– Porous, isotropic (nonlinear)
– Damaged, orthotropic (nonlinear; used in concrete, jointed material)

Plasticity models:

– Open surface, pressure independent (Mises)
– Open surface, pressure dependent (Drucker-Prager, Mohr-Coulomb)
– Closed surface (Cam-clay, Drucker-Prager with Cap)
– Multisurface (jointed material)
– Nested surfaces (bounding surface †)
Other inelastic models:

– Continuum damage theories †
– Endochronic theories †

None of the available models (with the possible exception of the jointed material model) is capable of accurately handling large stress reversals such as those occurring during cyclic loading or severe dynamic events.

†Not available in ABAQUS for geotechnical materials
Elasticity

Either linear elasticity or nonlinear, porous elasticity can be used with the plasticity models described in the following sections.

Classical linear isotropic elasticity is defined by Young’s modulus and Poisson’s ratio.

Porous elasticity is a nonlinear, isotropic, elasticity model in which the pressure stress varies as an exponential function of volumetric strain:

\[ p = -p_{t}^{el} + (p_0 + p_{t}^{el}) \exp \left[ \frac{1 + e_0}{\kappa} (1 - \exp(\varepsilon_{vol}^{el})) \right] \]

or

\[ \frac{\kappa}{(1 + e_0)} \ln \left( \frac{p_0 + p_t^{el}}{p + p_t^{el}} \right) = J^{el} - 1, \]

where \( J^{el} - 1 \) is the nominal volumetric strain.
Throughout these notes $J = \frac{dV}{dV^0}$ is the ratio of current volume to reference volume, so $\varepsilon_{vol} = \ln(J)$ and $J = \exp(\varepsilon_{vol})$, where $\varepsilon_{vol}$ is the logarithmic measure of volumetric strain.

This model allows a zero or nonzero elastic tensile stress limit, $p^el_t$. The deviatoric behavior is defined either by choosing a constant shear modulus, 

$$S = 2G e^{el},$$

thus making the deviatoric elastic stiffness independent of pressure stress, or choosing a constant Poisson’s ratio that makes the deviatoric stiffness dependent on the pressure stress,

$$dS = 2\hat{G} d e^{el},$$

where the instantaneous shear modulus, $\hat{G}$, is

$$\hat{G} = \frac{3(1 - 2\nu)(1 + e_0)}{2(1 + \nu)\kappa}(p + p^el_t)\exp(\varepsilon_{vol}^{el})$$

$\kappa$, $p^el_t$, $G$, $\nu$ are material parameters; $p_0$ is the initial value of hydrostatic pressure stress, and $e_0$ is the initial voids ratio.
\[ \varepsilon^{el}_{vol} \] has an arbitrary origin, defined so that \( p = p_0 \) at \( \varepsilon^{el}_{vol} = 0 \).
The Mohr-Coulomb plasticity model is intended for modeling granular materials such as soils under monotonic loading conditions and does not consider rate dependence.

The ABAQUS Mohr-Coulomb plasticity model has the following characteristics:

- There is a regime of purely linear elastic response, after which some of the material deformation is not recoverable and can, thus, be idealized as being plastic.
- The material is initially isotropic.
- The yield behavior depends on the hydrostatic pressure. One of the consequences of this is that the material becomes stronger as the confining pressure increases.
- The yield behavior may be influenced by the magnitude of the intermediate principal stress.
– The material may harden or soften isotropically.
– The inelastic behavior will generally be accompanied by some volume change: the flow rule may include inelastic dilation as well as inelastic shearing.
– The plastic flow potential is smooth and nonassociated.
– Temperature may affect the material properties.
– It does not consider rate-dependent material behavior.
Description

Linear isotropic elasticity must be used with the Mohr-Coulomb model. The Mohr-Coulomb yield function is written as

\[ F = R_{mc} q - p \tan \phi - c = 0, \]

where

\[ R_{mc}(\Theta, \phi) \] is a measure of the shape of the yield surface in the deviatoric plane,

\[ R_{mc} = \frac{1}{\sqrt{3} \cos \phi} \sin \left( \Theta + \frac{\pi}{3} \right) + \frac{1}{3} \cos \left( \Theta + \frac{\pi}{3} \right) \tan \phi, \]

\( \phi \) is the slope of the Mohr-Coulomb yield surface in the \( R_{mc} q - p \) stress plane, which is commonly referred to as the friction angle of the material, \( 0 \leq \phi < 90; \)
$c$ is the cohesion of the material; and

$\Theta$ is the deviatoric polar angle defined as

$$\cos(3\Theta) = \frac{r^3}{q^3}.$$ 

The Mohr-Coulomb model assumes that the hardening is defined in terms of the material’s cohesion, $c$. The cohesion can be defined as a function of plastic strain, temperature, or field variables.

The hardening is isotropic.
Mohr-Coulomb Model

Yield Surface in the Meridional Plane (a) and the Deviatoric Plane (b)
The flow potential, $G$, is chosen as a hyperbolic function in the meridional stress plane and the smooth elliptic function proposed by Menétrey and Willam (1995) in the deviatoric stress plane:

$$G = \sqrt{(\varepsilon c|_0 \tan \psi)^2 + (R_{mw} q)^2} - p \tan \psi.$$  

The initial cohesion of the material, $c|_0 = c(\bar{\varepsilon}^{pl} = 0.0)$; the dilation angle, $\psi$; and the meridional eccentricity, $\varepsilon$, control the shape of $G$ in the meridional plane.
\( \varepsilon \) defines the rate at which \( G \) approaches the asymptote (the flow potential tends to a straight line in the meridional stress plane as the meridional eccentricity tends to zero).

Mohr-Coulomb Flow Potential in the Meridional Plane
Mohr-Coulomb Model

\[ R_{mw}(\Theta, e, \phi) \text{ controls the shape of } G \text{ in the deviatoric plane:} \]

\[
R_{mw} = \frac{4(1 - e^2)(\cos \Theta)^2 + (2e - 1)^2}{2(1 - e^2)\cos \Theta + (2e - 1)\sqrt{4(1 - e^2)(\cos \Theta)^2 + 5e^2 - 4e}} R_{mc} \left( \frac{\pi}{3}, \phi \right)
\]

The deviatoric eccentricity, \( e \), describes the “out-of-roundedness” of the deviatoric section in terms of the ratio between the shear stress along the extension meridian (\( \Theta = 0 \)) and the shear stress along the compression meridian (\( \Theta = \pi/3 \)).

The default value of the deviatoric eccentricity is calculated by

\[ e = \frac{3 - \sin \phi}{3 + \sin \phi} \]

and allows the ABAQUS Mohr-Coulomb model to match the behavior of the classical Mohr-Coulomb model in triaxial compression and tension.
The deviatoric eccentricity may have the following range: $\frac{1}{2} < e \leq 1.0$.

If the user defines $e$ directly, ABAQUS matches the classical Mohr-Coulomb model only in triaxial compression.

Plastic flow in the deviatoric plane is always nonassociated.
Usage and Calibration

The *ELASTIC, TYPE=ISOTROPIC option (linear, isotropic elasticity) must be used.

The *MOHR COULOMB option is used to define $\psi$, $\phi$, $e$, and $\varepsilon$.

- The ECCENTRICITY parameter is used to define $\varepsilon$. The default value is 0.1.

- The DEVIATORIC ECCENTRICITY parameter can be used to define $e$. The deviatoric eccentricity may have the following range: $\frac{1}{2} < e \leq 1.0$.

- If the user defines $e$ directly, ABAQUS matches the classical Mohr-Coulomb model only in triaxial compression.

The friction angle, $\phi$, and the dilation angle, $\psi$, can be functions of temperature and field variables.
The *MOHR-COULOMB option must always be accompanied by the *MOHR-COULOMB HARDENING material option, where the evolution of the cohesion, \( c \), of the material is defined. This can be given as a function of temperature and predefined field variables.

*EXPANSION can be used to introduce thermal volume change effects.

Plastic flow in the deviatoric plane is always nonassociated; therefore, the unsymmetric solver (*STEP, UNSYMM=YES) should be used when a material has Mohr-Coulomb plastic deformation.
Typically the Mohr-Coulomb model is calibrated using the critical stress states from several different triaxial tests. These critical stress states are plotted in the meridional plane to provide an estimate of the friction angle, $\phi$, and the cohesion, $c$, of the material.

– The dilation angle, $\psi$, is chosen so that the volume change during the plastic deformation matches that seen experimentally.
– If the material is going to harden under plastic deformation, one of the triaxial tests should be used to provide the hardening data.
Because the ABAQUS Mohr-Coulomb model uses a smooth plastic flow potential, it does not always provide the same plastic behavior as a classical (associated) Mohr-Coulomb model, which has a faceted flow potential.

– With the default value of deviatoric eccentricity, $e$, ABAQUS does match classical Mohr-Coulomb behavior under triaxial extension or compression.

– Benchmark Problem 1.14.5, *Finite deformation of an elastic-plastic granular material*, shows how to match the ABAQUS Mohr-Coulomb model to the classical Mohr-Coulomb model for plane strain deformation.
Modified Drucker-Prager Models

This set of models is intended to simulate material response under essentially monotonic loading, such as the limit load analysis of a soil foundation.

These models are the simplest available for simulating frictional materials.

The basic characteristics of this set of models are:

– There is a regime of purely elastic response, after which some of the material deformation is not recoverable and can, thus, be idealized as being plastic.

– The material is initially isotropic.

– The yield behavior depends on the hydrostatic pressure. One of the consequences of this is that the material becomes stronger as the confining pressure increases. The material may harden or soften isotropically. The models differ in the manner in which the hydrostatic pressure dependence is introduced.
– The inelastic behavior will generally be accompanied by some volume change: the flow rule may include inelastic dilation as well as inelastic shearing. Two different flow rules are offered.
– The yield behavior may be influenced by the magnitude of the intermediate principal stress.
– The material may be sensitive to the rate of straining.
– Temperature may affect the material properties.
Either linear elasticity or nonlinear porous elasticity, as described in *Elasticity* (p. L3.12), can be used with these models.

A choice of three different yield criteria is provided. The differences are based on the shape of the yield surface in the meridional plane: a linear form, a hyperbolic form, or a general exponent form.

The hyperbolic and exponent models are available only in ABAQUS/Standard.

In ABAQUS/Explicit only the linear model is available.

The choice of model to be used depends largely on the kind of material, on the experimental data available for calibration of the model parameters, and on the range of pressure stress values that the material is likely to see. Calibration is discussed later.
Linear Drucker-Prager Model

The yield surface of the linear model is written as

\[ F = t - p \tan \beta - d = 0. \]

The cohesion, \( d \), is related to the hardening input data as

\[ d = (1 - \frac{1}{3} \tan \beta) \sigma_c \quad \text{if hardening is defined by uniaxial compression, } \sigma_c; \]

\[ d = \left( \frac{1}{K} + \frac{1}{3} \tan \beta \right) \sigma_t \quad \text{if hardening is defined by uniaxial tension, } \sigma_t; \text{ and} \]

\[ d = d \quad \text{if hardening is defined by shear (cohesion), } d. \]

\( \beta \) (the friction angle) and \( K \) are material parameters. \( d, \sigma_c, \text{ or } \sigma_t \) is used as the (isotropic) hardening parameter, which is assumed to depend on the equivalent plastic strain.
The measure of deviatoric stress, $t$, allows matching of different stress values in tension and compression in the deviatoric plane, thus providing flexibility in fitting experimental results. However, as mentioned previously, the surface is too smooth to be a close approximation to the Mohr-Coulomb surface.

\[ t = \frac{1}{2} q \left[ 1 + \frac{1}{K} \left( 1 - \frac{1}{K} \right) \left( \frac{r}{q} \right)^n \right] \]

<table>
<thead>
<tr>
<th>Curve</th>
<th>K</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>1.0</td>
</tr>
<tr>
<td>b</td>
<td>0.8</td>
</tr>
</tbody>
</table>
We assume a (possibly) nonassociated flow rule, where the direction of the inelastic deformation vector is normal to a linear plastic potential, $G$:

$$d\varepsilon_{pl} = \frac{d\bar{\varepsilon}_{pl}}{c} \frac{\partial G}{\partial \sigma},$$

where $G = t - p \tan \psi$, $c$ is a constant that depends on the type of hardening data,

$$d\bar{\varepsilon}_{pl} = |d\varepsilon_{11}^{pl}| \text{ in uniaxial compression,}$$

$$d\bar{\varepsilon}_{pl} = d\varepsilon_{11}^{pl} \text{ in uniaxial tension, and}$$

$$d\bar{\varepsilon}_{pl} = \frac{d\gamma_{3}^{pl}}{\sqrt{3}} \text{ in pure shear.}$$

$\psi$ is the dilation angle in the $p-t$ plane. This flow rule definition precludes dilation angles $\psi > 71.5^\circ (\tan \psi > 3)$, which is not likely to be a limitation for real materials.
Flow is associated in the deviatoric plane but nonassociated in the $p-t$ plane if $\psi \neq \beta$. For $\psi = 0$, the material is nondilatational; and if $\psi = \beta$, the model is fully associated.
Hyperbolic Model

The hyperbolic yield criterion is a continuous combination of the maximum tensile stress condition of Rankine (tensile cut-off) and the linear Drucker-Prager condition at high confining stress. It is written as

\[
F = \sqrt{l_0^2 + q^2} - p\tan\beta - d' = 0 ,
\]

where \(d'\) is the hardening parameter that is related to the hardening input data as

\[
d' = \sqrt{l_0^2 + \sigma_c^2} - \frac{\sigma_c}{3}\tan\beta \quad \text{if hardening is defined by uniaxial compression, } \sigma_c;
\]

\[
d' = \sqrt{l_0^2 + \sigma_t^2} + \frac{\sigma_t}{3}\tan\beta \quad \text{if hardening is defined by uniaxial tension, } \sigma_t;
\]

\[
d' = \sqrt{l_0^2 + d^2} \quad \text{if hardening is defined by shear (cohesion), } d.
\]
\[ l_0 = d'|_0 - p_t|_0 \tan \beta \]
determines how quickly the hyperbola approaches its asymptote (see sketch).

\( p_t|_0 \) is the initial hydrostatic tension strength of the material, \( d'|_0 \) is the initial value of \( d' \), and \( \beta \) is the friction angle measured at high confining pressure.

The model treats \( \beta \) and \( l_0 \) as constants during hardening.
The yield surface is a von Mises circle in the deviatoric stress plane. (The $K$ parameter is not available for this model.)
Exponent Model

The general exponent form provides the most general yield criterion available in this class of models. The yield function is written as

\[ F = a q^b - p - p_t = 0 , \]

where \( a \) and \( b \) are material parameters independent of plastic deformation and \( p_t \) is the hardening parameter that represents the hydrostatic tension strength of the material and is related to the input data as

\[ p_t = a \sigma_c^b - \frac{\sigma_c}{3} \quad \text{if hardening is defined by uniaxial compression, } \sigma_c ; \]

\[ p_t = a \sigma_t^b + \frac{\sigma_t}{3} \quad \text{if hardening is defined by uniaxial tension, } \sigma_t ; \text{ and} \]

\[ p_t = a d^b \quad \text{if hardening is defined by shear (cohesion), } d . \]
The yield surface is a von Mises circle in the deviatoric stress plane. (The \( K \) parameter is not available for this model.)

The material parameters \( a \), \( b \), and \( p_i \) can be given directly; or, if triaxial test data at different levels of confining pressure are available, ABAQUS will determine the material parameters from the triaxial test data using a least squares fit.
Flow in the Hyperbolic and Exponent Models

Plastic flow in the hyperbolic and general exponent models is governed by the hyperbolic flow potential

\[ G = \sqrt{(\varepsilon \bar{\sigma}_0 \tan \psi)^2 + q^2 - p \tan \psi}, \]

where \( \psi \) is the dilation angle in the meridional plane at high confining pressure; \( \bar{\sigma}_0 \) is the initial yield stress; and \( \varepsilon \) is a parameter (referred to as the eccentricity) that defines the rate at which the function approaches its asymptote (the flow potential tends to a straight line as the eccentricity tends to zero).

This flow potential, which is continuous and smooth, ensures that the flow direction is always defined uniquely.

The function approaches the linear Drucker-Prager flow potential asymptotically at high confining pressure stress and intersects the hydrostatic pressure axis at 90°. It is, therefore, preferred as a flow potential for the Drucker-Prager models over the straight line potential, which has a vertex on the hydrostatic pressure axis.
The potential is the von Mises circle in the deviatoric stress plane.
Associated flow is obtained in the hyperbolic model if $\beta = \psi$ and

$$\varepsilon = \frac{l_0}{\bar{\sigma}_0 \tan \psi}.$$ 

In the general exponent model flow is always nonassociated in the meridional plane. The default flow potential eccentricity is $\varepsilon = 0.1$, which implies that the material has almost the same dilation angle over a wide range of confining pressure stress values.

Increasing the value of the eccentricity provides more curvature to the flow potential, implying that the dilation angle increases more rapidly as the confining pressure decreases. Values of the eccentricity less than the default value may lead to convergence problems if the material is subjected to low confining pressures because of the very tight curvature of the flow potential near its intersection with the $p$-axis.
Usage

The modified Drucker-Prager plasticity models in ABAQUS are invoked with the *DRUCKER PRAGER material option. The SHEAR CRITERION parameter is set to LINEAR, HYPERBOLIC, or EXPONENT to define the yield surface shape.

The *DRUCKER PRAGER option must always be accompanied by the *DRUCKER PRAGER HARDENING option. This option defines the evolution of the yield stress in uniaxial compression (TYPE=COMPRESSION), in uniaxial tension (TYPE=TENSION), or in pure shear (TYPE=SHEAR).

It is possible to make the yield function rate dependent by using the *RATE DEPENDENT option or by specifying the yield stress as a function of the plastic strain rate. A rate dependency is rarely used for geotechnical materials, but these same yield models are sometimes used for other materials such as polymers where it is important. The manner of introducing rate dependence is described in the User’s Manual.
The elasticity is defined with the *ELASTIC material option in the case of linear elasticity or with the *POROUS ELASTIC option if porous elasticity is chosen.

All of the material parameters can be entered as functions of temperature and field variables.

*EXPANSION can be used to introduce thermal volume change effects.

*INITIAL CONDITIONS, TYPE=RATIO is required to define the initial voids ratio (porosity) of the material if porous elasticity is used.

Analyses using a nonassociated flow version of the model may require the use of the UNSYMM=YES parameter on the *STEP option because of the resulting unsymmetric plasticity equations. If UNSYMM=YES is not used when the flow is nonassociated, ABAQUS may not find a converged solution.
Matching Experimental Data

At least two experiments are required to calibrate the simplest version of the Drucker-Prager plasticity model (linear model, rate independent, temperature independent, and yielding independent of the third stress invariant).

For geotechnical materials the most common experiments performed for this purpose are uniaxial compression (for cohesive materials) and triaxial compression or tension tests. However, other experiments can be used as alternatives: for example, shear tests for cohesive materials.

The uniaxial compression test involves compressing the sample between two rigid platens. The load and displacement in the direction of loading are recorded. The lateral displacements should also be recorded so that the correct volume changes can be calibrated.

Triaxial test data are required for a more accurate calibration.
Triaxial compression/tension experiments are performed using a standard triaxial machine where a fixed confining pressure is maintained while the differential stress is applied. Several tests covering the range of confining pressures of interest are usually performed. Again, the stress and strain in the direction of loading are recorded, together with the lateral strain, so that the correct volume changes can be calibrated.
In a triaxial compression test the specimen is confined by pressure and an additional compression stress is superposed in one direction. Thus, the principal stresses are all negative, with \( 0 \geq \sigma_1 = \sigma_2 \geq \sigma_3 \).

The stress invariant values in triaxial compression are

\[
p = -\frac{1}{3}(2\sigma_1 + \sigma_3), \quad q = \sigma_1 - \sigma_3, \quad r = -q, \quad t = q.
\]

The triaxial results can, thus, be plotted in the \( q-p \) plane.
The stress state corresponding to some user-chosen critical level (the stress at onset of inelastic behavior or the ultimate yield stress) provides one data point for calibrating the yield surface material parameters.

Additional data points are obtained from triaxial tests at different levels of confinement. These data points define the shape and position of the yield surface in the meridional plane.
Defining the shape and position of the yield surface is adequate to define the model if it is to be used as a failure surface.

To incorporate isotropic hardening, one of the stress-strains curves from the triaxial tests can be used to define the hardening behavior. The curve that represents hardening most accurately over a wide range of loading conditions should be selected.

Unloading measurements in these tests are useful to calibrate the elasticity, particularly in cases where the initial elastic region is not well defined.
Linear Drucker-Prager Model

Fitting the best straight line through the results provides the friction angle $\beta$. 

\[ \frac{h_t}{h_c} = K \]

loading path in triaxial compression

best fit to triaxial compression data

best fit to triaxial tension data

\[ p \]

\[ q \]
Triaxial tension test data are also needed to define $K$. Under triaxial tension the specimen is again confined by pressure, then the pressure in one direction is reduced. In this case the principal stresses are $0 \geq \sigma_1 \geq \sigma_2 = \sigma_3$. The stress invariants are now

$$p = -\frac{1}{3}(\sigma_1 + 2\sigma_3), \quad q = \sigma_1 - \sigma_3, \quad r = q, \quad t = \frac{q}{K}.$$ 

$K$ can, thus, be found by plotting these test results as $q$ versus $p$ and again fitting the best straight line. The ratio of values of $q$ for triaxial tension and compression at the same value of $p$ then gives $K$.

The dilation angle $\psi$ must be chosen such that a reasonable match of the volume changes during yielding is obtained. Generally, $0 \leq \psi \leq \beta$. 
Hyperbolic Model

Fitting the best straight line through the triaxial compression results at high confining pressures provides $\beta$ and $d'$ for the hyperbolic model. In addition, hydrostatic tension data, $p_t$, are required to complete the calibration.

\[ F = \sqrt{(d'|_0 - p_t|_0 \tan \beta)^2 + q^2} - p \tan \beta - d' = 0 \]
Exponent Model

ABAQUS provides a capability to determine the material parameters $a$, $b$, and $p_t$ required for the exponent model from triaxial data, which is done on the basis of a “best fit” of the triaxial test data at different levels of confining stress.
The data points obtained from triaxial tests are specified using the 
*TRIAxIAL TEST DATA option. The TEST DATA parameter is 
required on the *DRUCKER PRAGER option to use this feature. The 
*TRIAxIAL TEST DATA option must be used with the *DRUCKER 
PRAGER option.

The capability allows all three parameters to be calibrated, or, if some 
of the parameters are known, to calibrate only the unknown 
parameters.
Matching Mohr-Coulomb Parameters

Sometimes only the friction angle and cohesion values for the Mohr-Coulomb model are provided. We need to calculate values for the parameters of the linear Drucker-Prager model to provide a reasonable match to the Mohr-Coulomb parameters.
The Mohr-Coulomb model is based on plotting Mohr’s circle for stresses at failure in the plane of the maximum and minimum principal stresses. The failure line is the best straight line that touches these Mohr’s circles.
The Mohr-Coulomb model is, thus,

\[ s + \sigma_m \sin\phi - c\cos\phi = 0, \]

where

\[ s = \frac{1}{2}(\sigma_1 - \sigma_3) \]

is half of the difference between the maximum and minimum principal stresses (and is, therefore, the maximum shear stress) and

\[ \sigma_m = \frac{1}{2}(\sigma_1 + \sigma_3) \]

is the average of the maximum and minimum principal stresses.

The Coulomb friction angle, \( \phi \), is different from the angle \( \beta \) used in the \((p, q)\) plane in the linear Drucker-Prager model.
We see that the Mohr-Coulomb model assumes that failure is independent of the value of the intermediate principal stress. The Drucker-Prager model does not. The failure of typical granular geotechnical materials generally includes only small dependence on the intermediate principal stress, so the Mohr-Coulomb model is generally more realistic than the Drucker-Prager model.
Matching Plane Strain Response

Plane strain problems are often encountered in geotechnical analysis. Therefore, the constitutive model parameters are often matched to provide the same flow and failure response as a Mohr-Coulomb model in plane strain.

