Application of the Single Hardening Model in the Finite Element Program ABAQUS

Jakobsen, Kim Parsberg

Publication date:
1999

Document Version
Publisher's PDF, also known as Version of record

Application of the Single Hardening Model in the Finite Element Program ABAQUS

Kim Parsberg Jakobsen

Geotechnical Engineering Group
Department of Civil Engineering
Aalborg University

AALBORG UNIVERSITET
AD NYE VEJE

1999

Application of the Single Hardening Model in the Finite Element Program ABAQUS

Kim Parsberg Jakobsen

Geotechnical Engineering Group
Department of Civil Engineering
Aalborg University
1999

CONTENTS

FIGURES

TABLES

1 INTRODUCTION

The persistent development of new and faster computer hardware, has in general eased the complicated two- and three dimensional analysis of stresses and strains in structures. This development is especially noticeably when it comes to cases where the load is gradually increased towards failure or in combined deformation and flow problems. Today, many of these problems are solved using various finite element computer softwares, capable of handling both geometric and material non-linearities. The latter is especially important in soil mechanics and soil-structure interaction problems. Despite the fact that several conceptual models, describing the non-linear and irreversible behaviour of soil, have been developed over the last three decades few of them are accessible in commercial finite element programs.

In the present study the Single Hardening Model, that is a time independent elastoplastic constitutive model, developed by Lade and Kim (Kim & Lade 1988, Lade & Kim 1988*a*, Lade & Kim 1988*b*) is implemented as a user defined material module, UMAT, in the commercial finite element program, ABAQUS. The advantages of the Single Hardening Model lie in its ability to predict elastic and plastic displacements during loading for various materials such as sand, clay and concrete.

1.1 Scope of work

This work was initially based on an incomplete version of the UMAT, received from Dr. P.V. Lade. Initially ABAQUS was run with this version, which however revealed that a number of revisions were necessary. Consequently, a new and revised UMAT, denoted SHM-module (Single Hardening Model), has been designed and coded.

The SHM-module handles the compatibility requirement, which is the core in elastoplastic modelling, in a more consistent and up-to-date way. The present module provides the opportunity of fulfilling the continuity requirements by various methods. These methods are described and their capabilities are demonstrated in this report. Further, the elastic part of the model has been updated and the behaviour in the softening regime can now be controlled by the user. The feasibility of the SHM-module is demonstrated by several examples including single element analysis for prediction of triaxial behaviour and multiple-element analysis for determination of bearing capacity and settlements in combined deformation and flow problems. Model parameters used for the simulations are all derived from conventional triaxial test performed on Eastern Scheldt Sand (Jakobsen & Praastrup 1998). An evaluation of the models capabilities to predict the true material behaviour is found to be beyond the scope of the present work, hence, no comparisons with test results are performed.

1.2 Report outline

The above mentioned topics are treated separately in the report and it is the idea that the user should be able to use the report as a work of reference. A brief outline of the chapters is given below.

Chapter 2 contains the definition of the commonly used stress and strain quantities.

Chapter 3 presents the governing elastic and plastic stress-strain relations for the Single Hardening Model incorporated in the original version of the UMAT. The elastic behaviour is prescribed by Hooke's law and a pressure dependent function for the elastic coefficients. The framework for plastic behaviour consisting of a failure criterion, a yield criterion, a nonassociated flow rule and work hardening or softening law is described in detail.

Chapter 4 explains the working principles of elastoplastic models and deals with the updating of stresses and the hardening parameter. The integration scheme, used for updating of stresses and the hardening parameter, in the original UMAT is presented and commented on.

Chapter 5 adds an updated elastic model and redefines the soil behaviour in the softening regime. Problems discovered during initial runs with the original UMAT are addressed and corrected in the new version. Furthermore, several changes have been made to the numerical schemes for improvement of the computational efficiency of the SHM-module.

Chapter 6 gives a description of the SHM-modules compatibility with ABAQUS. This includes specification of material and model properties, predefined input and output variables, initialization of state dependent variables etc.

Chapter 7 contains the documentation of the SHM-modules ability to function. The SHM is validated by performing numerous single element analyses of triaxial and true triaxial tests in the compression and extension regimes, following various stress paths.

The work is further documented in appendix A and B, containing the complete SHM-module source code for 2D modelling and matching flow chart.