Since we wish to match only the behavior in one plane, we can assume $K = 1$. Using the plane strain constraint, we can derive the relationships

$$\sin \phi = \frac{\tan \beta \sqrt{3(9 - \tan^2 \psi)}}{9 - \tan \beta \tan \psi},$$

$$c \cos \phi = \frac{\sqrt{3(9 - \tan^2 \psi)}}{9 - \tan \beta \tan \psi} \cdot d.$$
For associated flow $\psi = \beta$, which gives

$$\tan \beta = \frac{\sqrt{3} \sin \phi}{\sqrt{1 + \frac{1}{3} \sin^2 \phi}} \quad \text{and} \quad \frac{d}{c} = \frac{\sqrt{3} \cos \phi}{\sqrt{1 + \frac{1}{3} \sin^2 \phi}}.$$ 

For nondilatant flow $\psi = 0$, which gives

$$\tan \beta = \sqrt{3} \sin \phi \quad \text{and} \quad \frac{d}{c} = \sqrt{3} \cos \phi.$$
The difference between assuming associated or nondilatant flow increases with the friction angle, but for typical friction angles the results are not very different, as shown below:

<table>
<thead>
<tr>
<th>Mohr-Coulomb friction angle, $\phi$</th>
<th>$\beta$</th>
<th>$d/c$</th>
<th>$\beta$</th>
<th>$d/c$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10°</td>
<td>16.7°</td>
<td>1.70</td>
<td>16.7°</td>
<td>1.70</td>
</tr>
<tr>
<td>20°</td>
<td>30.2°</td>
<td>1.60</td>
<td>30.6°</td>
<td>1.63</td>
</tr>
<tr>
<td>30°</td>
<td>39.8°</td>
<td>1.44</td>
<td>40.9°</td>
<td>1.50</td>
</tr>
<tr>
<td>40°</td>
<td>46.2°</td>
<td>1.24</td>
<td>48.1°</td>
<td>1.33</td>
</tr>
<tr>
<td>50°</td>
<td>50.5°</td>
<td>1.02</td>
<td>53.0°</td>
<td>1.11</td>
</tr>
</tbody>
</table>

The results obtained for a foundation problem using the different Drucker-Prager matches described here are given in **Dry Problems** (p. L6.3) of these notes.
Matching Triaxial Test Response

An alternative approach to matching Mohr-Coulomb and linear Drucker-Prager model parameters is to make the two models provide the same failure definition in triaxial compression and tension. This approach yields the following Drucker-Prager parameters:

\[
\tan \beta = \frac{6 \sin \phi}{3 - \sin \phi},
\]

\[
K = \frac{3 - \sin \phi}{3 + \sin \phi},
\]

\[
\sigma_c^0 = 2c \frac{\cos \phi}{1 - \sin \phi}.
\]
The value of $K$ in the Drucker-Prager model is restricted to $K \geq 0.778$ for the yield surface to remain convex. Rewriting the second equation as

$$\sin\phi = 3\left(\frac{1 - K}{1 + K}\right)$$

shows that this implies $\phi \leq 22^\circ$.

Many real materials have a larger Mohr-Coulomb friction angle than this value. In such cases one approach is to choose $K = 0.778$ and then to use the first equation to define $\beta$ and the third equation to define $\sigma_c^0$. This matches the models for triaxial compression only, while providing the closest approximation that the model can provide to failure being independent of the intermediate principal stress. If $\phi$ is significantly larger than $22^\circ$, this approach may provide a poor Drucker-Prager match of the Mohr-Coulomb parameters.
Coupled Creep and Drucker-Prager Plasticity

Geomaterials may creep under certain conditions. When the loading rate is of the same order of magnitude as the creep time scale, the plasticity and creep equations must be solved using a coupled solution procedure.

ABAQUS has a creep model that can be used to augment the Drucker-Prager plasticity for such problems.

**Basic Assumptions**

ABAQUS always uses the coupled solution procedure when both Drucker-Prager plasticity and creep are active.

Using the Drucker-Prager creep model implies that the Drucker-Prager plasticity model uses isotropic linear elasticity, a hyperbolic plastic flow potential, and the linear Drucker-Prager yield surface with a circular yield surface in the deviatoric plane ($K = 1$).
The creep laws for the Drucker-Prager creep models are written in terms of an equivalent creep stress, $\bar{\sigma}^{cr}$, which is a measure of the creep “intensity” of the state of stress at a material point.

The definition of $\bar{\sigma}^{cr}$ depends upon the type of hardening (compression, tension, or shear) used with the linear Drucker-Prager plasticity model, but in all cases $\bar{\sigma}^{cr} = \bar{\sigma}^{cr}(q, p, \beta)$:

\[
\bar{\sigma}^{cr} = \frac{(q - p \tan \beta)}{(1 - (1/3) \tan \beta)} \quad \text{(compression)}
\]

\[
= \frac{(q - p \tan \beta)}{(1 + (1/3) \tan \beta)} \quad \text{(tension)}
\]

\[
= (q - p \tan \beta) \quad \text{(shear)}
\]

The equivalent creep stress defines surfaces that are parallel to the yield surface in the meridional plane.
Points on the same surface have the same creep “intensity.”

Figure 3–1. Equivalent Creep Surface in the Meridional Plane

There is a cone in the meridional plane in which no creep deformation will occur.
Creep Laws

The default creep laws provided are simple and are intended to model the secondary creep of the material.

**Time Hardening Creep Law**

Use this creep law when the stress in the material remains essentially constant:

\[ \dot{\varepsilon}^{cr} = A (\bar{\sigma}^{cr})^n t^m. \]

**Strain Hardening Creep Law**

Use this creep law when the stress in the material varies during the analysis:

\[ \dot{\varepsilon}^{cr} = \left( A (\bar{\sigma}^{cr})^n [ (m + 1) \bar{\varepsilon}^{cr} ]^m \right)^{\frac{1}{m+1}}. \]
**Singh-Mitchell Creep Law**

Use this creep law when an exponential relationship between stress and creep strain rate is needed:

\[ \dot{\varepsilon}^{cr} = Ae^{(\alpha \bar{\sigma}^{cr})}(t_1/t)^m. \]

**Creep Flow Potential**

The Drucker-Prager creep model uses a hyperbolic creep flow potential that ensures the creep (deformation) flow direction is always defined uniquely:

\[ G^{cr} = \sqrt{(\varepsilon \bar{\sigma} |_0 \tan \psi)^2 + q^2 - p \tan \psi}. \]

The initial yield stress, \( \bar{\sigma} |_0 \), is defined on the *DRUCKER PRAGER HARDENING* option.
**Usage**

The *DRUCKER PRAGER CREEP* option must be used in conjunction with the *DRUCKER PRAGER* and *DRUCKER PRAGER HARDENING* options.

The *DRUCKER PRAGER CREEP* option must be used with the linear Drucker-Prager model with a von Mises (circular) section in the deviatoric stress plane ($K = 1$; i.e., no third stress invariant effects are taken into account) and can be combined only with linear elasticity.

The material parameters in the default creep laws—$A$, $n$, $m$, $t_1$, and $\alpha$—can be defined as functions of temperature and/or field variables on the *DRUCKER PRAGER CREEP* option.

- To avoid numerical problems with round-off, the values of $A$ should be larger than $10^{-27}$.
The time in these creep laws is the total analysis time, so the duration of steps where creep is not considered (such as *STATIC steps) should be relatively short.

More complex creep laws are defined with user subroutine CREEP. The eccentricity of the creep potential, ε, is by default 0.1. Use the ECCENTRICITY parameter on the *DRUCKER PRAGER option to specify a different value.

– Using values much smaller than 0.1 can create convergence problems.

The creep flow potential uses the same dilation angle, ψ, as the Drucker-Prager plasticity model.

– Therefore, it is possible for the creep equations to be unsymmetric when β ≠ ψ. In this case the *STEP, UNSYMM=YES option should be used.
Modified Cam-Clay Model

This model is intended to simulate the constitutive behavior of cohesionless materials. It is an extended version of the classical critical state theories originally developed at Cambridge from 1960–1970.

The basic characteristics of the model are:

– There is a regime of nonlinear elastic response, after which some of the material deformation is not recoverable and can, thus, be idealized as being plastic.

– The material is initially isotropic.

– The yield behavior depends on the hydrostatic pressure. The critical state line separates two distinct regions of behavior: on the “dry” side of critical state the material softens, while on the “wet” side it hardens (and also stiffens). The hardening/softening behavior is a function of the volumetric plastic strain.
– The inelastic behavior is generally accompanied by volume changes: on the “dry” side the material dilates, while on the “wet” side it compacts. On the critical state line the material can yield indefinitely at constant shear stress without changing volume.

– The yield behavior may be influenced by the magnitude of the intermediate principal stress.

– The model assumes the material is cohesionless.

– Under large stress reversals the model provides a reasonable material response on the “wet” (cap) side of critical state; however, on the “dry” side the model is acceptable only for essentially monotonic loading.

– Temperature may affect the material properties.
Modified Cam-Clay Model

Analysis of Geotechnical Problems with ABAQUS

Response on the “Dry” Side of Critical State
Modified Cam-Clay Model

Response on the “Wet” Side of Critical State
**Description**

Either linear elasticity or nonlinear porous elasticity—see **Elasticity** (p. L3.12)—can be used with the Cam-clay model.

The modified Cam-clay yield surface is elliptical in the meridional plane and also includes a dependence on the third stress invariant:

for \( p > a \) (the “wet” side of critical state),

\[
f(p, q, r) = \frac{1}{\beta^2} \left( \frac{p}{a} - 1 \right)^2 + \left( \frac{t}{Ma} \right)^2 - 1 = 0 ;
\]

for \( p \leq a \) (the “dry” side of critical state),

\[
f(p, q, r) = \left( \frac{p}{a} - 1 \right)^2 + \left( \frac{t}{Ma} \right)^2 - 1 = 0 .
\]
Modified Cam-Clay Model

Analysis of Geotechnical Problems with ABAQUS
\( \beta \) is a constant used to modify the shape of the yield surface on the “wet” side of critical state so that the elliptic arc on the “wet” side of critical state has a different curvature from the elliptic arc used on the “dry” side: \( \beta = 1 \) on the “dry” side of critical state, while \( \beta < 1 \) in most cases on the “wet” side.

The measure of deviatoric stress, \( t \), allows matching of different stress values in tension and compression in the deviatoric plane, as discussed previously in the context of the Drucker-Prager models—see Linear Drucker-Prager Model (p. L3.31).

\[
t = \frac{1}{2} q \left[ 1 + \frac{1}{K} \cdot \left( 1 - \frac{1}{K} \right) \left( \frac{t}{q} \right)^3 \right]
\]

<table>
<thead>
<tr>
<th>Curve</th>
<th>K</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>1.0</td>
</tr>
<tr>
<td>b</td>
<td>0.8</td>
</tr>
</tbody>
</table>
$M$ is the slope of the critical state line in the $p-t$ plane (the ratio of $t$ to $p$ at critical state).
Associated flow is used with the modified Cam-clay model. The size of the yield surface is defined by $a$. The evolution of this variable, therefore, characterizes the hardening or softening of the material.

ABAQUS provides two approaches to defining the evolution $a(\varepsilon_{vol}^{pl})$. For some materials, over the range of confining pressures of interest it is observed experimentally that, during plastic deformation,

$$de = -\lambda d(\ln p),$$

where $\lambda$ is a constant and $e$ is the voids ratio. Integrating this equation:

$$a = a_0 \exp \left[ (1 + e_0) \frac{1-J^{pl}}{\lambda - \kappa J^{pl}} \right],$$

where $e_0$ is the initial voids ratio, $\kappa$ is the porous elasticity volumetric constant (the logarithmic bulk modulus), and $a_0$ defines the position of $a$ at the start of the analysis—the initial overconsolidation of the material.
The value of $a_0$ can be given directly or may be computed as

$$a_0 = \frac{1}{2} \exp \left( \frac{e_1 - e_0 - \kappa \ln p_0}{\lambda - \kappa} \right),$$

where $p_0$ is the initial value of the pressure stress and $e_1$ is the intercept of the virgin consolidation line with the voids ratio axis in a plot of voids ratio versus pressure stress.
Alternatively, the evolution of the yield surface can be defined as a piecewise linear function relating yield stress in hydrostatic compression, $p_c$, and volumetric plastic strain, $\varepsilon_{\text{vol}}^{pl}$ : 

\[ p_c = p_c(\varepsilon_{\text{vol}}^{pl}). \]

The evolution parameter is then given by 

\[ a = \frac{p_c}{(1 + \beta)}. \]
The volumetric plastic strain axis has an arbitrary origin: \( \varepsilon_{vol}^{pl} \big|_0 \) is the position on this axis corresponding to the initial state of the material, thus defining the initial hydrostatic pressure, \( p_c \big|_0 \), and, hence, the initial yield surface size, \( a_0 \). Data must be provided over a wide enough range of values of \( p_c \) to cover all situations that will arise in the application.
ABAQUS checks that the yield surface is not violated by the user-specified material constants and the initial effective stress conditions defined at each point in the material. At any material point where the yield function is violated and a warning message is issued, $a_0$ is adjusted so that the yield function is satisfied exactly (and, hence, the initial stress state lies on the yield surface).
**Usage and Calibration**

The modified Cam-clay model in ABAQUS is invoked with the \( \star \text{CLAY PLASTICITY} \) option. This option defines the yield. It also defines the hardening if logarithmic hardening is chosen. The \( \star \text{CLAY HARDENING} \) option can be used to define piecewise linear hardening. These parameters can be entered as functions of temperature and predefined field variables.

The elasticity is defined with the \( \star \text{ELASTIC} \) or \( \star \text{POROUS ELASTIC} \) option. If logarithmic hardening is used, the \( \star \text{POROUS ELASTIC} \) option must be used (in this case the logarithmic elastic bulk modulus, \( \kappa \), must be specified together with either a constant shear modulus, \( G \), or a constant Poisson’s ratio, \( \nu \)).

In the porous elasticity case, the version with zero tensile strength \( (p_{\text{el}}^t = 0) \) is normally used since the material is assumed to be cohesionless. The elasticity parameters can be entered as functions of temperature and predefined field variables.
*EXPANSION can be used to introduce thermal volume change effects.

*INITIAL CONDITIONS, TYPE=RATIO is required to define the initial voids ratio (porosity) of the material. User subroutine VOIDRI can be used to specify complex initial voids ratio distributions.

*INITIAL CONDITIONS, TYPE=STRESS is required to define the initial effective stress state in the material. User subroutine SIGINI can be used to specify complex initial stress distributions.
At least two experiments are required to calibrate the simplest version of the Cam-clay model: a hydrostatic compression test (an oedometer test is also acceptable), and a triaxial compression test (more than one triaxial test is useful for a more accurate calibration).

The hydrostatic compression test is performed by pressurizing the sample equally in all directions. The applied pressure and the volume change are recorded.

Triaxial compression experiments are performed using a standard triaxial machine where a fixed confining pressure is maintained while the differential stress is applied. Several tests covering the range of confining pressures of interest are usually performed. Again, the stress and strain in the direction of loading are recorded together with the lateral strain so that the correct volume changes can be calibrated.

Unloading measurements in these tests are useful to calibrate the elasticity, particularly in cases where the initial elastic region is not well defined. From these we can identify whether a constant shear modulus or a constant Poisson’s ratio should be used and what its value is.
The onset of yielding in the hydrostatic compression test immediately provides the initial position of the yield surface, $a_0$. The logarithmic bulk moduli, $\kappa$ and $\lambda$, are determined from the hydrostatic compression experimental data by plotting the logarithm of pressure versus voids ratio. The voids ratio, $e$, is related to the measured volume change as

$$J = \exp(\varepsilon_{vol}) = \frac{(1 + e)}{(1 + e_0)}.$$

The slope of the line obtained for the elastic regime is $-\kappa$, and the slope in the inelastic range is $-\lambda$. For a valid model $\lambda > \kappa$. 
The triaxial compression tests allow the calibration of the yield parameters $M$ and $\beta$. $M$ is the ratio of the shear stress, $q$, to the pressure stress, $p$, at critical state and can be obtained from the stress values when the material has become perfectly plastic (critical state). $\beta$ represents the curvature of the cap part of the yield surface and can be calibrated from a number of triaxial tests at high confining pressures (on the “wet” side of critical state). $\beta$ must be between 0 and 1.

To calibrate the parameter $K$, which controls the yield dependence on the third stress invariant, experimental results obtained from a true triaxial (cubical) test are necessary. These results are generally not available, and the user may have to guess (the value of $K$ is generally between 0.8 and 1.0) or ignore this effect.
Example

We consider the homogeneous deformation of a single element as a demonstration of the modified Cam-clay plasticity model (Benchmark Problem 3.2.4 Triaxial tests on a saturated clay). Our modified Cam-clay model provides two extensions of the original Cam-clay model: “capping” of the yield ellipse on the wet side of critical state and consideration of the third stress invariant in the yield function. Both of these extensions are included in this example. The specimen is initially stressed hydrostatically. Subsequently it is subjected to triaxial compression or triaxial extension. The material parameters used in this example are:

- Logarithmic elastic bulk modulus, $\kappa$: 0.026
- Poisson’s ratio, $\nu$: 0.3
- Logarithmic hardening modulus, $\lambda$: 0.174
- Critical state ratio, $M$: 1.0
- Wet cap parameter, $\beta$: 0.5
- Third stress invariant parameter, $K$: 0.75
- Initial overconsolidation parameter, $a_0$: 58.3 KN/m²
Material response:

Compression: "capped" modified Cam-clay ($\beta = 0.5$)
Compression: "standard" modified Cam-clay
Extension: "standard" modified Cam-clay
Extension: "capped" modified Cam-clay with third stress invariant dependence ($K = 0.7$)
Extension: modified Cam-clay with third stress invariant dependence ($K = 0.75$)
Yield surface evolution for triaxial extension:

- Equivalent shear stress, $q/P$
- Mean normal stress, $p/P$

Critical state lines:
- $q = M_p$
- $q = K M_p$

$K = 0.75$

Points on yield surface:
- S
- $S_1$
- O
Input Listing:

*HEADING
   CAM CLAY EXAMPLE - DRAINED TRIAXIAL TESTS
*NODE
  1
  3,1.
  23,1..1.
  21,,1.
*NGEN,NSET=BOTTOM
  1,3
*NGEN,NSET=TOP
  21,23
*NGEN
  1,21,10
  3,23,10
*NGEN,NSET=SOIL,GENERATE
  1,23
*NGEN,NSET=LHS
  1,11,21
*ELEMENT,TYPE=CAX8R,ELSET=SOIL
  1,1,3,23,21,2,13,22,11
*SOLID SECTION, MATERIAL=SAMPLE, ELSET=SOIL
*MATERIAL, NAME=SAMPLE
*POROUS ELASTIC
0.026, 0.3
*CLAY PLASTICITY
0.174, 0.1, 58.3
*INITIAL CONDITIONS, TYPE=RATIO
SOIL, 1.08, 0.1, 0.08, 1.
*INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
SOIL, -100, 0, -100, 10, 1.
*STEP
GEOSTATIC INITIAL STRESS STATE
*GEOSTATIC
*DLOAD
1, P2, 100.
*EL PRINT
S
SINV
E
PE
EE
*NODE PRINT
U
RF
*EL FILE
SINV
*NODE FILE,NSET=TOP
U
*BOUNDARY
TOP,2
BOTTOM,2
LHS,1
*END STEP
*STEP,INC=20
   TRIAXIAL COMPRESSION
*STATIC,DIRECT
1.,20.
*BOUNDARY
TOP,2,2,-.5
*END STEP
Modified Cap Model

This model is intended to simulate the constitutive response of cohesive geological materials. It adds a “cap” yield surface to the linear Drucker-Prager model, which serves two main purposes: it bounds the model in hydrostatic compression, and it helps control volume dilatancy when the material yields in shear.

The basic characteristics of this model are:

- There is a regime of purely elastic response, after which some of the material deformation is not recoverable and can, thus, be idealized as being plastic.

- The yield behavior depends on the hydrostatic pressure. There are two distinct regions of behavior: on the failure surface the material is perfectly plastic, while on the cap yield surface it hardens (and also stiffens). The hardening/softening behavior is a function of the volumetric plastic strain.
– The inelastic behavior is generally accompanied by volume changes: on the failure surface the material dilates, while on the cap surface it compacts. At the intersection of the surfaces, the material can yield indefinitely at constant shear stress without changing volume.

– The yield behavior may be influenced by the magnitude of the intermediate principal stress.

– Under large stress reversals the model provides reasonable material response on the cap region; however, on the failure surface region the model is acceptable only for essentially monotonic loading.

– The material is initially isotropic.

– Temperature may affect the material properties.
Description

Linear elasticity or nonlinear porous elasticity—see Elasticity (p. L3.12)—can be used with this model.

The model uses two main yield surface segments: a linearly pressure-dependent Drucker-Prager shear failure surface and a compression cap yield surface. The Drucker-Prager failure surface itself is perfectly plastic (no hardening), but plastic flow on this surface produces inelastic volume increase, which causes the cap to soften. The Drucker-Prager failure surface is

\[ F_s = t - p \tan \beta - d = 0. \]

\( \beta \) is the angle of friction, and \( d \) is the cohesion of the material.

The measure of deviatoric stress, \( t \), allows matching of different stress values in tension and compression in the deviatoric plane, as discussed previously in the context of the Drucker-Prager models.
Modified Cap Model

Analysis of Geotechnical Problems with ABAQUS

\[ \text{Shear failure, } F_s \]

\[ \text{Transition surface, } F_t \]

\[ \text{Cap, } F_c \]

\[ \alpha(d+p_a \tan \beta) \]

\[ d+p_a \tan \beta \]

\[ p_a \]

\[ p_b \]

\[ R(d+p_a \tan \beta) \]
The cap yield surface has an elliptical shape with constant eccentricity in the meridional \((p-t)\) plane and also includes dependence on the third stress invariant in the deviatoric plane. The cap surface hardens or softens as a function of the volumetric plastic strain: volumetric plastic compaction (when yielding on the cap) causes hardening, while volumetric plastic dilation (when yielding on the shear failure surface) causes softening.

The cap yield surface is

\[
F_c = \sqrt{(p - p_a)^2 + \left(\frac{Rt}{(1 + \alpha - \alpha/\cos \beta)}\right)^2} - R(d + p_a \tan \beta) = 0,
\]

where \(R\) is a material parameter that controls the shape of the cap, \(\alpha\) is a small number, and \(p_a(\varepsilon_{vol}^{pl})\) is an evolution parameter that represents the volumetric plastic strain driven hardening/softening.
The hardening/softening law is a user-defined piecewise linear function relating the hydrostatic compression yield stress, $p_b$, and the corresponding volumetric plastic strain,

$$p_b = p_b \left( \varepsilon_{\text{vol}}^p \bigg|_0 + \varepsilon_{\text{vol}}^p \right).$$

This relationship is defined in the *CAP HARDENING* option. The range of values for which $p_b$ is defined should be sufficient to include all values of effective pressure stress that the material will be subjected to during the analysis.
The volumetric plastic strain axis in the hardening curve has an arbitrary origin: $\varepsilon_{\text{vol}}^{pl}|_0$ is the position on this axis corresponding to the initial state of the material when the analysis begins, thus defining the position of the cap ($p_b$) at the start of the analysis.

The evolution parameter $p_a$ is then given as

$$p_a = \frac{p_b - Rd}{(1 + R \tan \beta)}.$$

The parameter $\alpha$ is a small number (typically 0.01 to 0.05) used to define a transition yield surface,

$$F_t = \sqrt{(p - p_a)^2 + \left[ t - \left( 1 - \frac{\alpha}{\cos \beta} \right)(d + p_a \tan \beta) \right]^2} - \alpha(d + p_a \tan \beta) = 0,$$

so that the model provides a smooth intersection between the cap and failure surfaces.
A larger value of $\alpha$ can be used to construct more complex (curved) failure surfaces:

This approach has practical value, in that the curved surface provides a better fit of experimentally observed shear failure and also provides a softening surface that is consistent with observed shear failure.

The value of $\alpha$ can be set to zero, in which case there is no transition surface and no true softening behavior (i.e., softening while yielding on the same surface) in the model.
Plastic flow is defined by a flow potential that is associated in the deviatoric plane, associated in the cap region in the meridional plane, and nonassociated in the failure surface and transition regions in the meridional plane. The flow potential surface we use in the meridional plane is made up of an elliptical portion in the cap region that is identical to the cap yield surface,

\[
G_c = \sqrt{(p - p_a)^2 + \left(\frac{Rt}{1 + \alpha - \alpha/\cos\beta}\right)^2},
\]

and another elliptical portion in the failure and transition regions that provides the nonassociated flow component in the model,

\[
G_s = \sqrt{[(p_a - p)\tan\beta]^2 + \left(\frac{t}{1 + \alpha - \alpha/\cos\beta}\right)^2}.
\]

The two elliptical portions form a continuous and smooth potential surface.
On the failure and transition regions the volumetric plastic strain rate is proportional to \((p_a - p)\tan \beta\) and the deviatoric plastic strain rate is proportional to \(t\). Thus, the flow is purely volumetric at the apex of the Drucker-Prager cone, while the flow is purely deviatoric at the intersection of the Drucker-Prager cone with the cap (i.e., the critical state condition). Moving along the Drucker-Prager failure surface between these two extremes, the ratio between the volumetric and deviatoric plastic strain rates varies linearly.
If the initial stress is given such that the stress point lies outside the initially defined cap or transition yield surfaces, ABAQUS will try to adjust the initial position of the cap to make the stress point lie on the yield surface and a warning will be issued.

If the stress point lies outside the Drucker-Prager failure surface, an error message will be issued and the analysis will be terminated.
Usage and Calibration

The modified Cap plasticity model in ABAQUS is invoked with the *CAP PLASTICITY material option. This option allows the yield and flow rule parameters to be made dependent on temperature and predefined field variables.

The *CAP PLASTICITY option must always be accompanied by the *CAP HARDENING material option, where the evolution of the yield stress in hydrostatic compression is defined. The yield stress can be given as a function of temperature and predefined field variables.

The elasticity is defined with the *ELASTIC material option in the case of linear elasticity or with the *POROUS ELASTIC option if nonlinear elasticity is chosen. The elasticity parameters can be entered as functions of temperature and predefined field variables.

*EXPANSION can be used to introduce thermal volume change effects.
*INITIAL CONDITIONS, TYPE=RATIO is required to define the initial voids ratio (porosity) of the material if nonlinear elasticity is used. User subroutine VOIDRI can be used to specify complex initial voids ratio distributions.

The nonassociated flow present in the failure surface of the model produces an unsymmetric material stiffness matrix. The UNSYMM=YES parameter on the *STEP option should be used if there is a significant plastic flow due to shearing. When ABAQUS uses a symmetric solver with unsymmetric equations, the rate of convergence may be slow. If the region of the model in which nonassociated flow occurs is confined, it may be possible to omit UNSYMM=YES and still get an acceptable rate of convergence.
At least three experiments are required to calibrate the simplest version of the Cap model: a hydrostatic compression test (an oedometer test is also acceptable) and two uniaxial and/or triaxial compression tests (more than two tests is useful for a more accurate calibration).

The hydrostatic compression test is performed by pressurizing the sample equally in all directions. The applied pressure and the volume change are recorded.

The uniaxial compression test involves compressing the sample between two rigid platens. The load and displacement in the direction of loading are recorded. The lateral displacements should also be recorded so that the correct volume changes can be calibrated.
Triaxial compression experiments are performed using a standard triaxial machine where a fixed confining pressure is maintained while the differential stress is applied. Several tests covering the range of confining pressures of interest are usually performed. Again, the stress and strain in the direction of loading are recorded, together with the lateral strain, so that the correct volume changes can be calibrated.

Unloading measurements in these tests are useful to calibrate the elasticity, particularly in cases where the initial elastic region is not well defined.

The hydrostatic compression test stress-strain curve gives the evolution of the hydrostatic compression yield stress, $p_b(\varepsilon_{vol}^{pl})$, required in the *CAP HARDENING option.
The friction angle, $\beta$, and cohesion, $d$, which define the shear failure dependence on hydrostatic pressure, are calculated by plotting the failure stresses of any two uniaxial and/or triaxial compression experiments in the pressure stress ($p$) versus shear stress ($q$) space: the slope of the straight line passing through the two points gives the angle $\beta$, and the intersection with the $q$-axis gives $d$. See the discussion in Linear Drucker-Prager Model (p. L3.31).

$R$ represents the curvature of the cap part of the yield surface and can be calibrated from a number of triaxial tests at high confining pressures (in the cap region). $R$ must be between 0 and 1.

To calibrate the parameter $K$, which controls the yield dependence on the third stress invariant, experimental results obtained from a true triaxial (cubical) test are necessary. These results are generally not available, and the user may have to guess (the value of $K$ is generally between 0.8 and 1.0) or ignore this effect. See the discussion in Linear Drucker-Prager Model (p. L3.31).
Example

We simulate the behavior of a sample of McCormack Ranch sand under uniaxial strain conditions. The response is compared to the experimental result given by DiMaggio and Sandler (1976).

The specimen has the following Cap model properties:

\[ E = 100 \text{ Ksi} \]
\[ \nu = 0.25 \]
\[ \beta = 14.56^\circ \]
\[ d = 0.1732 \text{ Ksi} \]
\[ R = 0.1 \]
\[ \varepsilon_{\text{vol}}^{p l} \bigg|_0 = 0.001 \]
\[ K = 1.0 \]

Hardening as shown in the *CAP HARDENING option in the input listing
Material response:

McCormack Ranch Sand - Uniaxial strain test

- Experimental (DiMaggio & Sandler)
- ABAQUS Cap model

Axial stress (Ksi) vs. Axial strain
Input listing:

*HEADING
CAP PLASTICITY,McCORMACK RANCH SAND,UNIAXIAL STRAIN, C3D8
*** KSI UNITS
*NODE, NSET=BOT
1,0.,0.,0.
2,1.,0.,0.
3,1.,1.,0.
4,0.,1.,0.
*NODE, NSET=TOP
5,0.,0.,1.
6,1.,0.,1.
7,1.,1.,1.
8,0.,1.,1.
*NSET, NSET=ALLN
BOT, TOP
*NSET, NSET=BACK
1,4,5,8
*NSET, NSET=FRONT
2,3,6,7
*NSET, NSET=LHS
1, 2, 5, 6
*NSET, NSET=RHS
3, 4, 7, 8
*ELEMENT, TYPE=C3D8, ELSET=EL1
1, 1, 2, 3, 4, 5, 6, 7, 8
*SOLID SECTION, ELSET=EL1, MATERIAL=CAPL
*MATERIAL, NAME=CAPL
*ELASTIC
100., .25
*CAP PLASTICITY
**COHESN, BETA, BIGR, EVOLPI, ALPHA, BIGK
   .1732, 14.56, 0.1, 0.001,  , 1.0
*CAP HARDENING
** PB, EVOLP
   .02 , .0
   .025 , .005
   .063 , .01
   .13 , .02
   .24 , .03
   .4 , .04
   .6 , .05
1.  ,.06
5.  ,.1
*BOUNDARY
BACK,1
LHS,2
BOT,3
RHS,2
TOP,3
*STEP, INC=20, UNSYMM=YES
*STATIC, DIRECT
1., 20.
*BOUNDARY
FRONT,1,,-.066
*EL PRINT
S
SINV
E
PE
PEQC
*NODE PRINT
U, RF
*END STEP
*STEP, INC=5, UNSYMM=YES
*STATIC, DIRECT
  1., 5.
*BOUNDARY
  FRONT, 1, , -.05
*END STEP
Coupled Creep and Cap Plasticity

Geomaterials can creep under certain conditions. When the loading rate is of the same order of magnitude as the creep time scale, the plasticity and creep equations must be solved using a coupled solution procedure.

ABAQUS has a creep model that can be used to augment the Cap plasticity for such problems.

**Basic Assumptions**

Cap plasticity with creep always uses the coupled solution procedure.

The Cap creep model can be used if the Cap plasticity model uses isotropic linear elasticity, a circular yield surface in the deviatoric plane ($K=1$), and no transition zone between the shear failure region and the cap region ($\alpha = 0$).
The creep model has two creep mechanisms:

- Cohesive creep, which is active in both the shear failure region and the cap region.
- Consolidation creep, which is active only in cap region.

Figure 3–2. Equivalent Creep Surface for the Cap Creep Model
Cohesion Creep

The cohesion creep mechanism is written in terms of an equivalent creep stress, $\bar{\sigma}^{cr}$, which is a measure of the creep “intensity” of the state of stress at a material point.

The cohesive creep properties must be measured in uniaxial compression. The format of $\bar{\sigma}^{cr}$ is

$$
\bar{\sigma}^{cr} = \frac{(q - p\tan\beta)}{\left(1 - \frac{1}{3}\tan\beta\right)}.
$$
The equivalent creep stress defines surfaces that are parallel to the shear failure surface in the meridional plane (see Figure 3–2). There is a cone in the meridional plane where no creep deformation occurs. ABAQUS also requires that $\sigma^{cr}$ be positive.
Consolidation Creep

The consolidation creep mechanism is dependent on the hydrostatic pressure above a threshold value of $p_a$, with a smooth transition to the areas in which the mechanism is not active.

Therefore, equivalent creep surfaces are constant hydrostatic pressure surfaces (vertical lines in the $p - q$ plane).

The consolidation creep laws are expressed in terms of an equivalent consolidation creep pressure stress, $\bar{p}^{cr} = p - p_a$.

Creep Laws

The default creep laws available for the Cap creep models are the same as those available for the Drucker-Prager creep model: time and strain hardening laws and a Singh-Mitchell creep law.

- See Coupled Creep and Drucker-Prager Plasticity (p. L3.64) for details.
Creep Flow Potentials

The cohesion creep mechanism uses a hyperbolic creep potential in the meridional plane.

This creep flow potential, which is continuous and smooth, ensures that the flow direction is always defined uniquely. The cohesion creep potential is the von Mises circle in the deviatoric stress plane.
The consolidation mechanism uses an elliptical flow potential that is similar to the Cap plasticity flow potential in the $p - q$ plane. The consolidation creep potential is the von Mises circle in the deviatoric stress plane.
Usage

The *CAP CREEP option must be used in conjunction with the *CAP PLASTICITY and *CAP HARDENING options.

The *CAP CREEP option must be used with a cap plasticity model that has no third stress invariant effects \((K = 1)\) and has no transition surface \((\alpha = 0)\). In addition, it can be combined only with linear elasticity.

The material parameters in the default creep laws—\(A, n, m, t_1\), and \(\alpha\)—can be defined as functions of temperature and/or field variables on the *CAP CREEP option.

- To avoid numerical problems with round-off, the values of \(A\) should be larger than \(10^{-27}\).
Use the MECHANISM parameter on the *CAP CREEP option to specify which behavior, CONSOLIDATION or COHESION, is being defined.

ABAQUS requires that cohesion creep properties be measured in a uniaxial compression test.

The time in these creep laws is the total analysis time, so the duration of steps where creep is not considered (such as *STATIC steps) should be relatively short.

More complex creep laws are defined with user subroutine CREEP.

The use of a creep potential for the cohesion mechanism different from the equivalent creep surface implies that the material stiffness matrix is not symmetric and the unsymmetric matrix storage and solution scheme (UNSYMM=YES) should be used.
Jointed Material Model

This model is intended to provide a simple, continuum model for materials containing a high density of parallel joint surfaces in different orientations.

The spacing of the joints of a particular orientation is assumed to be sufficiently close compared to characteristic dimensions in the domain of the model that the joints can be smeared into a continuum of slip systems.

An obvious application is the modeling of geotechnical problems where the medium of interest is composed of significantly faulted rock.
Figure 3–3. Cap Creep Flow Potentials
The basic characteristics of the model are:

- There is a regime of purely elastic response, after which some of the material deformation is not recoverable and can, thus, be idealized as being plastic.

- The model provides for up to three joint systems that may exhibit frictional sliding and may also open and close. Whenever any joint system is open, the material response becomes orthotropic.

- The model also includes a bulk failure mechanism based on a Drucker-Prager model.

- The inelastic sliding mechanisms on the joints and bulk material may be purely frictional or include dilation.

- The model provides a reasonable material response under large stress reversals (including joint opening/closing and cyclic shear).

- Temperature may affect the material properties.
Jointed Material Model

Analysis of Geotechnical Problems with ABAQUS

Jointed element and joint orientation

vertical joint set

inclined joint set

Joint set a

n_a

t_a1

t_a2
Description

Consider joint $a$ oriented by the normal to the joint surface $n_a$. $t_{a\alpha}$, $\alpha = 1, 2$, are two unit, orthogonal vectors in the joint surface. The local stresses are the pressure stress and the shear stresses across the joint

$$P_a = -n_a \cdot \sigma \cdot n_a,$$
$$\tau_{a\alpha} = n_a \cdot \sigma \cdot t_{a\alpha}.$$ 

We define the shear stress magnitude as

$$\tau_a = \sqrt{\tau_{a\alpha} \tau_{a\alpha}}.$$

The local strains are the normal strain across the joint

$$\varepsilon_{an} = n_a \cdot \varepsilon \cdot n_a$$

and the engineering shear strain in the $\alpha$-direction in the joint surface

$$\gamma_{a\alpha} = n_a \cdot \varepsilon \cdot t_{a\alpha} + t_{a\alpha} \cdot \varepsilon \cdot n_a.$$
We use a linear strain rate decomposition:

\[ d\varepsilon = d\varepsilon^e + d\varepsilon^p. \]

If several systems are active (we designate an active system by \( i \), where \( i = b \) indicates the bulk material system and \( i = a \) is a joint system \( a \)):

\[ d\varepsilon^p = \sum_i d\varepsilon_i^p. \]

When all joints at a point are closed, the elastic behavior of the material is isotropic and linear. We use a stress-based joint opening criterion, whereas joint closing is monitored based on strain. Joint system \( a \) opens when the estimated pressure stress across the joint (normal to the joint surface) is no longer positive:

\[ p_a \leq 0. \]
In this case the material has no elastic stiffness with respect to direct strain across the joint system and may or may not have stiffness with respect to shearing associated with this direction. Thus, open joints create anisotropic elastic response at a point. The joint system remains open so long as

\[ \varepsilon_{an(ps)}^{el} \leq \varepsilon_{an}^{el}, \]

where \( \varepsilon_{an}^{el} \) is the component of direct elastic strain across the joint and \( \varepsilon_{an(ps)}^{el} \) is the component of direct elastic strain across the joint calculated in plane stress as

\[ \varepsilon_{an(ps)}^{el} = -\frac{\nu}{E}(\sigma_{a1} + \sigma_{a2}), \]

where \( E \) is the Young’s modulus of the material, \( \nu \) is the Poisson’s ratio and

\[ \sigma_{a\alpha} = t_{a\alpha} \cdot \sigma \cdot t_{a\alpha} \]

are the direct stresses in the plane of the joint.
The failure surface for sliding on joint $a$ is

$$f_a = \tau_a - p_a \tan \beta_a - d_a = 0,$$

where $\beta$ is the friction angle and $d_a$ is the cohesion for system $a$. So long as $f_a < 0$, joint $a$ does not slip. When $f_a = 0$, joint $a$ slips. The inelastic strain on the system is

$$d\varepsilon_{pl}^a = d\varepsilon_{pl}^a \left( \sin \psi_a n_a n_a + \frac{\tau_{a\alpha}}{\tau_a} \cos \psi_a (n_a t_{a\alpha} + t_{a\alpha} n_a) \right),$$

where $d\varepsilon_{pl}^a$ is the magnitude of the inelastic strain rate and $\psi_a$ is the dilation angle for this joint system (choosing $\psi_a = 0$ provides pure shear flow on the joint, while $\psi_a > 0$ causes dilation of the joint as it slips).

The sliding of the different joint systems at a point is independent, in the sense that sliding on one system does not change the failure criterion or the dilation angle for any other joint system at the same point.
Joint system model:

\[ \tau_a \]

\[ \beta_a \]

\[ d_a \]

\[ p_a \]

\[ f_a \]

\[ \psi_a \]

\[ \text{de}^{pl}_a \]
In addition to the joint systems, the model includes a bulk material failure mechanism based on the linear Drucker-Prager failure criterion

\[ q - p \tan \beta_b - d_b = 0, \]

where \( \beta_b \) is the friction angle for the bulk material and \( d_b \) is the cohesion for the bulk material. If this failure criterion is reached, the bulk inelastic flow is defined by

\[ d\varepsilon_{b}^{pl} = d\varepsilon_{b}^{pl} \frac{1}{1 - \frac{1}{3} \tan \psi_b} \frac{\partial g_b}{\partial \sigma}, \]

\[ g_b = q - p \tan \psi_b, \]

where \( \psi_b \) is the dilation angle for the bulk material. This bulk failure model is a simplified version of the modified Drucker-Prager model described earlier in this lecture. As with the joint systems, this bulk failure system is independent of the joint systems in that bulk inelastic flow does not change the behavior of any joint system.
Bulk material model:
Usage and Calibration

The jointed material model in ABAQUS is invoked with the *JOINTED MATERIAL material option. This option allows the yield and flow rule parameters to be made dependent on temperature and predefined field variables. This option must be repeated for each existing system (bulk material and up to three joints); it may appear four times.

When defining a joint system, the JOINT DIRECTION parameter must be used to refer to the orientation definition corresponding to the joint. The *ORIENTATION option must then be used to define the joint orientation in the original configuration. Stress and strain components will still be output in global directions unless the *ORIENTATION option is also used on the section definition option associated with the material definition.

When defining the bulk material, the JOINT DIRECTION parameter must be omitted.

The *JOINTED MATERIAL option may appear a fifth time with the SHEAR RETENTION parameter to define a nonzero shear modulus for open joints.
The elasticity must be defined with the *ELASTIC, TYPE=ISOTROPIC material option since we assume that the material is linear elastic and isotropic when all joints are closed. The material cannot be elastically incompressible (Poisson’s ratio must be less than 0.5).

*EXPANSION can be used to introduce thermal volume change effects.

Analyses using a nonassociated flow version of the model may require the use of the UNSYMM=YES parameter on the *STEP option because of the resulting nonsymmetry of the plasticity equations.
At least two experiments are required to calibrate the behavior of each of the existing inelastic mechanisms.

For the bulk material the experiments commonly performed for this purpose are uniaxial compression (for cohesive materials) and triaxial compression tests.

The uniaxial compression test involves compressing the sample between two rigid platens. The load and displacement in the direction of loading are recorded. The lateral displacements should also be recorded so that the correct volume changes can be calibrated.

Triaxial compression experiments are performed using a standard triaxial machine where a fixed confining pressure is maintained while the differential stress is applied. Several tests covering the range of confining pressures of interest are usually performed. Again, the stress and strain in the direction of loading are recorded, together with the lateral strain, so that the correct volume changes can be calibrated.
For the joint systems the most common experiments are shear tests. In these tests a normal pressure is applied across the joint and then the joint is sheared. Several tests covering the range of pressures of interest are usually performed. Again, the stresses and strains in the normal and shear directions must be recorded.

Unloading measurements in all of the above tests are useful to calibrate the elasticity, particularly in cases where the initial elastic region is not well-defined.

The angle of friction, $\beta$, and the cohesion, $d$, defining the failure stress dependence on pressure are calculated by plotting the failure stresses of any two experiments in the pressure stress versus shear stress space: the slope of the straight line passing through the two points gives $\beta$, and the intersection with the shear stress axis gives $d$. 
If more than two experiments are available, a best fit straight line over the range of interest of pressure stress must be used to calculate the slope of the failure surface.

Refer also to the calibration discussion in **Linear Drucker-Prager Model** (p. L3.31).

The dilation angle, $\psi$, must be chosen such that a reasonable match of the volume changes during yielding is obtained. The volume changes are calculated from the strains in all directions.

The calibration of the temperature-dependent model requires repetition of the experiments at different temperatures over the range of interest.
Example

We consider a sample of material subjected to uniaxial compression/tension. The material has two sets of planes of weakness having an included angle of $2\alpha$. We construct the failure envelope of the material as the orientation ($\theta$) of the planes of weakness is varied. The joints have cohesion $d_a = 1000$ and friction angle $\beta_a = 45^\circ$. The bulk material has cohesion $d_b = 8000$ and friction angle $\beta_b = 45^\circ$. Plastic flow in the joints and bulk material is associated. When all joints are closed, the material is isotropic linear elastic with $E = 3 \times 10^5$ and $\nu = 0.3$. When a joint opens, the material is assumed to have no elastic stiffness. We show the variation of the compressive failure stress $P/2d_a$ with $\theta$ for $\alpha = 0^\circ, 20^\circ, 30^\circ$. For certain ranges of orientation of the joints, failure along the joints becomes increasingly improbable and failure of the bulk material takes place first. In tension the material cannot carry any stress since the joints open readily.
Problem Geometry (Benchmark Problem 3.2.5)
Jointed Material Model

Analysis of Geotechnical Problems with ABAQUS

Uniaxial Compression Failure Envelopes

Bulk Material Failure

$\frac{P}{2d_a}$ vs. $\theta$

- $\alpha = 0^\circ$ (Jeager's Theory)
- $\alpha = 20^\circ$
- $\alpha = 30^\circ$
**Input Listing:**

*HEADING
UNIAXIAL COMPRESSION TEST, 2 JOINTS, ALFA=20 THETA=0
*WAVEFRONT MINIMIZATION, SUPPRESS
*NODE, NSET=ALLN
1, 0., 0., 0.
2, 1., 0., 0.
3, 1., 1., 0.
4, 0., 1., 0.
5, 0., 0., 1.
6, 1., 0., 1.
7, 1., 1., 1.
8, 0., 1., 1.
*ELEMENT, TYPE=C3D8, ELSET=ALLE
1, 1, 2, 3, 4, 5, 6, 7, 8
*SOLID SECTION, ELSET=ALLE, MATERIAL=ALLE
*MATERIAL, NAME=ALLE
*ELASTIC
300. E3, .3
*JOINTED MATERIAL
45., 45., 8000.
*JOINTED MATERIAL,JOINT DIRECTION=JOINT1
  45.,45.,1000.
*JOINTED MATERIAL,JOINT DIRECTION=JOINT2
  45.,45.,1000.
*ORIENTATION,NAME=JOINT1
  -.9397,.342,0.,-.342,-.9397,0.
*ORIENTATION,NAME=JOINT2
  .9397,.342,0.,-.342,.9397,0.
*BOUNDARY
  1,PINNED
  2,2
  5,2
  6,2
  4,1
  5,1
  8,1
  2,3
  3,3
  4,3
*STEP,INC=20
*STATIC,DIRECT
  1.,20.
*BOUNDARY
7,2,,-.2
3,2,,-.2
4,2,,-.2
8,2,,-.2
*EL PRINT
S
SINV
E
PE
PEQC
EE
*NODE PRINT
U,RF
*EL FILE,FREQUENCY=2
S,E,PE
*END STEP
Numerical Implementation

For all the material models presented in this set of notes, the constitutive behavior is given in rate form—the inelastic strain is defined only as a strain rate.

This must be integrated over each finite time increment. We use backward Euler integration:

$$\Delta \varepsilon^{pl} = \frac{d\varepsilon^{pl}}{dt} \bigg|_{t+\Delta t} \Delta t,$$

where $\Delta t$ is the time increment and $t$ is the time at the beginning of the increment.

The backward Euler method (more traditionally called “radial return” because of its simple geometric interpretation for the basic case of a Mises yield and associated flow model) is chosen because it is unconditionally stable. In addition, it provides acceptable accuracy, especially when the strain increment is large compared to the strain to cause yield.
In some simple cases, such as a perfectly plastic Drucker-Prager model, the backward difference equations can be solved in closed form. However, in most cases, Newton’s method is used for the numerical solution of the integrated plasticity equations.

In models that involve opening/closing conditions (such as in the jointed material model) and in models that involve more than one yield surface (such as the capped Drucker-Prager model), additional logic is used to handle all the possible combinations of behavior available within the model.

Occasionally ABAQUS fails to find a solution to the integrated constitutive equations. It then gives a warning message: “PLASTICITY ALGORITHM FAILS TO CONVERGE AT n INTEGRATION POINTS.” If automatic time incrementation has been chosen (which is always recommended), the program then tries again with a smaller time increment and the problem is, thereby, resolved. If necessary, detailed information about where the algorithms are failing can be obtained by setting *PRINT, PLASTICITY=YES.

ABAQUS provides consistent material Jacobians (tangent stiffnesses) for the global equilibrium iterations so that quadratic convergence can be achieved.
Lecture 4
Analysis of Porous Media

Overview

- Basic Assumptions and Effective Stress
- Stress Equilibrium and Flow Continuity
- Types of Analyses and Usage
- Examples
ABAQUS has capabilities for the treatment of single phase flow through porous media, including fully saturated flow (encountered in many geotechnical applications), partially saturated flow (encountered in irrigation problems and hydrology problems), or a combination of the two (calculation of phreatic surfaces).

Fluid gravity effects may be considered. It is possible to perform analyses in terms of total pore fluid pressure or excess pore fluid pressure.

Total pore pressure analysis (including fluid weight) is required when the loading provided by fluid gravity is large, or when “wicking” (transient capillary suction of liquid into a dry body) is of interest.

Two other effects, “gel” swelling and moisture swelling, may be included in partially saturated cases. These are usually associated with modeling of moisture absorption into polymeric systems (such as paper towels) rather than with geotechnical systems and are not discussed in these notes.
Basic Assumptions and Effective Stress

We model a deforming porous medium using the conventional approach that considers the medium as a multi-phase material and adopts an effective stress principle to describe its behavior.