2 DEFINITIONS

Finite element analyses within the field of geotechnical engineering serve a twofold purpose. Calculation of settlements due to a given external force or determination of complete performance curves as the force is gradually increased towards failure. In both cases the foundation performance is evaluated on the basis of directional displacements and external forces. In most finite element formulations the displacements are the primary variables of the problem and the requested results are directly obtained. This view holds for users who are only interested in the global solution to the problem. When a single element is considered it gets more complicated as the displacements may lead to deformation of the element, and this in turn to internal forces. The magnitude of the internal forces will essentially depend on the relation between material properties and the deformation of the element. A description of the material response based on a relation between deformations and internal forces is, however, meaningless as such a description would depend on the size and geometry of the considered element. Instead the material behaviour must be described by robust and versatile relations between relative quantities as stresses and strains. The stress and strain quantities used throughout this report are defined in advance in order to avoid misunderstandings and repetitions. The description is mainly based on Spencer (1980), Crisfield (1991) and ABAQUS (1995).

2.1 Stresses

When describing the forces acting in the interior of an element it is, from an engineering point of view, obvious to use the Cauchy or true stress measure. This measure is physically easy to interpret, as it simply expresses the ratio between current force and current area. For porous materials in which the pores are interconnected the pressure of the fluid can affect the behaviour of the material greatly. As frictional materials deform and fail in response to normal effective stresses the pore pressure should be deducted. Stresses, which cause compression, are regarded positive and hence the principle of effective stresses can be expressed as:

$$\boldsymbol{\sigma}' = \boldsymbol{\sigma} - u\mathbf{I} \tag{2.1}$$

The stresses, $\boldsymbol{\sigma}$, and the identity $\mathbf{I}$, are both symmetric second order tensors, whereas the pore pressure, u, is a scalar:

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix}; \qquad \mathbf{I} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{2.2}$$

The principle of effective stresses is adopted throughout the report and $\boldsymbol{\sigma}$ generally refers to the effective stress state. The stresses are due to symmetry occasionally presented on vector form:

$$\boldsymbol{\sigma}^T = \begin{bmatrix} \sigma_{11} & \sigma_{22} & \sigma_{33} & \sigma_{23} & \sigma_{31} & \sigma_{12} \end{bmatrix} \tag{2.3}$$

In geotechnical engineering it is common to relate the material response to the pressure or the mean normal effective stress:

$$p = \frac{1}{3}\mathrm{tr}(\boldsymbol{\sigma}) \tag{2.4}$$

In connection with triaxial tests the term 'triaxial stress difference' is used for identification of the scalar difference between major and minor principal stresses:

$$q = \sigma_1 - \sigma_3 \tag{2.5}$$

For isotropic material models it is common to use the three independent Cauchy stress invariants of the stress tensor:

$$I_1 = \mathrm{tr}(\boldsymbol{\sigma}) = 3p \tag{2.6}$$

$$I_2 = -\frac{1}{2}\left[(\mathrm{tr}(\boldsymbol{\sigma}))^2 - \mathrm{tr}(\boldsymbol{\sigma})^2\right] \tag{2.7}$$

$$I_3 = \det(\boldsymbol{\sigma}) \tag{2.8}$$

The deviatoric stress tensor appears from a decomposition of the symmetric stress tensor and expresses the deviation from the isotropic mean stress:

$$\mathbf{s} = \boldsymbol{\sigma} - p\mathbf{I} = \begin{bmatrix} \sigma_{11}-p & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22}-p & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33}-p \end{bmatrix} \tag{2.9}$$

The influence of the deviatoric stresses is often expressed in term of the second deviatoric invariant, defined as:

$$J_2 = \frac{1}{2}\mathbf{s}\mathbf{s}^T = \frac{1}{3}I_1^2 + I_2 \tag{2.10}$$

2.2 Deformations and strains

Determination of displacements throughout the history of loading is as described earlier the analysts primary objective. The geometric configuration of the problem is initially described by the initial position vector $\mathbf{X}$ defined at discrete material points. During loading a material particle will move from its initial position $\mathbf{X}$ to a new position $\mathbf{x}$. By assuming mass conservation there will always be a one-to-one correspondence between $\mathbf{x}$ and $\mathbf{X}$. Hence, it is possible to describe the 'mapping' by the deformation gradient tensor $\mathbf{F}$. That is, having two neighbouring particles in the initial configuration, located at $\mathbf{X}$ and at $\mathbf{X} + d\mathbf{X}$ the distance in the current configuration is given by:

$$d\mathbf{x} = \mathbf{F}d\mathbf{X}; \qquad \mathbf{F} = \frac{\partial \mathbf{x}}{\partial \mathbf{X}} \tag{2.11}$$