An elementary volume, $dV$, is made up of a volume of grains of solid material, $dV_g$, and a volume of voids, $dV_v$, which is either fully or partly saturated with a volume of wetting fluid, $dV_w$. 

\[ dV = dV_g + dV_v + dV_w \]
The porosity of the medium, \( n \), is defined as the ratio of the volume of voids to the total volume:

\[
    n = \frac{dV_v}{dV}.
\]

ABAQUS generally uses voids ratio, \( e = (dV_v / dV_g) \), instead of porosity. Conversion relationships are:

\[
    e = \frac{n}{1 - n}, \quad n = \frac{e}{1 + e}, \quad 1 - n = \frac{1}{1 + e}.
\]

Saturation, \( s \), is defined as the ratio of wetting fluid volume to void volume:

\[
    s = \frac{dV_w}{dV_v}
\]

For a fully saturated medium \( s = 1 \), while for a completely dry medium \( s = 0 \).
The total stress acting at a point, \( \sigma \), is assumed to be made up of an average pressure stress in the wetting fluid, \( u_w \), called the “pore pressure” times a factor, \( \chi \), and an “effective stress” in the material skeleton, \( \bar{\sigma} \):

\[
\bar{\sigma} = \sigma + \chi u_w I.
\]

In general, \( \chi = \chi(s) \) can be measured experimentally. Typical experimental data are shown below:

Since such data are difficult to obtain, ABAQUS assumes \( \chi = s \).
The effective stress principle is the assumption that the constitutive response of the porous medium consists of simple bulk elasticity relationships for the fluid and for the solid grains, together with a constitutive theory for the material skeleton whereby $\bar{\sigma}$ is defined as a function of the strain history and temperature of the material:

$$\bar{\sigma} = \bar{\sigma}(\text{strain history, temperature, state variables}).$$
Any of the constitutive models in ABAQUS can be used to model the material skeleton of voided materials. Models suitable for soils and rocks were discussed in the previous lectures.

The strain rate decomposition is then

\[ d\varepsilon = (d\varepsilon^\text{vol}_g + d\varepsilon^\text{vol}_w)I + d\varepsilon^\text{el} + d\varepsilon^\text{pl} \]

where \( d\varepsilon^\text{vol}_g \), \( d\varepsilon^\text{vol}_w \) are the volume strain rates in the solid grains and fluid, and \( d\varepsilon^\text{el} \), \( d\varepsilon^\text{pl} \) are the elastic and plastic strain rates in the material skeleton.
Stress Equilibrium and Flow Continuity

Stress equilibrium for the solid phase of the material is expressed by writing the principle of virtual work for the volume under consideration in its current configuration at time \( t \):

\[
\int (\bar{\sigma} - \chi u_w \mathbf{I}) : \delta \varepsilon dV = \int_{S} t \cdot \delta v dS + \int_{V} f \cdot \delta v dV + \int_{V} sn \ \rho_w g \cdot \delta v dV,
\]

where \( \delta \varepsilon = \text{sym}(\partial \delta \mathbf{v} / \partial \mathbf{x}) \) is the virtual rate of deformation, \( \bar{\sigma} \) is the true (Cauchy) effective stress, \( \delta \mathbf{v} \) is a virtual velocity field, \( t \) are surface tractions per unit area, \( f \) are body forces (excluding fluid weight) per unit volume, \( \rho_w \) is the density of the fluid, and \( g \) is the gravitational acceleration (assumed constant and in a fixed direction).

This equation is then discretized using a Lagrangian formulation for the solid phase, with displacements as the nodal variables.
The porous medium is thus modeled by attaching the finite element mesh to the solid phase. Fluid may flow through this mesh.

A continuity equation is therefore required for the fluid, equating the rate of increase in fluid volume stored at a point to the rate of volume of fluid flowing into the point within the time increment:

\[
\frac{d}{dt} \left( \int_V \frac{\rho_w}{\rho_0^w} s n \, dV \right) = -\int_S \frac{\rho_w}{\rho_0^w} s n \cdot v_w \, dS
\]

where \( v_w \) is the average velocity of the fluid relative to the solid phase (the seepage velocity) and \( n \) is the outward normal to \( S \). This equation has been normalized by \( \rho_0^w \), the reference density of the fluid.

The continuity equation is integrated in time using the backward Euler approximation and discretized with finite elements using pore pressure as the variable.
The pore fluid flow behavior is assumed to be governed either by Darcy's law or by Forchheimer's law. Darcy's law is generally applicable to low fluid flow velocities, whereas Forchheimer's law is used for higher flow velocities. Darcy's law may be thought of as a linearized version of Forchheimer's law.

Forchheimer's law describes pore fluid flow as:

\[ v_w = -\frac{1}{sng \rho_w (1 + \beta \sqrt{v_w \cdot v_w})} \hat{k} \cdot \left( \frac{\partial u_w}{\partial x} - \rho_w g \right), \]

where \( g \) is the magnitude of the gravitational acceleration, \( \hat{k}(s, e) \) is the permeability of the medium (possibly anisotropic) with units of length/time, and \( \beta(e) \) is a “velocity coefficient”. [Some texts use a different definition of permeability. This is discussed later.]

Darcy's law is obtained by setting \( \beta = 0 \). We see that, as the fluid velocity tends to zero, Forchheimer's law approaches Darcy's law.
The permeability depends on the saturation of the fluid and on the porosity of the medium. We assume these dependencies are separable, so that

\[ \hat{k} = k_s k \]

where \( k_s(s) \) gives the saturation dependency, with \( k_s(1) = 1.0 \), and \( k(e) \) is the fully saturated permeability. For isotropic materials \( k = kI \).
Experimental observation often suggests that, in steady flow through a partially saturated medium, the permeability varies with $s^3$.

We, therefore, take $k_s = s^3$ by default. Different forms of behavior for $k_s(s)$ can be defined by using the *PERMEABILITY option.

Because $u_w$ measures pressure in the wetting fluid, the medium is fully saturated for $u_w > 0$. Negative values of $u_w$ represent capillary effects in the medium.
For $u_w < 0$ it is known that, at a given value of capillary pressure, $-u_w$, the saturation lies within certain limits. These limits are defined by using the *SORPTION option. Typical forms are shown below.
We write these limits as \( s^a \leq s \leq s^e \), where \( s^a(u_w) \) is the limit at which absorption will occur (so that \( \dot{s} > 0 \)) and \( s^e(u_w) \) is the limit at which exsorption will occur; thus, \( \dot{s} < 0 \). We assume that these relationships are uniquely invertible and can also be written as \( u_w^a(s) \) during absorption and \( u_w^e(s) \) during exsorption. We also assume that some wetting fluid will always be present in the medium: \( s > 0 \).

The transition between absorption and exsorption, and vice-versa, takes place along “scanning” lines, which are approximated by a single value of \( du_w/ds \) at all saturation levels.

Saturation is treated as a state variable that may have to change if the wetting liquid pressure is outside the range for which its value is admissible according to the actual data.
Note on permeability units:
In ABAQUS we define permeability in the flow constitutive equation

\[ v_w = - \frac{1}{s n g \rho_w (1 + \beta \sqrt{v_w \cdot v_w})} \hat{k} \cdot \left( \frac{\partial u_w}{\partial x} - \rho_w g \right) \]

as \( \hat{k} \), with units of length/time. In this equation \( \rho_w \) is the mass density of the fluid, \( s \) is the saturation, \( n \) is the porosity, \( \beta \) is the velocity coefficient and \( g \) is the gravitational acceleration. It is then clear that both sides of the equation have units of length/time.

However, one other definition of permeability (\( K \)) is often used:

\[ K = \frac{\mu}{g \rho_w} \hat{k} \]

where \( \mu \) is the fluid viscosity in poise units (mass/time-length). In this context, permeability \( K \) has length squared units (or Darcy) and what we refer to in ABAQUS as permeability, \( \hat{k} \), is called the hydraulic conductivity.
In the coupled problem, the stress equilibrium and fluid continuity equations must be solved simultaneously. In the general nonlinear case, we use a Newton scheme to solve the equations. The Newton equations for various cases of the formulation (transient or steady-state flow, etc.) are described in Appendix A.
The analysis of flow through porous media in ABAQUS is available for plane strain, axisymmetric, and three-dimensional problems. Special coupled displacement/pore pressure elements must be used: these elements have a linear distribution of pore pressure and either a first-order or a second-order distribution of displacement.

The coupled stress/flow problems are solved using the *SOILS procedure. By default, ABAQUS will solve the steady-state problem (*SOILS, STEADY STATE) while the transient problem is invoked with *SOILS, CONSOLIDATION.

The steady-state problem assumes that the time scale is so long that there is no transient effect in the pore fluid diffusion part of the problem. The time scale chosen is then only relevant to any possible rate effects in the constitutive model used for the material skeleton.

Mechanical loads and boundary conditions can be changed gradually over a step, to accommodate nonlinearities in the response.
Uncoupled (purely diffusion) pore pressure elements are not available. These are useful in cases when only the pore fluid flow part of the problem is of interest. In such problems we have used coupled elements and constrained all the displacement degrees of freedom to be computationally inefficient. Coupled deformation/pore pressure infinite elements are not available.

Dynamic coupled stress/fluid flow analysis cannot be performed, since we assume no inertia in the existing coupled analysis capability. This is important in cases such as the behavior of a dam subjected to earthquake loading. Again, liquefaction effects may be important.

Three-way coupled stress/fluid flow/temperature analysis cannot be performed. This is important in applications such as those encountered in nuclear waste disposal and oil reservoir simulation.

The capability for fluid flow through porous media assumes single-phase fluid flow; multi-phase fluid flow is important in cases such as oil reservoir simulation.

The vapor phase is ignored in the partially saturated flow formulation; there are situations for which this assumption is not adequate.
The transient problem includes the time integration of the diffusion effects; therefore, the choice of time increment is important. The integration procedure used in ABAQUS introduces a relationship between the minimum usable time increment and the element size. This minimum is a requirement only for second-order elements, but it is recommended for all diffusion elements.

A simple guideline that can be used for fully saturated flow is

\[ \Delta t > \frac{\rho_w g}{6E_k} \left(1 - \frac{E}{K_g}\right)^2 (\Delta l)^2, \]

where \( \Delta t \) is the time increment, \( E \) is the Young's modulus of the material skeleton, \( k \) is the permeability of the saturated medium (in units of length/time), \( K_g \) is the bulk modulus of the solid grains, and \( \Delta l \) is a typical element dimension.
A simple guideline that can be used for partially saturated flow is

$$\Delta t > \frac{\rho_w g n^0}{6 k_s k_s} \frac{ds}{du_w} (\Delta l)^2,$$

where $n^0$ is the initial porosity of the material, $k_s$ is the permeability-saturation relationship, and $ds/du_w$ is the rate of change of saturation with respect to pore pressure as defined in the *SORPTION material option.

If time increments smaller than this value are used, spurious oscillations may appear in the solution. If the problem requires analysis with smaller time increments than the above relationship allows, a finer mesh is required.

Generally there is no upper limit on the time increment size, except accuracy, since the integration procedure is unconditionally stable, unless nonlinearities cause numerical problems.
The accuracy of the time integration is controlled by the tolerance UTOL (maximum allowable change in pore pressure during the increment), which is also used to drive the automatic incrementation procedure for *SOILS, CONSOLIDATION analysis.

Transient analysis may be terminated by completing a specified time period, or it may be continued until steady-state conditions are reached, steady state being defined by all fluid pressures changing at less than a user-defined rate.

The porous medium coupled analysis capability can provide solutions either in terms of total or of “excess” pore fluid pressure.

The difference between total and excess pressure is relevant only for cases in which gravitational loading is important.

Total pressure solutions are provided when the GRAV distributed load type is used to define the gravity load on the model. Excess pressure solutions are provided in all other cases (for example, when gravity loading is defined with distributed load types BX, BY, or BZ).
In total pore pressure problems, the *DENSITY material option must be used to specify the density of the dry material only.

The gravity contribution from the pore fluid is defined through the SPECIFIC (weight) parameter on the *PERMEABILITY option, together with the direction of the gravity vector specified in the *DLOAD option with load type GRAV.

The *PERMEABILITY option is used to specify the permeability of the saturated medium, which can be isotropic or anisotropic and a function of void ratio.

The *PERMEABILITY option can be repeated with TYPE=VELOCITY to invoke Forchheimer's flow law instead of the default Darcy's law.

In partially saturated cases, the *PERMEABILITY option can also be repeated with TYPE=SATURATION to specify the dependence of permeability on saturation, \( k_s(s) \). By default, \( k_s = s^3 \).
In partially saturated cases the *SORPTION option is used to specify the dependence of negative (partially saturated) pore pressure on saturation. When used with TYPE=ABSORPTION (default), it defines the absorption curve. When used with TYPE=EXSORPTION, it defines the exsorption curve (by default, the exsorption curve is the same as absorption). Analytical (logarithmic) or tabular input data are permitted.

In partially saturated cases the *SORPTION, TYPE=SCANNING option is used to define the scanning slope between absorption and exsorption. ABAQUS will generate this slope automatically if the user does not specify it.

The *POROUS BULK MODULI material option can be used to specify the bulk modulus of the solid grains and fluid if the user wants to include the compressibility of these components of the porous medium.

*EXPANSION can be used to introduce thermal volume change effects for the solid grains and the pore fluid.
*INITIAL CONDITIONS, TYPE=RATIO is required to define the initial voids ratio (porosity) of the medium. User subroutine VOIDRI can be used to specify complex initial void ratio distributions.

*INITIAL CONDITIONS, TYPE=STRESS can be used to define the initial effective stress state in the material. User subroutine SIGINI can be used to specify complex initial effective stress distributions.
*INITIAL CONDITIONS, TYPE=PORE PRESSURE can be used to define the initial pore fluid pressure in the medium. The initial pore pressures can be defined as a linear function of elevation in the model or as a constant value. ABAQUS assumes that the vertical (elevation) direction is the 3-direction in three-dimensional models and is the 2-direction in two-dimensional (or axisymmetric) models.

User subroutine \texttt{UPOREP} is also available for complicated cases. To use this subroutine, the USER parameter must be included on the *INITIAL CONDITIONS, TYPE=PORE PRESSURE option. The user will be given the coordinates of each node in the user subroutine.

The specification of initial conditions in coupled stress partially saturated flow problems with gravity is not always trivial (see \textit{Element Technology} (page L5.2) for more information).
The *DFLOW option allows the outward normal flow velocity, $v_n$, to be prescribed across a surface. Complex dependencies of $v_n$ on time and position can be coded in user subroutine DFLOW.

The *FLOW option defines the outward flow velocity as

$$v_n = k_s (u_w - u_w^\infty),$$

where $k_s$ and $u_w^\infty$ are known values. Again, complex conditions can be coded in user subroutine FLOW.
If large-deformation analysis is required because of the presence of large strains, the NLGEOM parameter can be included on the *STEP option. The UNSYMM=YES parameter on the *STEP option is used automatically if the user requests steady-state analysis or any kind of partially saturated flow analysis. ABAQUS automatically uses UNSYMM=YES when fluid gravity effects are included in a step. For other unsymmetric simulations—such as using NLGEOM, nonlinear permeability, or nonassociated plastic flow—using UNSYMM=YES may improve the rate of convergence.
[Two other saturation-dependent effects can be included in partially saturated flow analysis:

The *MOISTURE SWELLING material option can be used to define saturation driven volumetric swelling (or shrinkage as negative swelling) of the solid skeleton. In this option, the reversible swelling strain is defined as a function of saturation. Anisotropic swelling can be specified by using the *RATIOS option:

\[
\varepsilon_{ii}^{ms} = r_{ii} \frac{1}{3} \left( \varepsilon_{ms}^m(s) - \varepsilon_{ms}^m(s^I) \right).
\]

The *GEL material option can be used to define the growth of gel particles that swell and trap fluid. The growth of the gel particles depends on the saturation of the wetting fluid, the size of the gel particles, and their number per unit of volume of porous material.

An example of the use of these options is given in **Dry Problems** (page L6.3) of these notes.]
Examples

Fully Saturated Example

We consider the one-dimensional Terzaghi consolidation problem. The problem is treated with and without finite-strain effects for illustration.

A body of soil is confined by impermeable, smooth, rigid walls on all but the top surface where perfect drainage is possible, and a load is applied suddenly. Gravity is neglected.

We wish to predict the response of the soil as a function of time, following the load application.

The properties of the soil are described in Benchmark Problem 1.14.1. The problem is run in two steps. The first step is a *SOILS, CONSOLIDATION analysis with an arbitrary time period, with no drainage allowed across the top surface (natural boundary condition).
This establishes the initial solution: uniform pore pressure equal to the load throughout the body, with no effective stress carried by the soil skeleton.

The consolidation is now done with a second *SOILS, CONSOLIDATION step, using automatic time incrementation. The accuracy of the time integration for the second step is controlled by the parameter UTOL.

The spatial element size and the time increment size are related, to the extent that time increments smaller than a certain size give no useful information. This coupling of the spatial and temporal approximations is most obvious at the start of diffusion problems, immediately after prescribed changes in the boundary values.

Using the time-space criterion described earlier, an initial time increment of .06 sec (0.001 min) is chosen. This gives an initial solution with no “overshoot.”
Examples

Analysis of Geotechnical Problems with ABAQUS

Terzaghi Problem

Uniform load, \( q = 689.5 \text{ MPa} (1.0 \times 10^8 \text{ lb/in}^2) \)

Perfectly drained

Soil

\( h = 2.54 \text{ m} (100.0 \text{ in}) \)

Impermeable, smooth and rigid

Impermeable and rigid
Examples

Analysis of Geotechnical Problems with ABAQUS

L4.32

"Overshooting"
Pore Pressure History

- Time = 0.06 s (10^{-3} min)
- Time = 0.6 s (10^{-2} min)
- Time = 27.72 s (0.462 min)

Normalized pore pressure, $p/q$ vs. Elevation, $z/h$
Effective Stress History

Vertical component of the effective (compressive) stress

Pore pressure

Stress, normalized by the overburden

Time, minutes

Time, s

0 0.2 0.4 0.6 0.8 1.0

10 20 30 40 50 60

0.1 0.2 0.3 0.4 0.5 0.6 0.7 0.8 0.9 1.0

Effective Stress History
Consolidation History

Degree of consolidation, %

Time factor

Terzaghi and Frölich

ABAQUS solution
Examples

Analysis of Geotechnical Problems with ABAQUS

Finite strain, permeability dependent on voids ratio

Finite strain, constant permeability

Small strain

Vertical displacement at surface, d/h

Time, minutes

Consolidation History
Input Listing:

*HEADING
   TERZAGHI CONSOLIDATION
*ELEMENT, TYPE=CPE8P, ELSET=ONE
  1, 1, 3, 23, 21, 2, 13, 22, 11
*ELGEN, ELSET=ALL
  1, 10, 20
*ELSET, ELSET=P1
  6, 7, 8
*NODE
  1,
  2, 25.
  3, 50.
  201,, 100.
  202, 25,, 100.
  203, 50,, 100.
*NGEN, NSET=NALL
  1, 201, 10
  2, 202, 20
  3, 203, 10
Examples

- *NSET,NSET=FILE 141,
- *NSET,NSET=TOP 201,202,203
- *NSET,NSET=BASE 1,2,3
- *SOLID SECTION,MATERIAL=A1,ELSET=ALL
- *MATERIAL,NAME=A1
- *ELASTIC 1.E8,.3
- *PERMEABILITY .0002
- *INITIAL CONDITIONS,TYPE=RATIO NALL,1.1,0.,1.1,1.
- *BOUNDARY BASE,1,2 NALL,1
- *RESTART,WRITE,FREQUENCY=999
- *STEP
  - SUDDENLY APPLIED LOAD
- *SOILS,CONSOLIDATION 1.E-7,1.E-7
*DLOAD
10,P3,1.E8
*PRINT,RESIDUAL=NO
*EL PRINT,FREQUENCY=999
COORD
S
*EL PRINT,ELSET=P1,FREQUENCY=10
S
*NODE PRINT,FREQUENCY=5
U,POR,RVT
*END STEP
*STEP,INC=100
  CONSOLIDATE
*SOILS,CONSOLIDATION,UTOL=5.E7,END=SS
  .001,100.,.001,100.,100.
*BOUNDARY
TOP,8
*NODE FILE,NSET=FILE
U,POR
*EL FILE,ELSET=ONE
S
*END STEP
Partially Saturated Example

We consider a one-dimensional “wicking” test where the absorption of fluid takes place against the gravity load caused by the weight of the fluid.

In such a test fluid is made available to the material at the bottom of a column and the material absorbs as much fluid as the weight of the rising fluid permits.

The material properties and initial conditions are described in Section 1.8.3 of the ABAQUS Benchmarks Manual.

The weight is applied by GRAV loading.

An initial step of *GEOSTATIC analysis is performed to establish the initial equilibrium state; the initial conditions in the column exactly balance the weight of the fluid and dry material so that no deformation or fluid flow takes place.
The bottom of the column is then exposed to fluid by prescribing zero pore pressure (corresponding to full saturation) at those nodes during a transient *SOILS, CONSOLIDATION step.

The fluid will seep up the column until the pore pressure gradient is equal to the weight of the fluid, at which time equilibrium is established.

At steady state the pore pressure gradient must equal the weight of the fluid so that pore pressure varies linearly with height and saturation varies in the same way (according to the absorption behavior) with respect to pressure or height.

Thus, points close to the bottom of the column are fully saturated, while those at the top are still at 5% saturation. This is illustrated in the last figure, which is nothing more than the absorption curve.
Examples

Analysis of Geotechnical Problems with ABAQUS

Wicking Model
Examples

Analysis of Geotechnical Problems with ABAQUS

Initial Conditions

- pore pressure, Pa
- initial conditions
- exsorption
- scanning
- absorption
Fluid Volume Absorbed

TIME (sec) (*10**4) VS. FLUID VOLUME (m**3) (*10**-2)

LINE VARIABLE        SCALE
FACTOR
1                    +1.00E+00

Fluid Volume Absorbed
Pore Pressures
Saturation Histories
Steady-State Profile
Input Listing:

*HEADING
ONE DIMENSIONAL WICKING PROBLEM, COUPLED
*** UNITS: M, TON, SEC, KN
*NODE,NSET=ALLN
1,0.,0.
3,.1,0.
101,0.,1.
103,.1,1.
*NGEN,NSET=BOT
1,3,1
*NGEN,NSET=TOP
101,103,1
*NFILL,NSET=ALLN
BOT,TOP,20,5
*NSET,NSET=LHS,G U N E R A T E
1,101,5
*NSET,NSET=RHS,G U N E R A T E
3,103,5
*NSET,NSET=POREP,G U N E R A T E
1, 101, 10
3, 103, 10
*ELEMENT, TYPE=CPE8RP, ELSET=BLOCK
1, 1, 3, 13, 11, 2, 8, 12, 6
*ELGEN, ELSET=BLOCK
1, 10, 10, 1
*ELSET, ELSET=OUTE
1, 3, 5, 7, 9
*SOLID SECTION, ELSET=BLOCK, MATERIAL=CORE
*MATERIAL, NAME=CORE
*ELASTIC
50., 0.
*DENSITY
.1
*POROUS BULK MODULI
, 2.E6
*PERMEABILITY, SPECIFIC=10.
3.7E-4
*SORPTION
-100., .04
-10., .05
-4.5, .1
-3.5, .18
-2., 0.45
-1., 0.91
0., 1.
*SORPTION, TYPE=EXSORPTION
-100., 0.09
-10., 1
-8., 11
-6., 18
-4.5, 0.33
-3., 0.79
-2., 0.91
0., 1.
*INITIAL CONDITIONS, TYPE=SATURATION
ALLN, 0.05
*INITIAL CONDITIONS, TYPE=PORE PRESSURE
POREP, -22.0, 1.0, -12.0, 0.0
*INITIAL CONDITIONS, TYPE=RATIO
ALLN, 5.
*INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
BLOCK, -1.1, 1., -2.016666667, 0., 0., 0.
*EQUATION
2
3,8,1.,1,8,-1.
*BOUNDARY
ALLN,1
BOT,2
*RESTART,WRITE,FREQUENCY=10
*STEP,INC=1
*GEOSTATIC
1.E-6,1.E-6
*BOUNDARY
1,8,,-12.
*DLOAD
BLOCK,GRAV,10.,0.,-1.,0.
*NODE PRINT,FREQUENCY=5,NSET=LHS
U,RF,POR,RVT
*EL PRINT,FREQUENCY=5,POSITION=AVERAGED AT NODES
S,E
SAT,POR,VOIDR
*NODE FILE,FREQUENCY=10,NSET=LHS
U,RF,POR,RVT
*EL FILE,FREQUENCY=10,ELSET=OUTE
S,E
SAT,POR,VOIDR
*END STEP
*STEP,INC=100
*SOILS,CONSOLIDATION,UTOL=20.
1.,1000000.
*BOUNDARY
1,8,,0.
*CONTROLS,ANALYSIS=DISCONTINUOUS
*CONTROLS,PARAMETERS=FIELD,FIELD=DISPLACEMENT
,1.,
*CONTROLS,PARAMETERS=FIELD,FIELD=PORE_FLUID_PRESSURE
,1.,
*END_STEP
Lecture 5
Modeling Aspects

Overview

- Element Technology
- Infinite Domains
- Pore Fluid Surface Interactions
- Element Addition and Removal

Modeling issues related to geotechnical problems are considered in this lecture.
Element Technology

The geotechnical constitutive models can be used in plane strain, generalized plane strain, axisymmetry, and three dimensions. All Drucker-Prager models are also available in plane stress, except for the linear Drucker-Prager model with creep.