If there is no motion, then $\mathbf{x} = \mathbf{X}$ and $\mathbf{F} = \mathbf{I}$. The deformation gradient tensor is not itself a suitable measure of deformation as it makes no distinction between rigid body motions and material straining. The latter is, however, decisive for the description of material behaviour. The deformation can instead be described by a pure body rotation, followed by pure stretch of three orthogonal directions or vice versa. This is the so-called polar decomposition theorem:

$$\mathbf{F} = \mathbf{V}\mathbf{R} \tag{2.12}$$

where $\mathbf{V}$ is the symmetric left stretch tensor and $\mathbf{R}$ is an orthogonal rotation tensor. To be able to evaluate the straining of the material the stretch tensor must be isolated. This is most easily done by defining the strain tensor $\mathbf{B}$ and utilising the properties of $\mathbf{V}$ and $\mathbf{R}$:

$$\mathbf{B} = \mathbf{F}\mathbf{F}^T = \mathbf{V}\mathbf{R}\mathbf{R}^T\mathbf{V} = \mathbf{V}\mathbf{V} \tag{2.13}$$

The principal stretches or stretch ratios $\lambda_1^2 - \lambda_3^2$ and hence $\lambda_1 - \lambda_3$ can be obtained along with the current principal directions $\mathbf{n}_1$, $\mathbf{n}_2$ and $\mathbf{n}_3$ by solving the eigenvalue problem:

$$\left[\mathbf{B} - \lambda^2\mathbf{I}\right]\mathbf{n} = 0 \tag{2.14}$$

For principal stretches of unity no straining occurs. The left stretch tensor, which defines the straining in the current configuration, is given by:

$$\mathbf{V} = \mathbf{q}\begin{bmatrix} \lambda_1 & 0 & 0 \\ 0 & \lambda_2 & 0 \\ 0 & 0 & \lambda_3 \end{bmatrix}\mathbf{q}^T \qquad \mathbf{q} = [\mathbf{n}_1, \mathbf{n}_2, \mathbf{n}_3] \tag{2.15}$$

The stretch itself can be seen as a measure of deformation or 'strain'. It is, however, desirable to use a strain measure that produces zero strain when only rigid body motions occur. This requirement can be satisfied by several different strain measures that are all functions of the stretch. These functions must, however, be differential so that the strain increases or decreases monotonically with the stretch, and defines a unique value of strain for a given stretch.

Obviously, many strain functions are possible and the choice is merely a matter of convenience and appropriateness. However, the strain function cannot be chosen arbitrarily as the stress and strain measures according to Malvern (1969) must be work conjugated and refer to the same configuration (reference or current) when constitutive relations are investigated. The most simple and widely used strain measure is the linear engineering strain. This measure is only useful when both strains and motions are small as products of displacement derivatives are neglected (Spencer 1980). Hence, a distinction between reference and current configuration becomes arbitrary and all stress and strain measures are work conjugated in this case. In situations where the strains are actually small, but large motions occur the finite and non-linear Green's strain may be used. The Green's strain is computationally attractive as it can be deduced directly from the deformation gradient tensor without having to solve for principal stretches and directions (Spencer 1980, Crisfield 1991, Krenk 1993). The use of the Green's strain is nevertheless impeded by its work conjugated stress measure, the second Piola-Kirchoff stresses (Crisfield 1991, Krenk 1993).

When dealing with plastic materials the small strain assumption is violated and both of the above strain measures become inappropriate. Consequently, the finite and non-linear logarithmic or natural strain measure is adopted. The use of the natural strain measure is from a computational point of view troublesome as it cannot be expressed in terms of the deformation gradient tensor. Instead the strain is calculated from to the stretch tensor and principal stretches and directions must be determined. The increment in natural strain, however, has the advantage of being work conjugated with the Cauchy stress measure (Crisfield 1991, ABAQUS 1995). The natural strain tensor is defined as:

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_{11} & \varepsilon_{12} & \varepsilon_{13} \\ \varepsilon_{21} & \varepsilon_{22} & \varepsilon_{23} \\ \varepsilon_{31} & \varepsilon_{32} & \varepsilon_{33} \end{bmatrix} = -\ln(\mathbf{V}) \tag{2.16}$$