Cylindrical (CCL) elements are available for modeling structures that are initially circular but are subjected to general, nonaxisymmetric loading. An example is the analysis of a pile foundation where the pile is subjected to axial, horizontal and moment loading.

These elements permit a coarse yet accurate discretization of a structure.

They can be used in contact calculations using the standard surface-based contact modeling approach.

They provide an attractive alternative to axisymmetric-asymmetric (CAXA_xyN) family of elements.
The analysis of flow through porous media in ABAQUS is available for plane strain, axisymmetric, axisymmetric-asymmetric, and three-dimensional problems. Special coupled displacement/pore pressure elements must be used: these elements have a linear distribution of pore pressure and either a first-order or a second-order distribution of displacement.

The modified tetrahedral element C3D10MP(H) is particularly well suited for meshing general, complex structures in three dimensions. The element works well in contact interactions.

The geotechnical material models can be used in time integration dynamic analysis. Eigenfrequencies of undamped models can also be extracted before or after static deformation.
Infinite Domains

Infinite elements for stress analysis are available in ABAQUS; these are used in conjunction with the standard elements in problems involving infinite or very large domains.

A family of first- and second-order axisymmetric, planar, and three-dimensional infinite elements is available.
Standard finite elements are used to model the region of interest, with the infinite elements modeling the far-field region.
The solution in the far field is assumed to be linear, so only linear behavior is provided in the infinite elements.
The static behavior of the infinite elements is based on modeling the basic solution variable, $u$ (in stress analysis $u$ is a displacement component), with respect to spatial distance $r$ measured from a “pole” of the solution, so that $u \to 0$ as $r \to \infty$, and $u \to \infty$ as $r \to 0$. The interpolation provides terms of order $1/r$, $1/r^2$. The far-field behavior of many common cases, such as a point load on a half-space, is thereby included.
Infinite Domains

Analysis of Geotechnical Problems with ABAQUS

3/03

CAX8R

CINAX5R
It is important to make an appropriate choice of the position of the nodes in the infinite direction with respect to the origin ("pole") of the far-field solution.

For example, the solution for a point load applied to the boundary of a half-space has its pole at the point of application of the load.

The second node along each infinite element edge pointing in the infinite direction must be positioned so that it is twice as far from the pole as the node on the same edge at the boundary between the finite and the infinite elements.

In addition, be careful when specifying the second nodes in the infinite direction so that the element edges in the infinite direction do not cross over.
The *NCOPY, POLE option provides a convenient way of defining these second nodes in the infinite direction.
In plane stress and plane strain problems in which the loading is not self-equilibrating, the far-field displacement solution is typically of the form \( u = \ln(r) \). This implies the displacement approaches infinity as \( r \to \infty \).

Infinite elements can still be used for such cases, provided the displacement results are treated as having an arbitrary reference value. Thus, strain, stress, and relative displacements within the finite element part of the model will converge to unique values as the model is refined; the total displacements will depend on the size of the region modeled with finite elements.

If the loading is self-equilibrating, the total displacements will also converge on a unique solution.
In many geotechnical problems, an initial stress field and a corresponding body force field must be defined.

For standard elements, the initial stress field is given in *INITIAL CONDITIONS, TYPE=STRESS, and the corresponding body force in the *DLOAD option. ABAQUS checks for equilibrium in the initial state (*GEOSTATIC step) at the start of the analysis.

For infinite elements, the body force cannot be defined (the elements are infinite). Therefore, ABAQUS automatically inserts forces at the nodes of the infinite elements that cause those nodes to be in equilibrium at the start of the analysis. These forces remain constant throughout the analysis.

This allows the initial geostatic stress field to be defined in the infinite elements but provides no check on the reasonableness of that stress field. The user must ensure that, when infinite elements are used in conjunction with an initial stress condition, the first analysis step must be a *GEOSTATIC step.
The Boussinesq (point load on a half-space) and Flamant (line load on a half-space) problems:
Infinite Domains

Analysis of Geotechnical Problems with ABAQUS

Boussinesq Problem – Displacement Results (Benchmark Problem 2.2.2)
Flamant Problem – Displacement Results (Benchmark Problem 2.2.2)
Input Listing for Boussinesq Problem:

*HEADING
BOUSSINESQ PROBLEM, 12 CAX4 + 4 CINAX4
*NODE
1, 0., 0.
4, 0., -1.
8, 1., 0.
6, .75, -.75
21, 0., -2.
25, 0., -4.
101, 2., 0.
105, 4., 0.
61, 1.5, -1.5
65, 2.833333333, -2.833333333
*NGEN
21, 25, 2
61, 65, 2
101, 105, 2
21, 61, 20
23, 63, 20
25, 65, 20
61,101,20
63,103,20
65,105,20
*NSET,NSET=INTER,Generate
25,105,20
*NCOPY,OLD SET=INTER,CHANGE NUMBER=2,POLE,NEW SET=FAR
1
*NSET,NSET=LHS,Generate
1,4,3
21,27,2
*Eлемент,TYPE=CAX4,ELSET=ALL
1,4,6,8,1
2,21,41,6,4
3,41,61,81,6
4,81,101,8,6
5,23,43,41,21
*ELGEN,ELSET=ALL
5,4,20,1,2,2,4
*Eлемент,TYPE=CINAX4,ELSET=ALL
13,45,25,27,47
14,65,45,47,67
15,85,65,67,87
16,105,85,87,107
*SOLID SECTION,ELSET=ALL,MATERIAL=ONE
*MATERIAL,NAME=ONE
*ELASTIC
  1.,.1
*BOUNDARY
LHS,1
*STEP
  *STATIC
  *CLOAD
  1,2,-1.
  *NODE FILE,NSET=LHS
U
*END STEP
In direct integration dynamic response analysis (*DYNAMIC) and in *STEADY STATE DYNAMICS, DIRECT frequency domain analysis, the elements provide “quiet” boundaries to the finite element model.

This means that they maintain the static force that was present at the start of the dynamic response analysis on the finite/infinite boundary.

As a consequence the far-field nodes in the infinite elements will not displace during the dynamic response (there is no dynamic response within the infinite elements).

The infinite elements will provide additional normal and shear tractions on the boundary, proportional to the normal and shear components of the velocity of the boundary.
The concept is simple. Consider one-dimensional wave propagation down the $x$-axis.

Equilibrium is

$$-\rho \ddot{u} + \frac{d\sigma}{dx} = 0.$$ 

The constitutive behavior is assumed to be linear elastic, and we also assume small deformation:

$$\sigma = E\varepsilon = E\frac{du}{dx}.$$ 

Combining,

$$-\rho \ddot{u} + E\frac{d^2u}{dx^2} = 0.$$
The general solution to this wave equation has the form

\[ u = f(x \pm ct), \]

where \( c = \sqrt{\frac{E}{\rho}} \) is the wave speed and \( f \) is any function. A wave traveling to the right (\( x \) increasing) has the form \( u = f_1(x - ct) \); one traveling to the left is \( f_2(x + ct) \).

Suppose we have a boundary to the right of the domain:

\[ \overline{\text{boundary}} \quad x \]

If an incident wave, \( u_I = f_I(x - ct) \) approaches this boundary, we want no reflection, \( u_R = f_R(x + ct) \), to occur.
For this purpose we introduce a dashpot at the boundary:

\[
\sigma = -d\dot{u}.
\]

so that, at the boundary, \( \sigma = -d\dot{u} \).

For both \( u_I = f_I(x - ct) \) and \( u_R = f_R(x + ct) \), at the boundary

\[
\sigma = E(f_I' + f_R') \text{ from elasticity}
\]

\[
= d(-cf_I' + cf_R') \text{ for the dashpot.}
\]

Thus, \( (E - dc)f_I' + (E + dc)f_R' = 0 \). But, we want \( f_R = 0 \), so \( f_R' = 0 \).

This is always achieved if we choose \( d = \frac{E}{c} = \rho c \).
This boundary damping is thus chosen to eliminate the reflection of wave energy back into the finite element mesh when plane waves cross the plane boundary:
Lysmer and Kuhlemeyer (1969) generalized this for 3-D cases, with damping of the normal velocity of the boundary:

\[ d_p = \rho c_p = \rho \sqrt{\frac{\lambda + 2G}{\rho}}, \]

and damping of the shear velocity:

\[ d_s = \rho c_s = \rho \sqrt{\frac{G}{\rho}}, \]

where \( c_p, c_s \) are the dilatational (pressure) and shear wave speeds, \( \rho \) is the mass density of the material, and \( \lambda, G \) are Lamé's constants:

\[ \lambda = \frac{E\nu}{(1 + \nu)(1 - 2\nu)}, \quad G = \frac{E}{2(1 + \nu)}. \]

This approach assumes that the material behavior close to the finite/infinite boundary is linear elastic (which is reasonable since the infinite elements are also assumed to be elastic).
Since, during dynamic analysis, the elements hold the static stress on the boundary constant but do not provide any stiffness, some rigid body motion of the region modeled will generally occur. This effect is usually small.

The elements are based on eliminating energy transmission for plane waves crossing a parallel plane boundary. Therefore, the ability of the elements to transmit energy out of the finite element mesh without trapping or reflecting it is optimized by making the finite/infinite boundary as close as possible to being orthogonal to the direction from which the waves will impinge on this boundary, and far enough from the detailed part of the mesh to be considered relatively plane.

Close to a free surface where Rayleigh waves may be important or to a material interface where Love waves may be important, the elements are most effective if they are orthogonal to this surface.

The elements do not provide any contribution to eigenmode-based analysis procedures.

A dynamic example is presented in **Dry Problems** (page L 6.3).
Pore Fluid Surface Interactions

The standard contact options offered in ABAQUS for stress analysis can be used for geotechnical applications. Their theory and usage is discussed in the Contact lecture notes.

The flow pattern is shown below:

Two configurations are possible:

1. Pore pressure continuum elements on either side of the interface.
2. Pore pressure elements on one side and “regular” elements on the other. This models fluid interaction with an impermeable surface.
The pore pressure is assumed to be continuous across the interface, regardless of whether the element is open or closed.

The contact condition is based on effective stress. Hence, contact points may open as the pore pressure increases (in a case where the total stress is constant and the effective stress decreases due to the pore pressure increase).

It is assumed that no fluid flow takes place tangential to the contact surface. In steady-state analysis, this implies that fluid that flows out of one side flows into the other side. In transient analysis, the flow into the interface is balanced with the rate of separation of the sides of the interface.

In a consolidation analysis fluid volume between the surfaces is considered when balancing the flow from each surface. The fluid in the interface is assumed incompressible.
Geostatic States of Stress

In most geotechnical problems, a nonzero state of stress exists in the medium. This typically consists of a vertical stress increasing linearly with depth, equilibrated by the weight of the material, and horizontal stresses caused by tectonic effects.

The active loading is applied on this initial stress state. Active loading could be the load on a foundation or the removal of material during an excavation.

It is clear that, except for purely linear analysis, with a different initial stress state, the response of the system would be different.

This well illustrates a point of nonlinear analysis: the response of a system to external loading depends on the state of the system when that loading sequence begins (and, by extension, to the sequence of loading). We can no longer think of superposing load cases as we do in linear analysis.
ABAQUS provides the *GEOSTATIC procedure to allow the user to establish the initial stress state.

The user will normally specify the initial effective stresses using *INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC and in the first step of analysis, apply the body (gravity) loads corresponding to the weight of the material.

Ideally, the loads and initial stresses should exactly equilibrate and produce zero deformations. However, in complex problems, it may be difficult to specify initial stresses and loads that exactly equilibrate.

The *GEOSTATIC procedure is used to reestablish initial equilibrium if the loads and initial stresses specified are not in equilibrium. It will also produce deformations while doing this.

If the deformations produced are significant compared to the deformations caused by subsequent loading, the definition of the initial state should be reexamined.
In a coupled deformation/flow analysis the *GEOSTATIC procedure is equivalent to the steady-state *SOILS procedure. In these problems it is important to establish initial stress equilibrium as well as steady-state flow conditions.

In fully or partially saturated flow problems, the initial void ratio, as well as the initial pore pressure and the initial effective stress, must be defined.

The initial conditions discussion that follows is based on the total pore pressure formulation (the magnitude and direction of the gravitational loading are defined by using the GRAV *DLOAD option).

Let us assume that the \( z \)-axis points vertically upwards. We assume that, in the geostatic state, the pore fluid is in hydrostatic equilibrium, so that

\[
\frac{du_w}{dz} = -\gamma_w,
\]

where \( \gamma_w \) is the specific weight of the pore fluid.
If we also take \( \gamma_w \) to be independent of \( z \) (which is usually the case, since the fluid is almost incompressible), this equation can be integrated:

\[
\frac{d\sigma_{zz}}{dz} = \rho g + s n^0 \gamma_w,
\]

where \( \rho \) is the dry density of the porous solid material, \( g \) is the gravitational acceleration, \( n^0 \) is the initial porosity, and \( s \) is the saturation \((0 \leq s \leq 1.0)\).

The *INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC* option defines the initial value of the effective stress, \( \bar{\sigma} \), as

\[
\bar{\sigma} = \sigma + su I.
\]
Combining this definition with the equilibrium statement in the \( z \)-direction and hydrostatic equilibrium in the pore fluid gives

\[
\frac{d \bar{\sigma}_{zz}}{dz} = \rho g - \gamma_w \left( s(1 - n^0) - \frac{ds}{dz}(z_w^0 - z) \right),
\]

using the assumption that \( \gamma_w \) is independent of \( z \).

In many cases \( s \) is constant. For example, in fully saturated flow \( s = 1.0 \) everywhere. If we further assume that the initial porosity, \( n^0 \), and the dry density of the porous medium, \( \rho \), are also constant, the above equation is readily integrated to give

\[
\bar{\sigma}_{zz} = (\rho g - \gamma_w s)(1 - n^0)(z - z^0),
\]

where \( z^0 \) is the position of the surface of the medium.

In more complicated cases where \( s, n^0 \) and/or \( \rho \) vary with height, the equation must be integrated in the vertical direction to define the initial values of \( \bar{\sigma}_{zz}(z) \).
In partially saturated cases the initial pore pressure and saturation values must lie on or between the absorption and exsorption curves.

In many geotechnical applications there is also horizontal stress. If the pore fluid is under hydrostatic equilibrium and $\tau_{xz} = \tau_{yz} = 0$, equilibrium in the horizontal directions requires that the horizontal components of effective stress do not vary with horizontal position: $\bar{\sigma}_h(z)$ only, where $\bar{\sigma}_h$ is any horizontal component of effective stress.

The horizontal stress is typically assumed to be a fraction of the vertical stress: those fractions are defined in the $x$- and $y$-directions with the *INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC option. If the horizontal stress is nonzero the boundary conditions on any nonhorizontal edges of the finite element model must be fixed in the horizontal direction, or infinite elements used, so that horizontal equilibrium is maintained.
Element Addition and Removal

Practical geotechnical excavations involve a sequence of steps, in each of which some part of the material mass is removed. Liners or retaining walls may be inserted during this process. Similar situations arise in the case of building an embankment.

Thus, geotechnical problems offer an interesting perspective on the need for generality in creating and using a finite element model: the model itself, and not just its response, changes with time—parts of the original model disappear, while other components that were not originally present are added.

The need for a close liaison between the analysis and geometric modeling is important.

The *MODEL CHANGE, REMOVE, ADD option is used to allow the user to remove or add elements to the model.
While elements are inactive, any distributed loads, fluxes, flows, and foundations specified for them are also inactive. A record of these loads is still kept, and continuation of loads across steps is not affected by removal, so on reactivation these loads are still present, unless they are removed by the user. Concentrated loads or fluxes are not removed. Therefore, the user must ensure that the concentrated loads or fluxes, which are carried by elements being removed, are also removed; otherwise, a solver problem will occur (a force is applied to a degree of freedom with zero stiffness).
The nodal variables are not changed by the *MODEL CHANGE option. However, the user can reset these variables by using the *BOUNDARY option while the elements are inactive. For example, if some elements that are removed are to be reintroduced with a different displacement, an intermediate step can be used in which the displacements for the nodes on these elements are reset by a *BOUNDARY that is removed when the elements are reactivated.

Elements can be reactivated either

• with strain (e.g., when simulating the refueling of a nuclear reactor, where the new fuel assembly must conform to the distortion of its old neighbors)

  *MODEL CHANGE, ADD=WITH STRAIN

• without strain (e.g., when simulating the addition of a new, strain-free layer to a strained construction)

  *MODEL CHANGE, ADD=STRAIN FREE

Example Problem 1.1.10 discusses the technique further.
Lecture 6
Example Problems

Overview

• Dry Problems
  – Limit Analysis of Foundation
  – Slope Stability Problem
  – A Dynamic Analysis

• Saturated Problems
  – Consolidation Problem (Transient)
  – Dam Problem (Steady-State)
• Partially Saturated Problems
  – Demand Wettability Problem (Uncoupled)
  – Desaturation of Soil Column (Transient)
  – Phreatic Surface Calculation (Steady-State)
• Excavation and Building Analysis
  – Tunneling Problem
Dry Problems

We present problems that involve the analysis of dry media: we solve the stress deformation equations only.

Limit Analysis of Foundation

This example presents solutions to limit load calculations for a strip of sand loaded by a rigid, perfectly rough footing (Benchmark Problem 1.14.4). It compares the results obtained with different parameters used in the modified Drucker-Prager model in ABAQUS, with and without a cap, matched to the classical Mohr-Coulomb model.

We may want to match the Drucker-Prager model to Mohr-Coulomb data for various reasons:

• Creep
• Rate dependence
• Compatibility with ABAQUS/Explicit
A mesh of finite/infinite elements is used to model the problem. In ABAQUS the infinite elements are always assumed to have linear elastic behavior; and, therefore, they are used beyond the region where plastic deformation takes place.

The elasticity is assumed to be linear, with $E = 30 \times 10^3$ psi and $\nu = 0.3$. Yield is governed by the Mohr-Coulomb surface, with $\phi = 20^\circ$ and $c = 10$ psi.

In *Stress Invariants and Spaces* (page L3.2) we showed alternative methods of converting these parameters to the parameters of the modified Drucker-Prager model. In this example we show results for the two standard conversions.

Matching the response of the models in triaxial compression and tension provides $\beta = 37.67^\circ$, $K = 0.795$, and $\sigma_c^0 = 28.56$ psi. The example is run for these parameter values with associated flow ($\psi = \beta$) and non-dilatant flow ($\psi = 0$).
Matching the limit load response of the models for plane strain provides \( \beta = 30.16^\circ \) and \( \sigma_c^0 = 19.8 \) psi for associated flow; \( \beta = 30.64^\circ \) and \( \sigma_c^0 = 20.2 \) psi for non-dilatant flow. The plane strain matching assumes that \( K = 1 \). The example is run using the associated flow parameters together with \( \psi = \beta \) and using the non-dilatant flow parameters with \( \psi = 0 \).

The Drucker-Prager/Cap model is run using both the triaxial and the plane strain matching of the Mohr-Coulomb parameters. The additional material parameters required for the Cap model are adopted from Mizuno and Chen (1983).
The cap eccentricity parameter is chosen as $R = 0.1$, the initial cap position is taken as $\varepsilon_{\text{vol}(0)}^{pl} = 0.00041$, and the cap hardening curve is shown below. The transition surface parameter $\alpha = 0.01$ is used.
The load-displacement responses are shown and compared to the limit analysis (slip line) Prandtl and Terzaghi solutions.

In this case the plane strain matching of the Mohr-Coulomb parameters provides significantly better predictions of the limit load than the triaxial compression/tension matching of the Mohr-Coulomb parameters.

This is attributable to the plane strain matching providing the same definition of plastic flow direction as well as of failure for plane strain. The triaxial matching only matches failure stress values under triaxial conditions.

The non-dilatant Drucker-Prager models give a softer response and a lower limit load than the corresponding dilatant versions. The Cap model provides responses that are comparable to the corresponding Drucker-Prager non-dilatant responses.

This is due to the addition of the cap and the nonassociated flow in the failure region, which combine to reduce the dilation in the model and therefore approximate the Drucker-Prager non-dilatant flow model.
The closest comparisons to Mohr-Coulomb behavior are those obtained with the plane strain matching, non-dilatant, Drucker-Prager model, and the plane strain matching Cap model.

They provide almost identical limit loads, which fall between the Prandtl and Terzaghi solutions.

This conclusion can be extended to general geotechnical problems that are analyzed under plane strain or axisymmetric assumptions.

![Finite/Infinite Element Model](image-url)
Dry Problems

Analysis of Geotechnical Problems with ABAQUS

Drucker-Prager and Cap Limit Load Results

- Drucker-Prager, plane strain match, dilatant flow
- Drucker-Prager, plane strain match, non-dilatant flow
- Drucker-Prager, triaxial match, dilatant flow
- Drucker-Prager, triaxial match, non-dilatant flow
- Cap model, plane strain match
- Cap model, triaxial match

Terzaghi (175 lb/in²)
Prandtl (143 lb/in²)
Limit Analysis Problem File (Drucker-Prager):

*HEADING
   LIMIT LOAD STUDIES, DRUCKER PRAGER, PE, NON-DILATANT FLOW
*RESTART,WRITE,FREQUENCY=10
*NODE
1
7,60.
13,180.
15,228.
19,348.
801,,144.
807,60,,144.
813,180,,144.
815,228,,144.
819,348,,144.
20,696.
820,696,,144.
*NGEN,NSET=BASE
1,7
7,13
13,15
15,19
*NSET,NSET=F1
  801,
*NSET,NSET=F2,GENERATE
  802,807
*NGEN,NSET=CENTER
  1,801,100
*NGEN,NSET=TOP
  801,807
  807,813
  813,815
  815,819
*NFILL
BASE,TOP,8,100
*NGEN,NSET=FAR
  20,820,200
*ELEMENT,TYPE=CPE8R
  1,1,3,203,201,2,103,202,101
*ELGEN,ELSET=ALL
  1,4,200,1,9,2,10
*ELSET,ELSET=PRINTELS
  1,2,3,4
*SOLID SECTION, ELSET=ALL, MATERIAL=A1
*MATERIAL, NAME= A1
*ELASTIC
30000., 0.3
*DRUCKER PRAGER HARDENDING
20.2, 0.
*DRUCKER PRAGER, SHEAR CRITERION=LINEAR
30.64, 1.0, 0.
*ELEMENT, TYPE=CINPE5R
101, 219, 19, 20, 220, 119
*ELGEN, ELSET=FAR
101, 4, 200, 1
*SOLID SECTION, ELSET=FAR, MATERIAL=A2
*MATERIAL, NAME= A2
*ELASTIC
30000., 0.3
*EQUATION
2
F2, 2, 1., 801, 2, -1.
*BOUNDARY
CENTER, 1
F2, 1
BASE, 1, 2
*STEP, INC=50, UNSYMM=YES
    PRESCRIBE DISPLACEMENT
*STATIC
  .025, 1, ., .1
*BOUNDARY
  801, 2, ., -5.0
*MONITOR, NODE=801, DOF=2
*CONTROLS, ANALYSIS=DISCONTINUOUS
*EL PRINT, ELSET=PRINTELS, FREQUENCY=10
S, E
SINV
ENER
E, IE
PE
*NODE PRINT, FREQUENCY=5
U, RF
*NODE PRINT, NSET=F1
U, RF
*EL FILE, ELSET=PRINTELS, FREQUENCY=10
S
SINV
ENER
IE
*NODE FILE,NSET=F1
U,RF
*END STEP
Limit Analysis Problem File (Cap Model):

*HEADING
  LIMIT LOAD STUDIES, CAP MODEL, TR, IE
*RESTART,WRITE,FREQUENCY=10
*NODE
  1
  7,60.
  13,180.
  15,228.
  19,348.
  801,,144.
  807,60,,144.
  813,180,,144.
  815,228,,144.
  819,348,,144.
  20,696.
  820,696,,144.
*NGEN,NSET=BASE
  1,7
  7,13
  13,15
15,19
*NSET,NSET=F1
801,
*NSET,NSET=F2,GENDRAT
802,807
*NGEN,NSET=CENTER
1,801,100
*NGEN,NSET=TOP
801,807
807,813
813,815
815,819
*NFILL
BASE,TOP,8,100
*NGEN,NSET=FAR
20,820,200
*ELEMENT,TYPE=CPE8R
1,1,3,203,201,2,103,202,101
*ELGEN,ELSET=ALL
1,4,200,1,9,2,10
*ELSET,ELSET=PRINTELS
1,2,3,4
*SOLID SECTION, ELSET=ALL, MATERIAL=A1
*MATERIAL, NAME= A1
*ELASTIC
30000., 0.3
*CAP PLASTICITY
16.212, 30.64, 0.1, 0.00041, 0.01, 1.0
*CAP HARDENING
2.15, 0.
20.96, 0.0005
46.6, 0.001
79.67, 0.0015
126.28, 0.002
205.95, 0.0025
311.27, 0.0028
655.6, 0.00299
*ELEMENT, TYPE=CINPE5R
101, 219, 19, 20, 220, 119
*ELGEN, ELSET=FAR
101, 4, 200, 1
*SOLID SECTION, ELSET=FAR, MATERIAL=A2
*MATERIAL, NAME= A2
*ELASTIC
30000., 0.3
*EQUATION
2
F2, 2, 1., 801, 2, -1.
*BOUNDARY
CENTER, 1
F2, 1
BASE, 1, 2
*STEP, INC=50, UNSYM= YES
    PRESERVE DISPLACEMENT
*STATIC
.025, 1., .1
*BOUNDARY
801, 2., -5.0
*MONITOR, NODE=801, DOF=2
*EL PRINT, ELSET=PRINTELS, FREQUENCY=10
S, E
SINV
ENER
E, IE
PE
PEQC
*NODE PRINT,FREQUENCY=5
U,RF
*NODE PRINT,NSET=F1
U,RF
*EL FILE,ELSET=PRINTELS,FREQUENCY=10
S
SINV
ENER
IE
PEQC
*NODE FILE,NSET=F1
U,RF
*END STEP
Slope Stability Problem

This is an illustration of the use of the jointed material model. We examine the stability of the excavation of part of a jointed rock mass, leaving a sloped embankment (Example Problem 1.1.6).