Due to symmetry only six of the nine strain components are independent. The independent components are more conveniently arranged on vector form:

$$\boldsymbol{\varepsilon}^T = \begin{bmatrix} \varepsilon_{11} & \varepsilon_{22} & \varepsilon_{33} & \varepsilon_{23} & \varepsilon_{31} & \varepsilon_{12} \end{bmatrix} \tag{2.17}$$

The principal strains are directly obtained from the principal stretch:

$$\boldsymbol{\varepsilon} = \begin{bmatrix} \varepsilon_1 & \varepsilon_2 & \varepsilon_3 \end{bmatrix}^T = -\ln(\boldsymbol{\lambda}) \tag{2.18}$$

The definition implies that strain is positive in compression and corresponds to the sign convention usually adopted in geomechanics. The computations are, furthermore, complicated by the fact that the principal directions generally change as the deformation takes place and it is seldom possible to calculate the total strain directly (Malvern 1969, ABAQUS 1995). This obstacle is evaded by using an incremental form of the polar decomposition theorem:

$$\Delta \mathbf{F} = \Delta \mathbf{V} \Delta \mathbf{R} \tag{2.19}$$

Thus, all quantities obtained from the previous increments are rotated to the current configuration and passed into the user defined material module together with the current strain increment:

$$\Delta \varepsilon = -\ln(\Delta \mathbf{V}) \tag{2.20}$$

It is essential that the increments are small as $\Delta\mathbf{R}$ represents the average rotation over the increment. This requirement is, however, in accordance with the principles of rate dependent material modelling. Since the natural strain measure is consistently adopted in ABAQUS all user defined strain dependent material parameters should be determined on this basis. In geotechnical engineering the material behaviour is commonly studied by performing triaxial and true triaxial tests. If these tests are performed under conditions that ensure a homogenous strain state during loading the total strain definition in 2.16 can be applied as the principal stretch directions remain unchanged throughout the test. It is, furthermore, common to refer to the volumetric strain defined as the sum of the principal strains. An investigation of how the application of the natural strain measure affects the analysis and prediction of the behaviour of geomaterials can be found in Praastrup, Jakobsen & Ibsen (1999).

3 THE SINGLE HARDENING MODEL

The Single Hardening Model has been developed with the purpose of forming a model that is applicable for various frictional materials, such as sand, clay and concrete. The background for the model is a thorough investigation of data from experiments performed on various frictional materials. Thus, results from conventional triaxial tests as well as true triaxial tests have been put into use, in order to be able to provide a precise material description. Besides experimental observations the model is based on the concepts from incremental elastoplastic theory.

The Single Hardening Model belongs to a category of relatively simple models where time dependent and anisotropic behaviour are not incorporated. This means that the model is incapable of capturing phenomena like creep, swelling and material or stress induced anisotropy. Acceptance of these limitations, however, makes the model more applicable to the practical geotechnical engineer as it can be calibrated from a limited number of conventional triaxial compression tests. Moreover, the simplifications facilitate the implementation of the model as it becomes possible to express the governing functions by use of stress invariants.

The present chapter describes the framework of the Single Hardening Model (Lade 1977, Kim & Lade 1988, Lade & Kim 1988*a*, Lade & Kim 1988*b*). It has been tried to follow a logical developmental sequence by describing the governing functions in the following order: Elastic deformations, failure criterion, plastic potential function, yield criterion and work hardening and softening laws. The distinction between plastic potential function and yield criterion imply the use of a so-called non-associated flow rule. The graphical presentation of the different functions is generally based on parameters determined from conventional triaxial compression tests on Eastern Scheldt Sand (Jakobsen & Praastrup 1998)

3.1 Elastic deformations

Upon unloading and reloading of a material recoverable elastic strains are produced. The elastic strains are calculated from an isotropic hypoelastic model where the elastic material coefficients depend on the current state of stress. Thus, the elastic strain must be expressed on an incremental form:

$$d\varepsilon^e = \mathbf{C}^{-1} d\boldsymbol{\sigma} \tag{3.1}$$

in which $\mathbf{C}$ is the stress dependent isotropic elastic tangential stiffness tensor derived from Hooke's extended law.

$$\mathbf{C}^{-1} = \frac{1}{E}\begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix} \tag{3.2}$$

Poisson's ratio ν is assumed to be constant, whereas Young's modulus is assumed to vary with the minor principal effective stress as proposed by Janbu (1963):

$$E = K p_a \left(\frac{\sigma_3}{p_a}\right)^n \tag{3.3}$$

The dimensionless parameters K and n are determined from unloading and reloading branches performed in triaxial tests at varying confining pressure normalized with the atmospheric pressure p_a.

3.2 Failure criterion

The failure criterion bounds a domain of possible stress states and simply prescribes the maximum load that the material can withstand. Experimental results from triaxial and true triaxial tests on frictional materials show that the shear strength increases with increasing mean normal stress.

$$\eta_1 = \left(\frac{I_1^3}{I_3} - 27\right)\left(\frac{I_1}{p_a}\right)^m \tag{3.4}$$

The parameter η_1 determines the opening angle in the triaxial plane (cf. Figure 3.1b) and is comparable to the effective friction angle of the material, whereas m determines the curvature of the failure surface in planes containing the hydrostatic axes (HA). An example of traces and cross sections of the failure criterion in the CO-plane (Praastrup, Ibsen & Lade 1999) and the triaxial plane is given in Figure 3.1.

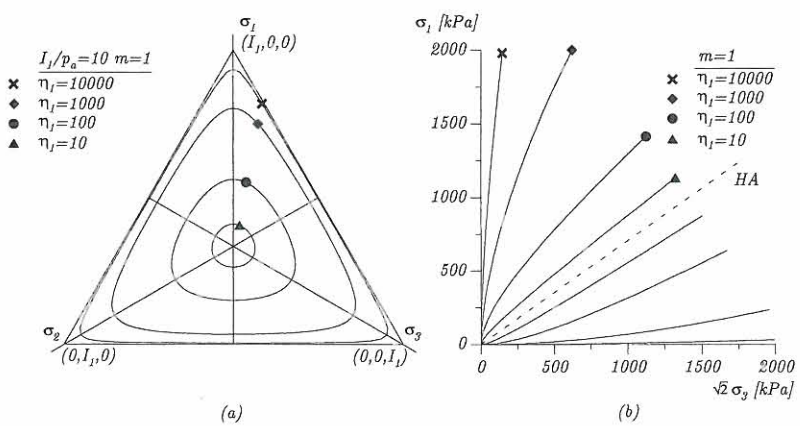

Figure 3.1: *Characteristics of the failure criterion in stress space. (a) Contours in the CO-plane. (b) Traces in the triaxial plane.*

The figure shows that the cross-sectional shape of the failure criteria in the CO-plane changes from circular to triangular with smoothly rounded edges for increasing values of η_1 and constant values of m and I_1. The traces of the failure criterion in the triaxial plane reveal that the formulation is not valid for materials with tensile strength as all the traces tend towards the origin.

However, it is possible to include the effect of a materials tensile strength by a simple translation of the co-ordinate system along the hydrostatic axis.

$$\boldsymbol{\sigma}_a = \boldsymbol{\sigma} + a p_a \mathbf{I} \tag{3.5}$$

The parameter a, which reflects the tensile strength of the material is simply added to the normal stresses before substitution in equation 3.4. The three material parameters may be determined from triaxial or true triaxial tests.

3.3 Flow rule and plastic potential function

The determination of the plastic deformations, within the domain of possible stress states, is based on a so-called flow rule. The flow rule is a stress path independent function that defines the magnitude and direction of the plastic strain increment $d\varepsilon^p$ as the material is subjected to further plastic loading:

$$d\varepsilon^p = d\lambda_p \frac{\partial g}{\partial \boldsymbol{\sigma}} \tag{3.6}$$

The derivative of the plastic potential function g at the current state of stress $\boldsymbol{\sigma}$ determines the direction of plastic strain increment and $d\lambda_p$ is a proportionality factor that determines the magnitude of the plastic strain increment. The determination of $d\lambda_p$ is described in Section 4.1. In cases where the plastic potential and yield function differ, the material is said to follow a non-associated flow rule. The plastic potential function is defined in terms of the stress invariants:

$$g = \left(\psi_1 \frac{I_1^3}{I_3} - \frac{I_1^2}{I_2} + \psi_2\right)\left(\frac{I_1}{p_a}\right)^\mu \tag{3.7}$$