This problem has been studied previously by Barton (1971) and Hoek (1970), who used limit equilibrium methods, and by Zienkiewicz and Pande (1977), who used a finite element model.

As in most geotechnical problems, we begin from a nonzero state of stress. The active “loading” in this case consists of removal of material to represent the excavation.

We examine the effect of joint cohesion on slope collapse through a sequence of solutions with different values of joint cohesion, with all other parameters kept fixed.
Dry Problems

Analysis of Geotechnical Problems with ABAQUS

Joint sets: $\beta_a = 45^\circ$
$\theta_a = \text{variable}$

Bulk rock: $\beta_b = 45^\circ$
$\theta_b = 5600 \text{ kPa}$

$E = 28 \text{ GPa}$
$\nu = 0.2$
$K_0 = 1/3$
$\rho = 2500 \text{ kg/m}^3$

$g = 9.81 \text{ m/s}^2$

Joint sets:
- 90° Joint set 1
- 52.5° Joint set 2

Removed in single stage

70 m

60°
Jointed Rock Slope Problem
The displacement results show the variation of horizontal displacements as cohesion is reduced. They suggest that the slope collapses if the cohesion is less than 24 kPa for the case of associated flow or less than 26 kPa for the case of nondilatant flow.

These values compare well with the value calculated by Barton (26 kPa) using a planar failure assumption in his limit equilibrium calculations. Barton's calculations also include “tension cracking” (akin to joint opening with no tension strength) as we do.

Hoek calculated a cohesion value of 24 kPa for collapse of the slope. Although he also makes the planar failure assumption, he does not include tension cracking. This may be why his calculated value is lower than Barton's.

Zienkiewicz and Pande assume the joints have a tension strength of 1/10 of the cohesion and calculate the cohesion value necessary for collapse as 23 kPa for associated flow and 25 kPa for nondilatant flow.
Displacement Results
Displaced Shape
Joint Set 1 (Vertical Joints) - Plastic Strains (Nonassociated Flow)
Joint Set 2 (Inclined Joints) - Plastic Strains (Nonassociated Flow)
Jointed Slope Stability Input File:

*HEADING
JOINTED ROCK SLOPE, 2 JOINTS, C=30, NONASSOC FLOW, .5 SH
RET

*NODE
1,0.,0.
11,100.,0.
23,272.9,0.
241,0.,74.
251,100.,74.
263,272.9,74.
731,140.4,144.
743,272.9,144.

*NGEN,NSET=BLHS
1,241,40

*NGEN,NSET=BCEN
11,251,40

*NGEN,NSET=BRHS
23,263,40

*NGEN,NSET=TCEN
251,731,40
*GEN, NSET=TRHS
263, 743, 40
*NFILL, BIAS=1.5, TWO STEP
BLHS, BCEN, 10
*NFILL, BIAS=.66666666, TWO STEP
BCEN, BRHS, 12
*NFILL, BIAS=.66666666, TWO STEP
TCEN, TRHS, 12
*NSET, NSET=SLHS, GENERATE
241, 251
*NSET, NSET=SRHS, GENERATE
731, 743
*NSET, NSET=BOT, GENERATE
1, 23
*NSET, NSET=FILN
251, 411, 731
*ELEMENT, TYPE=CPE4
1, 1, 2, 42, 41
101, 11, 12, 52, 51
*ELGEN, ELSET=ALLE
1, 6, 40, 1, 10, 1, 10
101, 18, 40, 1, 12, 1, 20
*SOLID SECTION, ELSET=ALLE, MATERIAL=ALLE
*MATERIAL, NAME=ALLE
*ELASTIC
  2.8E7, .2
*JOINTED MATERIAL, JOINT DIRECTION=JOINT1
  45., 22.5, 30.
*JOINTED MATERIAL, JOINT DIRECTION=JOINT2
  45., 22.5, 30.
*JOINTED MATERIAL, SHEAR RETENTION
  .5
*ORIENTATION, NAME=JOINT1
  1., 0., 0., 0., 0., 0., 0.
*ORIENTATION, NAME=JOINT2
  .7934, -.6088, 0., .6088, .7934, 0.
*INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
  ALLE, 0., 144., -3600., 0., .333333
*RESTART, WRITE, FREQUENCY=5
*STEP, UNSYMM=YES
*GEOSTATIC
  1., 1.
*DLOAD
  ALLE, BY, -25.
*BOUNDARY
BOT,2,2,0.
BLHS,1,1,0.
BRHS,1,1,0.
TRHS,1,1,0.
SLHS,1,2,0.
TCEN,1,2,0.
SRHS,1,2,0.
*EL PRINT
S,MISES,PRESS
E
PE
PEQC
*NODE PRINT
U,RF
*NODE FILE,NSET=FILN
U
*END STEP
*STEP,INC=20, UNSYMM=YES
*STATIC
.1,1.,.001,.1
*CONTROLS,ANALYSIS=DISCONTINUOUS
*BOUNDARY, OP=NEW
BOT, 2, 2, 0.
BLHS, 1, 1, 0.
BRHS, 1, 1, 0.
TRHS, 1, 1, 0.
*MONITOR, NODE=411, DOF=1
*END STEP
A Dynamic Analysis

This example shows the effectiveness of the infinite element quiet boundary formulation in a wave propagation problem (Benchmark Problem 2.2.1).

We compare the results obtained using a small mesh including infinite element quiet boundaries with an extended mesh of finite elements only. Results obtained using the small mesh without the infinite element quiet boundaries are also given to show how the solution is affected by the reflection of the propagating waves.

The three plane strain meshes used for the infinite half-space problem excited by a vertical pulse line load are shown.

The finite element meshes are assumed to have free boundaries at the far field and will reflect the propagating waves.

The material is elastic with $E = 73$ GPa, $\nu = 0.33$, and $\rho = 2842$ kg/m$^3$. Material damping is not included. The applied vertical pulse has a triangular amplitude variation with amplitude of $1.0E+9$. 
The speed of propagation of push waves in the material is approximately 6169 m/s and the speed of propagation of shear waves is approximately 3107 m/s.

The predominant push waves should reach the boundary of the small mesh in 0.324 μs and reach the boundary of the extended mesh in 0.97 μs. The analyses are run for 1.5 μs so the waves are allowed to reflect into the meshes.

The results are shown as time histories of vertical displacements at nodes 13, 103, and 601. It is clear that the small finite/infinite element and the extended finite element meshes give very similar results. The small finite element mesh response is very different as soon as the waves have had time to reflect.
Infinite Element Quiet BoundaryMesh
Small Finite Element Mesh
Dry Problems

Analysis of Geotechnical Problems with ABAQUS

Node 13 Vertical Displacement Response
Node 103 Vertical Displacement Response
Dry Problems

Analysis of Geotechnical Problems with ABAQUS

Node 601 Vertical Displacement Response
Wave Propagation Problem Input File:

*HEADING
VERTICAL PULSE LOADING, QUIET BOUND, 16X16 CPE4R + 32 CINPE4
*NODE
1,0.,0.
801,2e-3,0.
17,0.,-2e-3
817,2e-3,-2e-3
*NGEN,NSET=LHS
1,17
*NGEN,NSET=RHS
801,817
*NFILL
LHS,RHS,16,50
*NSET,NSET=INTER1,GEN
17,817,50
*NSET,NSET=INTER2,GEN
801,816
*NSET,NSET=INTER
INTER1,INTER2
*NCOPY, OLD SET=INTER, CHANGE NUMBER=1000, POLE, NEW SET=FAR
1,
*NSET, NSET=FILEN
13, 103, 601
*ELEMENT, TYPE=CPE4R, ELSET=FE
1, 2, 52, 51, 1
*ELGEN, ELSET=FE
1, 16, 1, 1, 16, 50, 16
*SOLID SECTION, ELSET=FE, MATERIAL=MAT1
*ELEMENT, TYPE=CINPE4, ELSET=IE
257, 67, 17, 1017, 1067
273, 816, 817, 1817, 1816
*ELGEN, ELSET=IE
257, 16, 50, 1
273, 16, -1, 1
*ELSET, ELSET=LOAD, GEN
1, 81, 16
*SOLID SECTION, ELSET=IE, MATERIAL=MAT1
*MATERIAL, NAME=MAT1
*ELASTIC
7.3E10, .33
*DENSITY
2842.0,
*BOUNDARY
LHS,1
*AMPLITUDE,NAME=PULSE
0,0,1e-7,1,2e-7,0
**
*STEP,INC=400
*DYNAMIC,NOMAF
8e-9,1.5e-6
*DLOAD,AMPLITUDE=PULSE
LOAD,p3,1e+9
*OUTPUT,FIELD
*NODE OUTPUT
U,V,A
*OUTPUT,HISTORY
*NODE OUTPUT,NSET=FILEN
U,V,A
*NODE FILE, FREQUENCY=1000, NSET=FILEN
U
*END STEP
Saturated Problems

We present problems involving the analysis of saturated media: we solve stress/fluid flow coupled problems.

Consolidation Problem (Transient)

This example involves the large scale consolidation of a two-dimensional solid (Benchmark Problem 1.14.3).

Nonlinearities caused by the large geometry changes are considered, as well as the effects of the change in the voids ratio on the permeability of the material.

The model, material properties, and boundary conditions used are shown in the first figure.

The load is applied in two equal time increments of a first *SOILS, CONSOLIDATION step, and it is kept constant thereafter.

Practical consolidation analyses require solutions across several orders of magnitude of time, and the automatic time incrementation scheme is designed to generate cost effective solutions for such cases.
The algorithm is based on the user supplying a tolerance on the pore pressure change permitted in any increment, UTOL.

ABAQUS uses this value in the following manner: if the maximum change in pore pressure at any node is greater than UTOL, the increment is repeated with a proportionally reduced time increment.

If the maximum change in pore pressure at any node is consistently less than UTOL, the time increment size is proportionally increased.

In this case UTOL is set to 15 psi. This is about 3% of the maximum pore pressure in the model following application of the load.

With this value the first time increment is 7.2 seconds and the final time increment is $1.266 \times 10^4$ seconds. This is quite typical of diffusion processes: at early times the time rates of pore pressure are significant and at later times these time rates are very low.

The first analysis considers finite-strain effects, and the soil permeability varies with void ratio. A small-strain analysis is also run, with constant permeability. The predictions of the midpoint settlement versus time are shown.
The two analyses predict large differences in the final consolidation: the small-strain result shows about 40% more deformation than the finite-strain case. This is consistent with results from the one-dimensional Terzaghi consolidation solutions.

Clearly, in cases where settlement magnitudes are significant, finite-strain effects are important.
Material:

- Young's modulus = 6.895 MPa (1.0 x 10^3 lb/in^2)
- Poisson's ratio = 0.0
- Initial void ratio = 1.5
- Permeability = 5.08 x 10^{-7} m/s (2.0 x 10^{-5} in/s) at void ratio = 1.5
- Permeability = 5.08 x 10^{-8} m/s (2.0 x 10^{-6} in/s) at void ratio = 1.0

Loading:

- Pressure = q = 3.4475 MPa (500.0 lb/in^2)

Boundary conditions:

- Free drainage across top surface
- Other surfaces impermeable and smooth
Saturated Problems

Analysis of Geotechnical Problems with ABAQUS

Deformation Histories

Vertical displacement of midpoint, $\delta B$

Finite strain

Small strain

Time, s

Deformation Histories
Normalized pore fluid pressure, $u/u_0$

Pore Pressure Histories

Point a

Point b

Time, s

$10^0$ $10^1$ $10^2$ $10^3$ $10^4$ $10^5$ $10^6$
2-D Finite-Strain Consolidation Input File:

*HEADING
   2-D CONSOLIDATION - FINITE STRAIN EXAMPLE
*NODE, INPUT=CONSOL.NOD, NSET=NODES
*NSET, NSET=CLINE, GENERATE
  1,11
*NSET, NSET=BOT, GENERATE
  11,1411,100
*NSET, NSET=WALL, GENERATE
  1401,1411
*NSET, NSET=TOP, GENERATE
  1,1401,200
*ELEMENT, TYPE=CPE8RP
  101,1,201,203,3,101,202,103,2
*ELGEN, ELSET=SOIL
  101,7,200,100,5,2,1
*SOLID SECTION, ELSET=SOIL, MATERIAL=A1
*MATERIAL, NAME=A1
*ELASTIC
  1000.
*PERMEABILITY
2. E-6, 1.
2. E-5, 1.5
*BOUNDARY
CLINE, 1
BOT, 2
WALL, 1
TOP, 8
*INITIAL CONDITIONS, TYPE=RATIO
NODES, 1.5
*RESTART, WRITE, FREQUENCY=25
*STEP, NLGEOM, AMPLITUDE=RAMP
SET UP INITIAL PORE PRESSURES
*SOILS, CONSOLIDATION
3.6, 7.2
*DLOAD
101, P1, 500.
201, P1, 500.
301, P1, 500.
*NODE PRINT, FREQUENCY=5, NSET=CLINE
U, RF, POR, RVT
***PRINT, RESIDUAL=NO, FREQUENCY=5
*EL PRINT, FREQUENCY=25, POSITION=CENTROID
S, MISES, E
*NODE FILE, NSET=CLINE, FREQUENCY=25
U
POR
*END STEP
*STEP, NLGEOM, INC=500
   CONSOLIDATE
*SOILS, CONSOLIDATION, UTOL=15.
    7.2, 7.2E5, 7.2
*END STEP
Dam Problem (Steady-State)

This is a benchmark problem run on ABAQUS regarding the analysis of a concrete dam on a rock foundation. The original description of the benchmark problem provided by Dr. G. Pande of Swansea University formed the basis of the analysis.

The problem consists of a concrete dam on a rock foundation. The dam is 30 m high and the rock foundation extends to a depth of 30 m where an impervious boundary is assumed. The benchmark definition calls for a model of the foundation that extends horizontally 30 m on either side of the base of the dam. We assume that no horizontal displacements take place at the ends of the foundation model; we also assume zero vertical displacements at the bottom, impervious boundary of the rock foundation. The finite element mesh used in the analysis contains 110 CPE8RP elements.
The material comprising the rock foundation has a Young's modulus $E = 30000 \text{ MPa}$ and a Poisson's ratio $\nu = 0.2$. We assume the rock behaves as a Drucker-Prager material with nonassociated flow. The material constants used in the Drucker-Prager model are: cohesion of $0.1 \text{ MPa}$, friction angle of $40^\circ$, and dilation angle of $20^\circ$. The rock exhibits orthotropic permeability with $k_h = 0.0002 \text{ m/sec}$ and $k_v = 0.00001 \text{ m/sec}$. The mass density of the rock is $2400 \text{ kg/m}^3$. The concrete material that forms the dam wall has a Young's modulus of $E = 20000 \text{ MPa}$ and a Poisson's ratio $\nu = 0.25$. In addition, we have used the concrete model in ABAQUS where we assumed that the concrete cracks in tension at a stress of $0.15 \text{ MPa}$ to simulate the requirement that the material should resist a minimal amount of tensile stress. This is a very low tensile strength, perhaps less than one tenth of the real tensile strength of the material. The permeability of the concrete is $0.00001 \text{ m/sec}$ and its mass density is $2400 \text{ kg/m}^3$. 
The problem is done in three stages.

In the first stage, we establish geostatic equilibrium where the in situ stresses balance the gravity loads in the rock. At this point, the rock has a vertical stress of zero at the surface increasing linearly to approximately −0.7 MPa at the impervious bottom boundary; the horizontal stresses are 0.6 times the vertical stress. The excess pore pressure in the rock is zero. This state of stress corresponds to the undeformed configuration of the rock mass.

In the second stage of analysis, we apply the gravity loads due to the concrete dam construction and the pressure loads on the upstream face of the dam representing the filling of the reservoir. In this step of analysis we are concerned with the short-term behavior and assume that no pore pressures develop because the concrete and rock have very low permeabilities. We show the short-term results in the form of a deformed shape and contours of principal stresses and plastic strain.
In the final stage of analysis, we perform a steady-state pore fluid diffusion/stress analysis to calculate the state of the structure 25 years after filling the reservoir. The boundary conditions on the seepage part of the problem consist of an impervious boundary at the bottom of the rock foundation, zero excess pore fluid pressure on the downstream boundary of the rock foundation, and nonzero excess pore pressures on the upstream face of the dam and foundation corresponding to the hydrostatic pressure caused by the head of water in the full reservoir. We also include a phreatic surface in the concrete dam wall. In this case the position of this zero pore pressure surface is assumed, the analysis is performed, and the validity of the assumption is checked; an iterative procedure can then be used to correct the location of the phreatic surface at steady state. Although not done in this case, ABAQUS is capable of calculating the phreatic surface automatically, as illustrated later in **Phreatic Surface Calculation (Steady-State)** (page L6.114). Steady-state results are shown.
Finite Element Model
Short-Term Deformed Shape
Saturated Problems

Analysis of Geotechnical Problems with ABAQUS

Short-Term Minimum Principal Stress
Short-Term Intermediate Principal Stress
Short-Term Maximum Principal Stress
Saturated Problems

Short-Term Plastic Strain
Saturated Problems

Analysis of Geotechnical Problems with ABAQUS

Long-Term Deformed Shape
Long-Term Minimum Principal Stress
Long-Term Intermediate Principal Stress
Long-Term Maximum Principal Stress
Long-Term Plastic Strain
Saturated Problems

Long-Term Pore Pressure
Dam Problem Input File:

*HEADING
AB AQUS BENCHMARK FOR UNITED NATIONS DAM IN INDIA
*** SI UNITS (METER, KILOGRAM, SECOND)
*NODE, NSET=ALLN
  1, 0., 0.
  7, 30., 0.
 15, 51., 0.
 21, 81., 0.
 401, 0., 30.
 407, 30., 30.
 415, 51., 30.
 421, 81., 30.
 807, 31., 40.
1607, 31., 60.
1215, 35., 50.
1615, 35., 60.
*NGEN, NSET=BOTF
  1, 7
  7, 15
  15, 21
*NGEN,NSET=TOPF
401,407
407,415
415,421
*NFILL,NSET=NODF
BOTF,TOPF,10,40
*NSET,NSET=LHSF,GEN
1,401,40
*NSET,NSET=RHSF,GEN
21,421,40
*NGEN,NSET=LHSD
407,807,40
807,1607,40
*NSET,NSET=RHSD
415,1215,40
1215,1615,40
*NFILL,NSET=NODD
LHSD,RHSD,8,1
*NSET,NSET=DRAIN,GEN
21,421,80
415,421,2
415,1615,80
Saturated Problems

Analysis of Geotechnical Problems with ABAQUS

813, 1613, 80
971, 1611, 80
1129, 1609, 80
1287, 1607, 80
*NSET, NSET=ALLN
NODF, NODD
*ELEMENT, TYPE=CPE8RP
1, 1, 3, 83, 81, 2, 43, 82, 41
54, 407, 409, 489, 487, 408, 449, 488, 447
*ELGEN, ELSET=FOUND
1, 10, 2, 1, 5, 80, 10
*ELGEN, ELSET=DAM
54, 4, 2, 1, 15, 80, 10
*ELSET, ELSET=ALLE
FOUND, DAM
*ELSET, ELSET=OUTE, GEN
4, 194, 10
7, 197, 10
*SOLID SECTION, ELSET=FOUND, MATERIAL=ROCK
1.
*SOLID SECTION, ELSET=DAM, MATERIAL=CONCRETE
1.
*MATERIAL, NAME=ROCK
  *ELASTIC
  30000.E6,.2
  *YIELD
  .139E6
  *DRUCKER PRAGER
  40.,1.,20.
  *DENSITY
  2400.
  *PERMEABILITY, SPECIFIC=9810., TYPE=ORTHO
  .0002,.00001,.0002
  *MATERIAL, NAME=CONCRETE
  *ELASTIC
  20000.E6,.25
  *CONCRETE
  10.E6
  20.E6,.01
  *FAILURE RATIOS
  1.16,.0075
  *TENSION STIFFENING
  1.,0.
  0.,.5E-3
*SHEAR RETENTION
1.,1000.
*DENSITY
2400.
*PERMEABILITY, SPECIFIC=9810.
.00001
*BOUNDARY
BOTF, 2
LHSF, 1
RHSF, 1
*INITIAL CONDITIONS, TYPE=RATIO
ALLN, .5
*INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
FOUND, 0., 30., -706320., 0., .6
*RESTART, WRITE, FREQ=10
*STEP
GEOSTATIC STATE OF STRESS IN FOUNDATION
*STATIC
1., 1.
*DLOAD
FOUND, BY, -23544.
*EL PRINT, ELSET=OUTE
S,PRESS,MISES,POR,E11,E22
PE
*EL PRINT,ELSET=DAM
CRACK
CONF
*NODE PRINT
U,RF,POR
*END STEP
*STEP
IMMEDIATELY AFTER FILLING: DAM GRAVITY LOADS+FULL RESERVOIR LOADS
*STATIC
1.,1.
*DLOAD
DAM,BY,-23544.
41,P3,215820.
42,P3,215820.
43,P3,215820.
54,P4,215820.
64,P4,196200.
74,P4,176580.
84,P4,156960.
*END STEP
*STEP
25 YEARS AFTER FILLING: DAM GRAVITY LOADS+FULL RESERVOIR LOADS
*SOILS
.7884E9,.7884E9
*BOUNDARY
DRAIN,8,8,0.
1,8,8,519930.
81,8,8,461070.
161,8,8,402210.
241,8,8,343350.
321,8,8,284490.
401,8,8,225630.
403,8,8,225630.
405, 8, 8, 225630.
407, 8, 8, 225630.
487, 8, 8, 206010.
567, 8, 8, 186390.
647, 8, 8, 166770.
727, 8, 8, 147150.
807, 8, 8, 127530.
887, 8, 8, 107910.
967, 8, 8, 88290.
1047, 8, 8, 68670.
1127, 8, 8, 49050.
1207, 8, 8, 29430.
*END STEP
Partially Saturated Problems

We present problems that involve the analysis of partially saturated media: we solve coupled and uncoupled problems.

Demand Wettability Problem (Uncoupled)

This example illustrates the ABAQUS capability to solve uncoupled partially saturated porous fluid flow problems (Benchmark Problem 1.8.1).

We consider a “constrained demand wettability” test. The demand wettability test is a common way of measuring the absorption properties of porous materials. In such a test, fluid is made available to the material at a certain location and the material is allowed to absorb as much fluid as it can.

We consider a square specimen of material and allow it to absorb fluid at its center. The quarter mesh of reduced-integration elements is shown.
We investigate two cases: one in which the material contains a large number of gel particles that entrap fluid and, as a result, enhance the fluid retention capability of the material, and the other in which the material does not contain gel. We also study the cyclic wetting behavior in the case of the sample containing gel particles.

The “loading” consists of prescribing a zero pore pressure (corresponding to full saturation) at the center of the sample (node 1). This pore pressure is held fixed for 600 seconds to model the fluid acquisition process.

Draining for 600 seconds is modeled by prescribing a pore pressure of −10000.0 at node 1; this corresponds to a saturation of 10%, which is the least saturation the sample can have once it has been wetted.

Finally, we model the rewetting process over a time period of 800 seconds in the third step, by once again prescribing a zero pressure corresponding to full saturation at the center of the sample.
The analysis is performed with the *SOILS, CONSOLIDATION procedure using automatic time incrementation. UTOL, the pore pressure tolerance that controls the automatic incrementation, is set to a large value since we expect the nonlinearity of the material to restrict the size of the time increments during the transient stages of the analysis.

Since the volume occupied by the sample is fixed (all displacements have been constrained) we must expect the volume of fluid absorbed to be the same in the cases of the sample with and without gel particles. The difference will be in the proportions of the volume of the sample that will be occupied by free fluid and fluid trapped in the gel particles.

Time histories of the response at six nodes along the diagonal of the sample are shown.