The dimensionless material parameters ψ_1, ψ_2 and μ are obtained from laboratory tests, such as triaxial compression tests. The parameter ψ_1 is a weighting factor that controls the two invariant term's influence on the shape of the plastic potential function in the octahedral plane. ψ_1 can therefore only be experimentally determined from true triaxial test. However, test results indicate that ψ_1 and the curvature parameter m for the failure criterion are related (Kim & Lade 1988):

$$\psi_1 = 0.00155 m^{-1.27} \tag{3.8}$$

The parameters ψ_2 and μ control the intersection with the hydrostatic axis and the curvature of the meridians, respectively. A family of plastic potential surfaces for Eastern Scheldt Sand is shown in Figure 3.2. It appears that the contours of the plastic potential function in the triaxial plane are similar. However, in the CO-plane the contours change shape from circular to triangular as the function value, g, increases.

3.4 Yield criterion

The stress space is, by the elastoplastic theory, divided into a purely elastic domain and a domain in which the material exhibits elastic as well as plastic deformations. The transition between the domains is described by a yield criterion, that defines a closed surface in stress space:

$$f(\boldsymbol{\sigma}, W_p) = f'(\boldsymbol{\sigma}) - f''(W_p) \leq 0 \tag{3.9}$$

The yield function f' is solely dependent on the current state of stress, whereas the work hardening or softening term given by f'' is stress path dependent as it depends on the total plastic work. If the current stress state lies on the yield surface, i.e. the stress state fulfills the condition $f' = f''$, a change in stresses leads to either elastic or elastoplastic deformations. The loading conditions are formally written:

$$\left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T d\boldsymbol{\sigma} \begin{cases} < 0 \text{ elastic unloading} \\ = 0 \text{ neutral loading} \\ > 0 \text{ elastoplastic loading} \end{cases} \tag{3.10}$$

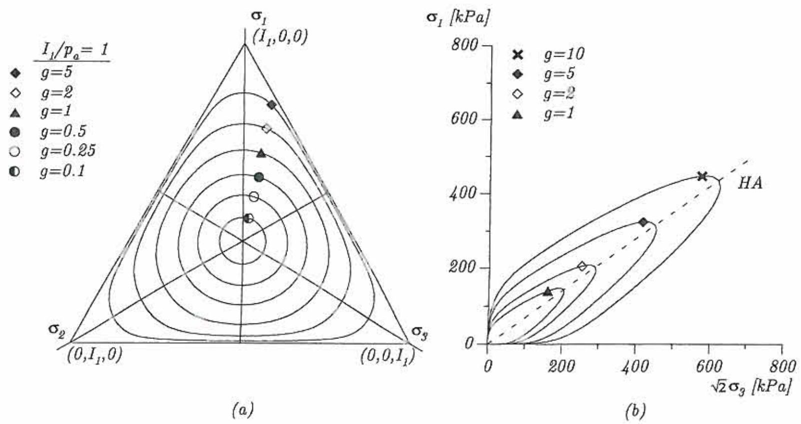

Figure 3.2: *Characteristics of the plastic potential function in stress space. (a) Contours in the CO-plane. (b) Contours in the triaxial plane.* ($m = 0.2879$, $\psi_2 = -3.1540$ *and* $\mu = 2.0611$)

where $\partial f/\partial \sigma$ is the outward normal to the current yield surface. If the stress increment leads to a stress state located inside the yield surface $f' < f''$ only elastic deformations will occur. The condition where df equals zero is termed neutral loading as the new stress state remains on the yield surface, but only elastic deformations will be produced. If the stress increment on the other hand points in the outward direction of the yield surface it corresponds to a state of elastoplastic loading.

In case of elastoplastic loading the size of the yield surface changes, i.e. the yield surface expands during hardening and contracts during softening. To avoid inconsistency it is required that the new stress point is located on the current yield surface. Consistency is ensured by fulfillment of the following condition:

$$df = \left(\frac{\partial f}{\partial \sigma}\right)^T + \frac{\partial f}{\partial W_p} dW_p \tag{3.11}$$

3.4.1 Yield function

The yield function is expressed in terms of stress invariants:

$$f' = \left(\psi_1 \frac{I_1^3}{I_3} - \frac{I_1^2}{I_2}\right)\left(\frac{I_1}{p_a}\right)^h e^q \tag{3.12}$$

The parameters ψ_1 and h are material constants of which the former is given by Equation 3.8. q is a variable, that depends on a material constant α and the stress level S:

$$q = \frac{\alpha S}{1 - (1 - \alpha) S} \tag{3.13}$$

The stress level evolves from zero at the hydrostatic axis to unity at failure:

$$S = \frac{1}{\eta_1}\left(\frac{I_1^3}{I_3} - 27\right)\left(\frac{I_1}{p_a}\right)^m \tag{3.14}$$

3.4.2 Hardening law

Frictional materials generally harden as a consequence of generated plastic strains. By adopting the plastic work, which includes the effect of both plastic volumetric and shear strains, as a state parameter it becomes possible to use a single function for description of the materials hardening behaviour.