The cyclic wetting response of the sample is also shown.
Partially Saturated Problems

Analysis of Geotechnical Problems with ABAQUS

Finite Element Model
Absorption/Exsorption Curves
Pore Pressure for Samples With and Without Gel
Fluid Volume Absorbed for Samples With and Without Gel
Saturation for Samples With and Without Gel
Void Ratio for Sample With Gel
Gel Volume Ratio for Sample With Gel
Partially Saturated Problems

Volume of Different Phases (Without Gel)

Volume of Different Phases (With Gel)
Pore Pressure in Cyclic Wettability Test
Fluid Volume Absorbed in Cyclic Test
Saturation in Cyclic Wettability Test
Gel Volume Ratio in Cyclic Wettability Test
Void Ratio in Cyclic Wettability Test
Cyclic Demand Wettability Test Input File:

*HEADING
2-D CYCLIC DEMAND WETTABILITY, WITH GEL
*** UNITS: M, SEC, NEWTON
*NODE, NSET=ALLN
1, 0., 0.
21, .0508, 0.
801, 0., .0508
821, .0508, .0508
*NGEN, NSET=BOT
1, 21, 1
*NGEN, NSET=TOP
801, 821, 1
*NFILL, NSET=ALLN
BOT, TOP, 20, 40
*NSET, NSET=OUTN
1, 165, 329, 493, 657, 821
*ELEMENT, TYPE=CPE8RP, ELSET=BLOCK
1, 1, 3, 83, 81, 2, 43, 82, 41
*ELGEN, ELSET=BLOCK
1, 10, 2, 1, 10, 80, 10
*ELSET, ELSET=OUTE
1, 23, 45, 67, 89, 100
*SOLID SECTION, ELSET=BLOCK, MATERIAL=CORE
  .02
*MATERIAL, NAME=CORE
*PERMEABILITY, SPECIFIC WEIGHT=10000.
  3.7E-4
*SORPTION
 -100000., .04
 -10000., .05
 -4500., .1
 -3500., .18
 -2000., .45
 -1000., .91
  0., 1.
*SORPTION, TYPE=EXSORPTION
 -100000., .09
 -10000., .1
 -8000., .11
 -6000., .18
 -4500., .33
 -3000., .79
-2000., .91
0., 1.
*POROUS BULK MODULI
, 100000000.
*GEL
.0005, .0015, 1.E8, 500.
*INITIAL CONDITIONS, TYPE = SATURATION
ALLN, .05
*NSET, NSET = NPOR, GENERATE
1, 21, 2
81, 101, 2
161, 181, 2
241, 261, 2
321, 341, 2
401, 421, 2
481, 501, 2
561, 581, 2
641, 661, 2
721, 741, 2
801, 821, 2
*INITIAL CONDITIONS, TYPE = PORE PRESSURE
NPOR, -10000.
*INITIAL CONDITIONS,TYPE=RATIO
ALLN, 5.
*RESTART,WRITE,FREQUENCY=10
*STEP,INC=50,AMPLITUDE=STEP
*SOILS,CONSOLIDATION,UTOL=10000.1.,600.,,100.0
*BOUNDARY
ALLN,PINNED
1,8,,,-100.
*CONTROLS,PARAMETERS=FIELD,FIELD=PORE FLUID PRESSURE .01,1.,,,1.E-6
*CONTROLS,ANALYSIS=DISCONTINUOUS
*NODE PRINT,FREQUENCY=10,NSET=NPOR
POR,RVF,RVT
*EL PRINT,FREQUENCY=10
SAT,POR,VOIDR,GELVR
*NODE FILE,FREQUENCY=1,NSET=OUTN
POR,RVF,RVT
*EL FILE,FREQUENCY=1,ELSET=OUTE
SAT,POR,VOIDR,GELVR
*END STEP
*STEP,INC=20,AMPLITUDE=STEP
*SOILS, CONSOLIDATION, UTOL=10000.
1., 600., 100.0
*BOUNDARY
ALLN, PINNED
1, 8, -10000.
*END STEP
*STEP, INC=20, AMPLITUDE=STEP
*SOILS, CONSOLIDATION, UTOL=10000.
1., 800., 100.0
*BOUNDARY
ALLN, PINNED
1, 8, -100.
*END STEP
Desaturation of Soil Column (Transient)

This example validates the ABAQUS capability to solve coupled fluid flow problems in partially saturated porous media where the effects of gravity are important (Benchmark Problem 1.8.4).

We compare ABAQUS results with the experimental work of Liakopoulos (1965). The Liakopoulos experiment consists of the drainage of water from a vertical column of sand. A column is filled with sand and instrumented to measure the moisture pressure through its height.

Prior to the start of the experiment, water is added continually at the top and allowed to drain freely at the bottom. The flow is regulated until zero pore pressure readings are obtained throughout the column.

At this point flow is stopped and the experiment starts: the top of the column is made impermeable and the water is allowed to drain out of the column, under gravity. Pore pressures are measured.

We investigate two cases: one in which the column is not allowed to deform (uncoupled flow problem), and the other in which we consider the deformation of the sand (coupled problem).
The weight is applied by GRAV loading. In the case of the deforming column, an initial step of *GEOSTATIC analysis is performed to establish the initial equilibrium state.

The initial conditions exactly balance the weight of the fluid and dry material so that no deformation takes place, while the zero pore pressure boundary conditions enforce the initial steady state of fluid flow.

Then the fluid is allowed to drain through the bottom of the column, by prescribing zero pore pressures at these nodes, during a *SOILS, CONSOLIDATION step. The fluid will drain until the pressure gradient is equal to the weight of the fluid, at which time equilibrium is established.

The transient analysis is performed using automatic time incrementation. UTOL, the pore pressure tolerance that controls the automatic incrementation, is set to a large value since we expect the nonlinearity of the material to restrict the time increment size.
The pore pressure results of the coupled analysis are closer to the experiment than those of the uncoupled analysis; the uncoupled analysis overestimates the pore pressures in the early stages of the transient. It suggests that the coupled analysis is a better approximation of reality, as we would expect.

As the transient continues, the material deformation slows (see the displacement histories of six points along the height of the column) and therefore the rigid column assumption becomes closer to reality; as steady state is approached both numerical solutions are in good agreement with the experiment.

At steady state, the pore pressure gradient equals the fluid weight density as required by Darcy's law.
Finite Element Model
Absorption/Exsorption Curve

pore pressure, Pa vs saturation
Pore Pressure Profiles (Deformable Column)
Pore Pressure Profiles (Rigid Column)

- Liakopoulos experiment
- ABAQUS (uncoupled)
Displacement Histories (Deformable Column)
Fluid Volume Lost (Deformable and Rigid Columns)
Pore Pressure Histories

<table>
<thead>
<tr>
<th>LINE</th>
<th>VARIABLE</th>
<th>SCALE FACTOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td></td>
<td>+1.00E+00</td>
</tr>
<tr>
<td>2</td>
<td></td>
<td>+1.00E+00</td>
</tr>
<tr>
<td>3</td>
<td></td>
<td>+1.00E+00</td>
</tr>
<tr>
<td>4</td>
<td></td>
<td>+1.00E+00</td>
</tr>
<tr>
<td>5</td>
<td></td>
<td>+1.00E+00</td>
</tr>
<tr>
<td>6</td>
<td></td>
<td>+1.00E+00</td>
</tr>
</tbody>
</table>
Desaturation Example Input File:

*HEADING
ONE DIMENSIONAL DESATURATION PROBLEM, COUPLED
*** UNITS: M, TON, SEC, KN
*NODE,NSET=ALLN
1,0.,0.
3,.1,0.
101,0.,1.
103,.1,1.
*NGEN,NSET=BOT
1,3,1
*NGEN,NSET=TOP
101,103,1
*NFILL,NSET=ALLN
BOT,TOP,20,5
*NSET,NSET=LHS,G generate
1,101,5
*NSET,NSET=RHS,G generate
3,103,5
*ELEMENT,TYPE=CPE8RP,ELSET=BLOCK
1,1,3,13,11,2,8,12,6
*ELGEN,ELSET=BLOCK
1,10,10,1
*ELSET,ELSET=OUTE
1,3,5,7,9
*SOLID SECTION,ELSET=BLOCK,MATERIAL=CORE
*MATERIAL,NAME=CORE
*ELASTIC
1.3E3,0.
*DENSITY
1.5
*POROUS BULK MODULI
,2.E6
*PERMEABILITY,SPECIFIC WEIGHT=10.
4.5E-6
*PERMEABILITY,TYPE=SATURATION
.666666,.85
1.,1.
*SORPTION
-100.,.8
-10.,.85
-9.5,.9
-9.,.922
-8.,.947
-7.,.961
-6.,.973
-4.,.988
-2.,.999
0.,1.
*SORPTION, TYPE=EXSORPTION
-100.,.8
-10.,.85
-9.5,.9
-9.,.922
-8.,.947
-7.,.961
-6.,.973
-4.,.988
-2.,.999
0.,1.
*INITIAL CONDITIONS, TYPE=SATURATION
ALLN,1.
*NSET,NSET=NPOR,GIVEATE
1,101,10
3,103,10
*INITIAL CONDITIONS, TYPE=PORE PRESSURE
NPOR, 0.
*INITIAL CONDITIONS, TYPE=RATIO
ALLN, .4235
*INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
BLOCK, 0., 1., -17.9750615, 0., 0., 0.
*EQUATION
2
3, 8, 1., 1, 8, -1.
*BOUNDARY
ALLN, 1
BOT, 2
*RESTART, WRITE, FREQUENCY=10
*STEP, INC=1
*GEOSTATIC
1.E-6, 1.E-6
*DLOAD
BLOCK, GRAV, 10., 0., -1., 0.
*BOUNDARY
NPOR, 8, 0.
*NODE PRINT, FREQUENCY=5, NSET=LHS
U, RF, POR, RVT
*EL PRINT,FREQUENCY=5,ELSET=OUTE
S,E
SAT,POR,VOIDR
*NODE FILE,FREQUENCY=10,NSET=LHS
U,RF,POR,RVT
*EL FILE,FREQUENCY=10,ELSET=OUTE
S,E
SAT,POR,VOIDR
*END STEP
*STEP,INC=100
*SOILS,CONSOLIDATION,UTOL=10.
20.,50000.
*BOUNDARY,OP=NEW
1,8,,0.
ALLN,1,,0.
BOT,2,,0.
*CONTROLS,ANALYSIS=DISCONTINUOUS
*END STEP
Phreatic Surface Calculation (Steady-State)

This example (Example Problem 8.1.2) illustrates the use of ABAQUS to solve for the flow through a porous medium in which fluid flow is occurring in a gravity field and only part of the region is fully saturated, so that the location of the phreatic surface is a part of the solution. Such problems are common in hydrology. An example is the well draw-down problem, where the phreatic surface of an aquifer must be located, based on pumping rates at particular well locations.

The basic approach takes advantage of the ABAQUS capability to perform partially and fully saturated analysis: the phreatic surface is located as the boundary of the fully saturated part of the model. This approach has the advantage that the capillary zone, just above the phreatic surface, is also identified.

We consider fluid flow only: deformation is ignored.
The upstream face of the dam (surface $S_1$) is exposed to water in the reservoir behind the dam. Since ABAQUS uses a total pore pressure formulation, the pore pressure on this face must be prescribed to be $u_w = (H_1 - z)g\rho_w$. Likewise, on the downstream face of the dam (surface $S_2$), $u_w = (H_2 - z)g\rho_w$. 
The bottom of the dam (surface $S_3$) is assumed to rest on an impermeable foundation. Since the natural boundary condition in the pore fluid flow formulation provides no flow of fluid across a surface of the model, no further specification is needed on this surface.

The phreatic surface in the dam, $S_4$, is found as the locus of points at which the pore fluid pressure, $u_w$, is zero. Above this surface the pore fluid pressure is negative, representing capillary tension causing the fluid to rise against the gravitational force and thus creating a capillary zone. The saturation associated with particular values of capillary pressure is given by the absorption/exsorption curves.

A special boundary condition is needed if the phreatic surface reaches an open, freely draining surface, as indicated on surface $S_5$. In such a case the pore fluid can drain freely down the face of the dam, so that $u_w = 0$ at all points on this surface below its intersection with the phreatic surface. Above this point $u_w < 0$, with its particular value depending on the solution.
This example is specifically chosen to include this effect, to illustrate the use of the ABAQUS drainage-only flow boundary condition (*FLOW with the drainage-only flow type label QnD).
Drainage-only pore fluid flow boundary condition

Finite Element Mesh

The finite element model shows the element edges where the drainage-only boundary condition is applied. On these edges, the pore fluid pressure, $u_w$, is constrained by a penalty method to be less than or equal to zero, thus enforcing the proper drainage-only behavior.
The weight of the water is applied by GRAV loading and the upstream and downstream pore pressures are prescribed as discussed above. A steady-state *SOILS analysis is performed in five increments to allow ABAQUS to resolve the high degree of nonlinearity.

Examining the steady-state contours of pore pressure we see that the upper right part of the dam shows negative pore pressures, indicating that it is partly saturated or dry.
The phreatic surface is best shown when we draw the contours in the vicinity of zero pore pressure. This surface compares well with the analytical phreatic surface calculated by Harr (1962).
The contours of saturation show a fully saturated region under the phreatic surface and decreasing saturation in and above the phreatic zone.
Phreatic Surface Calculation Input File:

*HEADING
EARTH DAM - STEADY STATE FREE SURFACE SEEPAGE
*** UNITS: M, KG, SEC, NEWTON
*NODE,NSET=ALLN
  1,0.,0.
  39,4.8768,0.
  601,1.8288,1.8288
  639,3.048,1.8288
*NGEN,NSET=BOT
  1,39,1
*NGEN,NSET=TOP
  601,639,1
*NFILL,NSET=ALLN
BOT,TOP,12,50
*NSET,NSET=POR0,GGENERATE
  1,39,2
*NSET,NSET=POR1,GGENERATE
  101,139,2
*NSET,NSET=POR2,GGENERATE
  201,239,2
*NSET,NSET=POR3,GENERATE
301,339,2
*NSET,NSET=POR4,GENERATE
401,439,2
*NSET,NSET=POR5,GENERATE
501,539,2
*NSET,NSET=POR6,GENERATE
601,639,2
*NSET,NSET=POR
   POR0, POR1, POR2, POR3
*NSET,NSET=OUTN,GENERATE
1,601,100
21,621,100
*ELEMENT,TYPE=CPE8RP,ELSET=DAM
1,1,3,103,101,2,53,102,51
*ELGEN,ELSET=DAM
1,19,2,1,6,100,20
*ELSET,ELSET=FSIDE,GENERATE
19,119,20
*ELSET,ELSET=FBOT,GENERATE
16,19
*ELSET,ELSET=OUTE,GENERATE
11,111,20
*SOLID SECTION, ELSET=DAM, MATERIAL=FILL
*MATERIAL, NAME=FILL
*ELASTIC
1000.,
*DENSITY
2000.,
*PERMEABILITY, SPECIFIC=10000.
2.1167E-4,
*SORPTION
-100000., .04
-10000., .05
0., 1.
*INITIAL CONDITIONS, TYPE=SATURATION
ALLN, 1.
*INITIAL CONDITIONS, TYPE=PORE PRESSURE
POR, 12192.0, 0.0, 0.0, 1.2192
POR4, 0.
POR5, 0.
POR6, 0.
*INITIAL CONDITIONS, TYPE=RATIO
ALLN, 1.
*BOUNDARY
ALLN,1
ALLN,2
*RESTART,WRITE,FREQUENCY=10
*STEP,INC=5
*SOILS
.2,1.
*DLOAD
DAM,GRAV,10.,0.,-1.,0.
*BOUNDARY
1,8,,12192.
101,8,,9144.
201,8,,6096.
301,8,,3048.
401,8,,0.
*FLOW
FSIDE,Q2D,0.1
FBOT,Q1D,0.1
*CONTROLS,ANALYSIS=DISCONTINUOUS
*NODE PRINT,FREQUENCY=10,NSET=OUTN
POR,
*EL PRINT, FREQUENCY=10, ELSET=OUTE SAT, POR
*NODE FILE, FREQUENCY=10, NSET=OUTN POR,
*EL FILE, FREQUENCY=10, ELSET=OUTE SAT, POR
*END STEP
Excavation and Building Analysis

Many geotechnical applications involve a sequence of steps, in each of which some material is removed or added.

Thus, geotechnical problems offer an interesting perspective on the need for generality in creating and using a finite element model: the model itself, and not just its response, changes with time.

The need for close liaison between the analysis and geometric modeling is important.

The *MODEL CHANGE option provides a convenient way of modeling the addition and/or removal of elements.

In this section, we present an example of such a problem.
Tunneling Problem

This tunneling example was provided by Sauer Corporation. It deals with the sequential excavation of a tunnel designed according to the principles of the New Austrian Tunneling Method. The excavation is done in steps and shotcrete is used as the initial liner. The problem geometry given in the first figure shows the cross-section of a tunnel (shaded area) to be excavated in layered rock. The tunnel is to be lined with shotcrete and some concrete is to be cast at the bottom of the tunnel after excavation. The finite element model is quite coarse (164 CPE4 rock and concrete elements and 15 B21 lining elements) and serves only for illustration purposes. The rock is modeled with the modified Drucker-Prager model where the elastic properties are made dependent on a predefined field variable so that the stiffness can be degraded during the excavation process as called for in the design procedure. The concrete and the shotcrete are modeled as linear elastic.
In the first step of analysis we remove the lining elements (*MODEL CHANGE, REMOVE) since they have to be included in the original mesh but are not in place at the start of the operation. In the second step we apply the gravity loads that equilibrate the virgin stress state in the rock—this causes no deformation. In Step 3 we degrade the stiffness of the top heading of the tunnel, which is excavated in Step 4 by removing the corresponding elements. In Step 4 we also activate (strain free) the shotcrete lining elements on the excavated surface (*MODEL CHANGE, ADD=STRAIN FREE). Steps 5–8 are repetitions of the previous sequence, leading to the excavation of the bench and invert parts of the tunnel and their respective lining. Finally, in Step 9 concrete is cast in place at the bottom of the tunnel.
Finite Element Model
Displacement After Top Heading Excavation/Lining
Displacement After Bench Excavation/Lining
Displacement After Invert Excavation/Lining
Final Displacement After Concrete Casting
Tunneling Problem Input File:

*HEADING
COARSE MESH OF TEST TUNNEL, CPE4, COMPLETE SIMULATION

*NODE
1, 0., 32.4
3, 4., 32.4
9, 25., 32.4
31, 0., 24.1
33, 4., 24.1
39, 25., 24.1
101, 0., 13.
103, 4., 13.
141, 0., 0.
143, 4., 0.
149, 25., 0.
53, 4., 20.3
59, 25., 20.3
63, 4., 18.9
69, 25., 18.9
*NGEN, NSET=LTOP
1, 31, 10
*NGEN, NSET=MTOP
3, 33, 10
*NGEN, NSET=LBOT
101, 141, 10
*NGEN, NSET=MBOT
103, 143, 10
*NGEN, NSET=MCEN
33, 53, 10
63, 103, 10
*NGEN, NSET=RTOP
9, 39, 10
*NGEN, NSET=RCEN
39, 59, 10
69, 109, 10
*NGEN, NSET=RBOT
109, 149, 10
*NFILL, NSET=TOP
LTOP, MTOP, 2
MTOP, RTOP, 6
*NFILL, NSET=CEN
MCEN, RCEN, 6
*NFILL, NSET=BOT
LBOT, MBOT, 2
MBOT, RBOT, 6
*ELEMENT, TYPE=CPE4
  1, 11, 12, 2, 1
  33, 43, 44, 34, 33
  101, 111, 112, 102, 101
*ELGEN, ELSET=OUTERE
  1, 8, 1, 1, 3, 10, 10
  33, 6, 1, 1, 7, 10, 10
  101, 8, 1, 1, 4, 10, 10
***
*NODE
  205, 0., 24.1
  225, 4., 24.1
  245, 4., 20.3
  255, 4., 18.9
  315, 0., 13.
  295, 4., 13.
  201, 0., 21.8
221,1.2,21.8
291,1.2,15.8
311,0.,15.8
203,0.,23.4
243,2.816,20.4
283,2.816,15.8
313,0.,14.2
*NGEN,NSET=MIDOUT
205,225,10
225,245,10
255,295,10
295,315,10
*NGEN,NSET=MIDIN
201,221,10
221,291,10
291,311,10
*NGEN,NSET=MIDMID,LINE=P
203,243,10,,2.1,22.4,0.
243,283,10,,2.9,18.1,0.
283,313,10,,1.408,14.6,0.
*NFILL,NSET=TUNBOU
MIDIN,MIDMID,2
MIDMID,MIDOUT,2
*ELEMENT, TYPE=CPE4
201,211,212,202,201
*ELGEN, ELSET=MIDE
201,4,1,1,11,10,10
***
*NODE
401,0.,21.8
403,1.2,21.8
471,0.,15.8
473,1.2,15.8
*NGEN
401,471,10
403,473,10
401,403,1
471,473,1
402,472,10
*ELEMENT, TYPE=CPE4
401,411,412,402,401
*ELGEN, ELSET=CENE
401,2,1,1,7,10,10
**
**--------DEFINITIONS LINING ELEMENTS--------
**
*ELEMENT, TYPE=B21
1001, 203, 213
1002, 213, 223
1003, 223, 233
1004, 233, 243
1005, 243, 253
1006, 253, 263
2001, 263, 273
2002, 273, 283
3001, 283, 293
3002, 293, 303
3003, 303, 313

**
**--DEFINITIONS OF ELSETS FOR HISTORY--------
**
*ELSET, ELSET=MT11, GENERATE
101, 138, 1
*ELSET, ELSET=MT12, GENERATE
63, 98, 1
*ELSET, ELSET=MT13
253, 254, 263, 264, 273, 274, 283, 284, 293, 294, 303, 304
*ELSET, ELSET=MAT1
MT11, MT12, MT13
**
*ELSET, ELSET=MT21, GENERATE
53, 58, 1
*ELSET, ELSET=MT22
243, 244
*ELSET, ELSET=MAT2
MT21, MT22
**
*ELSET, ELSET=MT31, GENERATE
33, 38, 1
*ELSET, ELSET=MT32, GENERATE
43, 48, 1
*ELSET, ELSET=MT33
203, 204, 213, 214, 223, 224, 233, 234
*ELSET, ELSET=MAT3
MT31, MT32, MT33
**
*ELSET, ELSET=MAT4, GENERATE
1, 28, 1
**
*ELSET, ELSET=TOPH1
202, 212, 222, 232, 201, 211, 221, 231, 401, 402, 411, 412
**
*ELSET, ELSET=TOPH2
421, 422, 241, 242
**
*ELSET, ELSET=TOPH3
431, 432, 251, 252
**
*ELSET, ELSET=BENCH
441, 442, 451, 452, 261, 262, 271, 272
**
*ELSET, ELSET=INVER
461, 462, 281, 282, 291, 292, 301, 302
**
*ELCOPY, OLDSET=INVER, NEWSET=CINV, ELEMENT SHIFT=10000, SHIFT NODES=0
**
*ELSET, ELSET=M3TH1
MAT3, TOPH1
*ELSET, ELSET=M2TH2
MAT2, TOPH2
*ELSET, ELSET=M1TBI
MAT1, TOPH3, BENCH, INVER
**
*ELSET, ELSET=SHHEA, GENERATE
1001, 1006, 1
*ELSET, ELSET=SHBEN
2001, 2002
*ELSET, ELSET=SHINV
3001, 3002, 3003
*ELSET, ELSET=ALLSH
SHHEA, SHBEN, SHINV
**
*ELCOPY, OLDSET=SHHEA, NEWSET=CLIN1, ELEMENT SHIFT=1000, SHIFT NODES=0
*ELCOPY, OLDSET=SHBEN, NEWSET=CLIN2, ELEMENT SHIFT=1000, SHIFT NODES=0
*ELSET, ELSET=CLIN
CLIN1, CLIN2
*ELSET, ELSET=CLIN36, GEN
2003, 2006, 1
**
*ELSET, ELSET=ALGEO
MAT4, M3TH1, M2TH2, M1TBI
**ELSET, ELSET=ALLEL, GENERATE
1,500,1
**
**---DEFINITION OF NODESETS FOR TEMP AND BOUND---**
**
*NSET, NSET=SYMM1
1,11,21,111,121,131,141,205,204,203,202,401,411
*NSET, NSET=SYMM2
421,431,441,451,461,471,312,313,314,315
*NSET, NSET=SYMM
SYMM1, SYMM2
*NSET, NSET=BOTT, GENERATE
141,149,1
*NSET, NSET=EDGE, GENERATE
9,149,10
*NSET, NSET=ALNOD, GENERATE
1,1000,1
**
**---DEFINITIONS MATERIAL PROPERTIES---**
**
**---MATERIAL 1---**
*SOLID SECTION, ELSET=MAT1, MATERIAL=M1
*MATERIAL, NAME=M1
*ELASTIC, TYPE=ISOTROPIC
2414., .27
*DRUCKER PRAGER
54.82, 1, 54.82
*DRUCKER PRAGER HARDENING
2.15
*DENSITY
.0247
**
**-----------------------------**MATERIAL 2----------------------------**
**
*SOLID SECTION, ELSET=MAT2, MATERIAL=M2
*MATERIAL, NAME=M2
*ELASTIC, TYPE=ISOTROPIC
690., .30
*DRUCKER PRAGER
50.19, 1, 50.19
*DRUCKER PRAGER HARDENING
1.
*DENSITY
.021
**
**---------------------MATERIAL 3---------------------**
**
*SOLID SECTION,ELSET=MAT3,MATERIAL=M3
*MATERIAL,NAME=M3
*ELASTIC,TYPE=ISOTROPIC
60.,.325
*DRUCKER PRAGER
44.53, 1, 44.53
*DRUCKER PRAGER HARDENING
.146
*DENSITY
.019
**