The used hardening law is based on observations made from isotropic compression tests:

$$f'' = \left(\frac{1}{D}\right)^{\frac{1}{\rho}}\left(\frac{W_p}{p_a}\right)^{\frac{1}{\rho}} \tag{3.15}$$

in which

$$D = \frac{C}{(27\psi_1 + 3)^\rho} \tag{3.16}$$

and

$$\rho = \frac{p}{h} \tag{3.17}$$

ψ_1 and h are identical to the parameters determined for the yield function, whereas C and p must be determined from an isotropic compression test. The plastic work W_p is determined and continuously updated whenever plastic deformations occur.

$$W_p = \int \sigma d\varepsilon^p \tag{3.18}$$

A family of isotropic yield surfaces for Eastern Scheldt Sand that fulfill the yield criterion is shown in Figure 3.3.

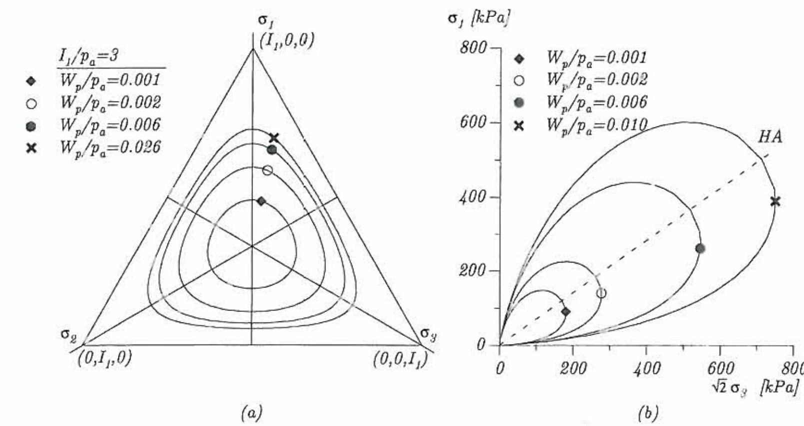

Figure 3.3: *Characteristics of the yield function in stress space. (a) Contours in the CO-plane. (b) Contours in the triaxial plane.* ($m = 0.2879$, $\eta_1 = 70.19$, $C = 1.2748 \cdot 10^{-4}$, $p = 1.6078$, $h = 0.6166$ *and* $\alpha = 0.5525$)

The contours in the triaxial plane, corresponding to constant plastic work, are seen to maintain their shape and merely expand as the plastic work increases. In the CO-plane the contours evolve from circular to triangular as the plastic work and stress level increase.

3.4.3 Softening law

As the stress level reaches unity the material fails to withstand the load and starts to soften. Little information is available about the softening behaviour and for simplicity an isotropic softening law is applied. The softening law is described by an exponential decay function using the plastic work as a state parameter:

$$f'' = Ae^{-B\frac{W_p}{p_a}} \tag{3.19}$$

in which

$$A = f''e^{B\frac{W_p}{p_a}}\Big|_{S=1} \tag{3.20}$$

and

$$B = \frac{df''}{d\left(\frac{W_p}{p_a}\right)}\frac{1}{f''}\Bigg|_{S=1} \tag{3.21}$$

A and B are both positive constants derived from the slope of the hardening curve at failure. The formulation implies that the initial slope of the softening is identical to the slope of the hardening curve at failure, but with the sign reversed. Thus, the model prescribes an abrupt transition from hardening to softening as failure is reached and the yield surface starts to diminish.