**---------------------MATERIAL 4---------------------**
**
*SOLID SECTION,ELSET=MAT4,MATERIAL=M4
*MATERIAL,NAME=M4
*ELASTIC,TYPE=ISOTROPIC
60.,.375
*DRUCKER PRAGER
43.26, 1, 43.26
*DRUCKER PRAGER HARDENING
.205
*DENSITY
.019
**
**-------------------MATERIAL MTOPH1-------------------
**
*SOLID SECTION, ELSET=TOPH1, MATERIAL=MTOPH1
*MATERIAL, NAME=MTOPH1
*ELASTIC, TYPE=ISOTROPIC
60., .325, 0.
24., .325, 1.
24., .325, 8.
*DRUCKER PRAGER
44.53, 1, 44.53
*DRUCKER PRAGER HARDENING
.146
*DENSITY
.019
**
**-------------------MATERIAL MTOPH2-------------------
**
*SOLID SECTION, ELSET=TOPH2, MATERIAL=MTOPH2
*MATERIAL, NAME=MTOPH2
*ELASTIC, TYPE=ISOTROPIC
690., .3, 0.
276., .3, 1.
276., .3, 8.
*DRUCKER PRAGER
50.19, 1, 50.19
*DRUCKER PRAGER HARDENING
1.
*DENSITY
.021
**
**----------------------MATERIAL MTOPH3----------------------
**
*SOLID SECTION, ELSET=TOPH3, MATERIAL=MTOPH3
*MATERIAL, NAME=MTOPH3
*ELASTIC, TYPE=ISOTROPIC
2414., .27, 0.
965.6, .27, 1.
965.6, .27, 8.
*DRUCKER PRAGER
54.82, 1, 54.82
*DRUCKER PRAGER HARDENING
.464
*DENSITY
.0247
**
**-----------------------------MATERIAL MBENCH--------------------------------
**
*SOLID SECTION, ELSET=BENCH, MATERIAL=MBENCH
*MATERIAL, NAME=MBENCH
*ELASTIC, TYPE=ISOTROPIC
 2414., 27, 0.
 2414., 27, 1.
 965.6, 27, 2.
 965.6, 27, 8.
*DRUCKER PRAGER
 54.82, 1, 54.82
*DRUCKER PRAGER HARDENING
  .464
*DENSITY
  .0247
**
**------------------------MATERIAL MINVER------------------------
**
*SOLID SECTION, ELSET=INVER, MATERIAL=MINVER
*MATERIAL, NAME=MINVER
*ELASTIC, TYPE=ISOTROPIC
2414., .27, 0.
2414., .27, 2.
965.6, .27, 3.
965.6, .27, 8.
*DRUCKER PRAGER
54.82, 1, 54.82
*DRUCKER PRAGER HARDENING
.464
*DENSITY
.0247
**
**-----------------MATERIAL SHOTCRETE-----------------**

**
*BEAM SECTION, SECTION=RECT, ELSET=ALLSH, MATERIAL=SHOCR 1., 2.
*MATERIAL, NAME=SHOCR
*ELASTIC, TYPE=ISOTROPIC 15000., 17
*DENSITY .025
**

**-----------------MATERIAL CONCRETE-------------------**

**
*BEAM SECTION, SECTION=RECT, ELSET=CLIN, MATERIAL=CONCR 1., 2.
*SOLID SECTION, ELSET=CINV, MATERIAL=CONCR
*MATERIAL, NAME=CONCR
*ELASTIC, TYPE=ISOTROPIC 27000., 17
*DENSITY .025
*SOLID SECTION, ELSET=CINV, MATERIAL=CONCR
**-------------MPCS AND BOUNDARY-------------**

*MPC
TIE,201,401
TIE,211,402
TIE,221,403
TIE,231,413
TIE,241,423
TIE,251,433
TIE,261,443
TIE,271,453
TIE,281,463
TIE,291,473
TIE,301,472
TIE,311,471
***
TIE,31,205
TIE,32,215
TIE,33,225
TIE,43,235
TIE,53,245
TIE,63,255
TIE,73,265
TIE,83,275
ABAQUS

Excavation and Building Analysis

TIE, 93, 285
TIE, 103, 295
TIE, 102, 305
TIE, 101, 315
**
*BOUNDARY
SYMM, 1
BOTT, 2
EDGE, 1
202, 6
313, 6
**
**-------------------HISTORY-------------------
**
*RESTART, WRITE, FREQUENCY=5
*INITIAL CONDITIONS, TYPE=STRESS, GEOSTATIC
MAT4, 0, 32.4, -.1589, 24.1, .8
M3TH1, -.1589, 24.1, -.2313, 20.3, .45
M2TH2, -.2313, 20.3, -.26, 18.9, .45
M1TBI, -.26, 18.9, -.727, 0.0, .4
**
*INITIAL CONDITIONS, TYPE=TEMPERATURE
ALNOD, 0.
**
**---------- REMOVE ALL LINING ELEMENTS ---------**

**
*STEP, AMPLITUDE=RAMP
*STATIC
*MODEL CHANGE, REMOVE
ALLSH, CLIN, CINV
*END STEP
**

**---------- GRAVITY LOADING ----------**

**
*STEP, AMPLITUDE=RAMP
*STATIC
*DLOAD
ALLEL, GRAV, 1.,-,1.
*EL PRINT, FREQUENCY=10
S, MISES, PRESS
E
PE
*NODE PRINT, FREQUENCY=10
U
RF
*END STEP
**-- EXCAVATION TOP HEADING (1 / 2.5) --**

**
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 1.
*END STEP
**

**-- SHOTCRETE TOP HEADING --**

**
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 1.
*MODEL CHANGE, ADD=STRAIN FREE
SHELA
*EL PRINT, FREQUENCY=10
S, MISES, PRESS
E
PE
SF
*END STEP
**
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD,1.
*MODEL CHANGE, REMOVE
TOPH1, TOPH2, TOPH3
*END STEP
**--------EXCAVATION BENCH ( 1 / 2.5 )----------
**
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD,2.
*END STEP
**----------SHOTCRETE BENCH ----------------------
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD,2.
*MODEL CHANGE, ADD=STRAIN FREE
SHBEN
*END STEP
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 2.
*MODEL CHANGE, REMOVE BENCH
*END STEP
**-----------EXCAVATION INVERT (1 / 2.5)----------
**
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 3.
*END STEP
**----------------SHOTCRETE INVERT ----------------
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 3.
*MODEL CHANGE, ADD=STRAIN FREE SHINV
*END STEP
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 3.
*MODEL CHANGE, REMOVE
INVER
*END STEP
**
**-------CAST IN PLACE CONCRETE----------
**
*STEP, AMPLITUDE=RAMP
*STATIC
*TEMPERATURE
ALNOD, 3.
*MODEL CHANGE, ADD=STRAIN FREE
CLIN36, CINV
*END STEP

Note that the last step does not change the results from the previous step; its presence, however, would impact subsequent steps.
Appendix A

Stress Equilibrium and Fluid Continuity Equations

To study the different couplings and nonlinearities of the coupled problem, we can write the resulting Newton equations at a node as

\[
\begin{bmatrix}
K_{dd} & K_{du} \\
K_{ud} & K_{uu}
\end{bmatrix}
\begin{bmatrix}
d_c \\
u_c
\end{bmatrix}
= 
\begin{bmatrix}
F_r \\
\Delta V_r
\end{bmatrix}
\]

where \(d_c\) represents the vector of displacement corrections, \(F_r\) are the force residuals conjugate to the displacements, \(u_c\) is the pore pressure correction, and \(\Delta V_r\) is the residual change in fluid volume over the time increment conjugate to the pore pressure.

Special cases of the fully and partly saturated problems are presented in the following sections.
Fully Saturated Fluid Flow

\( K_{dd}, K_{du}, K_{ud}, K_{uu} \) have the following components:

\[
K_{dd} = K_s(\bar{\sigma}, d) + L(d, u) + K_{gd}(d)
\]

\[
K_s = \int_V \beta : \bar{D} : \beta \ dV
\]

\[
L = -\int_V \beta : I \ u \ I : \beta \ dV
\]

\[
K_{gd} = -\int_V N : g \ f_1 \ I : \beta \ dV
\]

\[
K_{du} = B(d) + K_{sg}(\bar{\sigma}, d)
\]

\[
B = -\int_V \beta : I \ dV
\]
\[
K_{sg} = \int_V \frac{1}{3K_g} \beta : \bar{\mathbf{D}} : \mathbf{I} \, dV
\]

\[
K_{ud} = \mathbf{B}^T(d) + K_{sg}^T(\bar{\sigma}, \mathbf{d}) + \Delta t L_c(d, e) + \Delta t k_{ec}(\bar{\sigma}, \mathbf{d}, e)
\]

\[
\mathbf{B}^T = -\int_V \mathbf{I} : \beta \, dV
\]

\[
K_{sg}^T = \int_V \frac{1}{3K_g} \mathbf{I} : \bar{\mathbf{D}} : \beta \, dV
\]

\[
L_c = -\int_V \frac{\partial \delta u}{\partial x} \cdot k^* \cdot \left( \frac{\partial u}{\partial x} - \rho_w \mathbf{g} \right) \mathbf{I} : \beta \, dV
\]

\[
k_{ec} = \int_V \frac{\partial \delta u}{\partial x} \cdot \frac{dk^*}{de} \cdot \left( \frac{\partial u}{\partial x} - \rho_w \mathbf{g} \right) - \frac{1}{(1-n)^2} \left[ \frac{\mathbf{I} : \bar{\mathbf{D}}}{3K_g} - \frac{1-n^0}{JK_g} \mathbf{I} \right] : \beta \, dV
\]
\[ K_{uu} = \Delta t k(d, e) + K^*_w(d) + K^*_s(\bar{\sigma}) + \Delta t k_e(\bar{\sigma}, d, e) \]

\[ k = -\int \frac{\partial \delta u}{\partial x} \cdot k^* \cdot \frac{\partial du}{\partial x} \; dV \]

\[ K^*_s = \int \frac{1}{9K_g^2} I: \bar{D} : I \; dV \]

\[ K^*_g = -\int \frac{1-n^0}{J K_g} \; dV \]

\[ K^*_w = \int \frac{1-n^0}{J K_w} - \frac{1}{K_w} \; dV \]

\[ k_e = \int \frac{\partial \delta u}{\partial x} \cdot \frac{dk^*}{\partial e} \cdot \left( \frac{\partial u}{\partial x} - \rho_w g \right) \frac{1}{(1-n)^2} \left[ I: \bar{D} : I - \frac{1-n^0}{9K_g^2} \right] \; dV \]

where \( \beta \) is the strain-displacement matrix, \( \bar{D} \) is the constitutive matrix, \( N \) is the interpolator, \( K_g \) and \( K_w \) are the solid grains and fluid bulk moduli.
**Special Cases**

To facilitate the understanding of the coupled equations, let us consider some special cases of the **transient** problem *without fluid gravity* effects:

- Linear material, small strain, incompressible grains and fluid, constant permeability:

\[
\begin{bmatrix}
K_s & B \\
B^T & \Delta t \ k
\end{bmatrix}
\]

where $K_s$ is the usual stress stiffness; $B, B^T$ are the stress/pore pressure coupling terms; and $\Delta t \ k$ is the porous medium permeability term. The resulting system of equations is linear and symmetric.
• Nonlinear material, large strain, incompressible grains and fluid, nonlinear permeability:

\[
\begin{align*}
K_s (\sigma, d) + L (d, u) & \quad B(d) \\
B^T (d) + \Delta t L_c (d, e) & \quad \Delta t k (d, e)
\end{align*}
\]

where \( K_s (\sigma, d) \) is the stress stiffness term with material and geometric nonlinearities; \( L(d, u) \) is a large volume change term; \( B(d), B^T (d) \) are the stress/pore pressure coupling terms with geometric nonlinearity; \( \Delta t L_c (d, e) \) is a large-strain coupling term with nonlinear permeability; and \( \Delta t k (d, e) \) is the permeability term with geometric and permeability nonlinearities. The problem now becomes nonlinear and unsymmetric. The loss of symmetry is due to the inclusion of finite strains.
• Linear material, small strain, compressible grains and fluid, constant permeability:

\[
\begin{bmatrix}
K_S & B + K_{sg} \\
B^T + K_{sg}^T & \Delta t \ k + K_{sg}^* + K_g^* + K_w^*
\end{bmatrix}
\]

where \(K_s\) is the usual stress stiffness; \(B, B^T\) are the stress/pore pressure coupling terms; \(K_{sg}, K_{sg}^T, K_{sg}^*, K_g^*\) are grain compressibility terms; \(K_w^*\) is a fluid compressibility term; and \(\Delta t \ k\) is the porous medium permeability term. The resulting system of equations is linear and symmetric.
• Nonlinear material, large strain, compressible grains and fluid, nonlinear permeability:

\[
\begin{bmatrix}
K_S(\bar{\sigma},d) + L(d,u) & B(d) + K_{sg}(\bar{\sigma},d) \\
B^T(d) + K_{sg}^T(\bar{\sigma},d) + \Delta t L_C(d,e) & \Delta t k(d,e) + K_{sg}^*(\bar{\sigma}) + K_g^*(d) + K_w^*(d) + \Delta t k_e(\bar{\sigma},d,e)
\end{bmatrix}
\]

where \( \Delta t k_{ec}(\bar{\sigma}, d, e) \), \( \Delta t k_e(\bar{\sigma}, d, e) \) are additional nonlinear grain compressibility terms with large-strain effects. This most general problem is nonlinear and unsymmetric. The loss of symmetry is now due to two reasons: finite strains and nonlinear permeability.
Consider now the **steady-state** problem *without fluid gravity* effects. In its simplest case (linear material, small strain, incompressible grains and fluid, constant permeability), it is only one way coupled:

\[
\begin{bmatrix}
K_S & B \\
0 & k
\end{bmatrix}
\]

The equations are clearly unsymmetric.

The steady-state problem becomes fully coupled if large strains and/or nonlinear permeability are considered. However, the equations retain their unsymmetric characteristic for these cases.

In ABAQUS the steady-state equations are always solved directly in the coupled form for all cases.
The porous media coupled analysis capability can provide solutions either in terms of total or of *excess* pore pressure. The excess pore pressure at a point is the pore pressure in excess of the hydrostatic pressure required to support the weight of pore fluid above the elevation of the material point.

[Total pore pressure solutions are provided when the GRAV distributed load type is used to define the gravity load on the model. Excess pore pressure solutions are provided in all other cases (for example, when gravity loading is defined with distributed load types BX, BY, or BZ)].

One important aspect arising from the inclusion of pore fluid gravity (in total pore pressure analyses) is that it generates a Newton method Jacobian term that is always unsymmetric:

\[
K_{gd} = -\int_V N : g f_1 I : \beta \, dV. (K_{dd})
\]
Partially Saturated Fluid Flow

\( K_{dd}, K_{du}, K_{ud}, K_{uu} \) have the following components:

\[
K_{dd} = K_s(\bar{\sigma}, d) + L(d, u) + K_{gd}(d)
\]

\[
K_s = \int_V \beta : \bar{D} : \beta \, dV
\]

\[
L = -\int_V \beta : I \, su \, I : \beta \, dV
\]

\[
K_{gd} = -\int_V N : g \, sf_1 \, I : \beta \, dV
\]

\[
K_{du} = B(d) + B_s(d, u) + K_{sg}(\bar{\sigma}, d) + K_{sgs}(\bar{\sigma}, d, u) + K_{gu}(d)
\]

\[
B = -\int_V s \, \beta : I \, dV
\]
\[ B_s = -\int_V u \frac{ds}{du} \beta : I \ dV \]

\[ K_{sg} = \int_V \frac{s}{3K_g} \beta : \bar{D} : I \ dV \]

\[ K_{sgs} = \int_V \frac{u}{3K_g} \frac{ds}{du} \beta : \bar{D} : I \ dV \]

\[ K_{gu} = -\int_V N : g \ f_2 \frac{ds}{du} \ dV \]

\[ K_{ud} = B^T (d) + K_{sg}^T (\bar{\sigma}, d) + \Delta t L_c (d, e) + \Delta t k_{ec} (\bar{\sigma}, d, e) \]

\[ B^T = -\int_V s I : \beta \ dV \]
\[ K_{sg}^T = \int_V \frac{s}{3K_g} \mathbf{I : \bar{D}} : \beta \, dV \]

\[ L_c = -\int_V s \frac{\partial \delta u}{\partial \mathbf{x}} \cdot k^* \cdot \left( \frac{\partial u}{\partial \mathbf{x}} - \rho_w g \right) \mathbf{I : \beta} \, dV \]

\[ k_{ec} = \int_V \frac{\partial \delta u}{\partial \mathbf{x}} \cdot \frac{dk^*}{\partial \mathbf{e}} \cdot \left( \frac{\partial u}{\partial \mathbf{x}} - \rho_w g \right) \frac{s}{(1-n)^2} \left[ \mathbf{I : \bar{D}} - \frac{1-n^0}{JK_g} \mathbf{I} \right] : \beta \, dV \]

\[ K_{uu} = \Delta t k(d, e) + K_{sg}(\bar{\sigma}) + K_g(d) + K_w(d) + K_s(d) \]

\[ + \Delta t \, k_e(\bar{\sigma}, d, e) + \Delta t \, k_{es}(d, e) \]

\[ k = -\int_V s \frac{\partial \delta u}{\partial \mathbf{x}} \cdot k^* \cdot \frac{\partial u}{\partial \mathbf{x}} \, dV \]

\[ K_{sg}^* = \int_V \frac{s}{9K_g^2} \mathbf{I : \bar{D}} : \mathbf{I} \, dV \]
where $\beta$ is the strain-displacement matrix, $\bar{D}$ is the constitutive matrix, $N$ is the interpolator, $K_g$ and $K_w$ are the solid grains and fluid bulk moduli.
Special Cases

To facilitate the understanding of the coupled equations, let us consider some special cases of the transient problem without fluid gravity effects:

- Linear material, small strain, incompressible grains and fluid, constant permeability:

\[
\begin{bmatrix}
K_s & B + B_s \\
B^T & \Delta t \ k + \Delta t \ k_{es} + K^*_s
\end{bmatrix}
\]

where \( K_s \) is the usual stress stiffness; \( B, B^T \) are stress/pore pressure coupling terms; \( B_s \) is a partially saturated coupling term; \( \Delta t \ k \) is the permeability term; and \( \Delta t \ k_{es}, K^*_s \) are partially saturated permeability terms.

The resulting system of equations is unsymmetric.
• Nonlinear material, large strain, incompressible grains and fluid, nonlinear permeability:

\[
\begin{align*}
K_s(\bar{\sigma},d) + L(d,u) & \quad B(d) + B_s(d,u) \\
B^T(d) + \Delta t L_c(d,e) & \quad \Delta t \, k(d,e) \\
& \quad + \Delta t \, k_{es}(d,e) + K^*_s(d)
\end{align*}
\]

where \(K_s(\bar{\sigma}, d)\) is the stress stiffness term with material and geometric nonlinearities; \(L(d, u)\) is a large volume change term; \(B(d), B^T(d)\) are coupling terms with geometric nonlinearity; \(B_s(d, u)\) is a partially saturated coupling term with geometric nonlinearity; \(\Delta t \, L_c(d, e)\) is a large-strain coupling term with nonlinear permeability; \(\Delta t \, k(d, e)\) is the permeability term with geometric and permeability nonlinearities; and \(\Delta t \, k_{es}(d, e), K^*_s(d)\) are partially saturated permeability terms with geometric and permeability nonlinearities.

The problem remains unsymmetric.
• Linear material, small strain, compressible grains and fluid, constant permeability:

\[
\begin{bmatrix}
K_s & B + B_s + K_{sg} + K_{sgs} \\
B^T + K_{sg}^T & \Delta t k + \Delta t k_{es} + K_s^* + K_{sg}^* + K_g^* + K_w^*
\end{bmatrix}
\]

where \( K_s \) is the usual stress stiffness; \( B, B^T \) are stress/pore pressure coupling terms; \( B_s \) is a partially saturated coupling term; \( K_{sg}, K_{sg}^T, K_{sgs}, K_{sg}^*, K_g^* \) are grain compressibility terms; \( K_w^* \) is a fluid compressibility term; \( \Delta t k \) is the porous medium permeability term; and \( \Delta t k_{es}, K_s^* \) are partially saturated permeability terms.

The resulting system of equations is unsymmetric.
Nonlinear material, large strain, compressible grains and fluid, nonlinear permeability:

\[
\begin{align*}
K_S (\bar{\sigma}, d) + L (d,u) & \quad \text{B}(d) + B_s(d,u) + K_{sg} (\bar{\sigma}, d) \\
B^T(d) + K_{sg} (\bar{\sigma}, d) + \Delta t L_C (d,e) & \quad + K_{sgs} (\bar{\sigma}, d, u) \\
+ \Delta t k_{ec} (\bar{\sigma}, d, e) & \quad \Delta t k (d,e) + \Delta t k_{es} (d,e) \\
\end{align*}
\]

where \( \Delta t k_{ec}(\bar{\sigma}, d, e), \Delta t k_e(\bar{\sigma}, d, e) \) are additional nonlinear grain compressibility terms with large-strain effects.

This most general problem is unsymmetric.

In conclusion, the transient partially saturated flow problem is always unsymmetric.
Consider now the steady-state problem *without fluid gravity* effects. In its simplest case (linear material, small strain, incompressible grains and fluid, constant permeability), it is only one way coupled:

\[
\begin{bmatrix}
K_s & B + B_s \\
0 & k + k_{es}
\end{bmatrix}
\]

The equations are unsymmetric.

The steady-state problem becomes fully coupled if large strains and/or nonlinear permeability are considered. The equations retain their unsymmetric characteristic for these cases.

In ABAQUS the steady-state equations are always solved directly in the coupled form for all cases.
The porous media coupled analysis capability can provide solutions either in terms of total or of *excess* pore pressure.

[Total pore pressure solutions are provided when the GRAV distributed load type is used to define the gravity load on the model. Excess pore pressure solutions are provided in all other cases (for example, when gravity loading is defined with distributed load types BX, BY, or BZ)].

One important aspect arising from the inclusion of pore fluid gravity (in total pore pressure analyses) is that it generates Newton method Jacobian terms that are always unsymmetric:

\[
K_{gd} = - \int \limits_{V} N : g \; sf_1 \; I : \beta \; dV \quad \text{in} \quad K_{dd}
\]

\[
K_{gu} = - \int \limits_{V} N : g \; f_2 \; \frac{ds}{du} \; dV \quad \text{in} \quad K_{du}
\]
Appendix B

Bibliography of Geotechnical Example Problems

The following is a list of ABAQUS Example and Benchmark Problems that show the use of capabilities for geotechnical modeling:

Example problems:

1.1.6:  Jointed rock slope stability
1.1.10: Stress-free element reactivation

8.1.1:  Plane strain consolidation
8.1.2:  Calculation of phreatic surface in an earth dam
8.1.3:  Axisymmetric simulation of an oil well
8.1.4:  Analysis of a pipeline buried in soil
Benchmark problems:

1.1.10: Concrete slump test

1.8.1: Partially saturated flow in porous media
1.8.2: Demand wettability of a porous medium: coupled analysis
1.8.3: Wicking in a partially saturated porous medium
1.8.4: Desaturation in a column of porous material

1.14.1: The Terzaghi consolidation problem
1.14.2: Consolidation of triaxial test specimen
1.14.3: Finite-strain consolidation of a two-dimensional solid
1.14.4: Limit load calculations with granular materials
1.14.5: Finite deformation of an elastic-plastic granular material
2.2.1: Wave propagation in an infinite medium
2.2.2: Infinite elements: the Boussinesq and Flamant problems
2.2.3: Infinite elements: circular load on half-space
2.2.4: Spherical cavity in an infinite medium

3.2.4: Triaxial tests on a saturated clay
3.2.5: Uniaxial tests on jointed material
3.2.6: Verification of creep integration