3.5 Summary

The governing functions of the Single Hardening Model have been presented and the material variables identified. The material variables will depend on the specific material and must be calibrated to triaxial or true triaxial tests in accordance with the principles outlined in Lade (1977), Kim & Lade (1988), Lade & Kim (1988*a*) and Lade & Kim (1988*b*). The governing functions and the corresponding material variables are listed below:

Elastic properties: K, n, ν
Failure criterion: m, η_1, a
Plastic potential function: ψ_1, ψ_2, μ
Yield function: h, α
Hardening and softening laws: C, p

4 IMPLEMENTATION OF THE SINGLE HARDENING MODEL

As described previously the solution of non-linear finite element problems usually consists of a series of load steps, each involving iterations to establish equilibrium between internal and external forces at the new load level. This global iteration process is entirely handled by the finite element program e.g. ABAQUS. However, the evaluation of the internal forces and displacements, used in the global iterations, is dependent on the applied stress-strain relation. The purpose of the user material module, which include the Single Hardening Model, is therefore twofold. Firstly, it must provide an update of stresses for evaluation of internal forces and secondly, a material stiffness for establishment of the global tangent stiffness matrix used for equilibrium iterations and estimation of the corresponding displacement field.

As ABAQUS uses a Gauss integration scheme to establish the tangent stiffness matrix for each element it is only necessary to consider a single material point within an element. Whenever a new estimate of updated stresses and material stiffness is needed the user material module is called once for each Gauss point. The determination of the updated quantities will essentially depend on the imposed strain increment, but as the material behaviour is of the path-dependent type knowledge of the stress and strain history is also required. The present chapter contains the derivation of the material stiffness, also known as the elastoplastic stiffness matrix, and the method for updating of stress used in the original version of the UMAT. The derived set of formulas presume the use of a work hardening material model, as the Single Hardening Model, but is applicable to nonlinear stress-strain relations in general.

4.1 Derivation of the elastoplastic stiffness matrix

The relations for the Single Hardening Model described in the previous chapter treat elastic and plastic strains separately. Practical computations with finite elements are, however, based on total strain increments and a relation between stress increments and total strain increment is therefore needed. The incremental stress-strain relation is derived on a general form (Chen & Mizuno 1990, Krenk 1993) and the Single Hardening Model is finally implemented (Lade & Nelson 1984).

4.1.1 General stress strain relation

The total strain increment is composed of an elastic and a plastic contribution:

$$d\boldsymbol{\varepsilon} = d\boldsymbol{\varepsilon}^e + d\boldsymbol{\varepsilon}^p \tag{4.1}$$

The plastic strain increment $d\boldsymbol{\varepsilon}^p$ is determined by the non-associated flow rule in (3.8). As the stress increment is common for both the elastic and plastic strain increment the general incremental stress-strain relation can be written as:

$$d\boldsymbol{\sigma} = \mathbf{C}d\boldsymbol{\varepsilon}^e = \mathbf{C}\left(d\boldsymbol{\varepsilon} - d\boldsymbol{\varepsilon}^p\right) = \mathbf{C}\left(d\boldsymbol{\varepsilon} - d\lambda_p\frac{\partial g}{\partial \boldsymbol{\sigma}}\right) \tag{4.2}$$

The proportionality factor, $d\lambda_p$, is unknown and it must be expressed by use of the consistency condition given by (3.13). The consistency condition is on a more general form given by:

$$df = \left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T d\boldsymbol{\sigma} + \left(\frac{\partial f}{\partial \kappa}\right)^T d\kappa = 0 \tag{4.3}$$

where κ is a hardening parameter, that somehow depends upon the plastic strains through $d\lambda_p$:

$$df = \left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T d\boldsymbol{\sigma} - Hd\lambda_p = 0 \tag{4.4}$$

with the hardening modulus, H, defined as:

$$H = -\left(\frac{\partial f}{\partial \kappa}\right)^T \frac{d\kappa}{d\lambda_p} \tag{4.5}$$

The hardening modulus has the dimension of stress and solely describes the effect of hardening. The value of the hardening modulus lies in the range $0 \leq H < \infty$, where the lower limit corresponds to a perfect plastic material or state of failure and the upper limit corresponds to a state of vanishing plastic strain, i.e. purely elastic behaviour.

The proportionality factor, $d\lambda_p$, can be directly obtained from 4.2 and 4.4. However, to avoid dividing with the hardening modulus, which eventually will be zero, 4.2 is first multiplied by $\partial f/\partial \boldsymbol{\sigma}$ and the sum of the two equations gives:

$$d\lambda_p = \frac{\mathbf{C}\left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T}{\left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{C}\left(\frac{\partial g}{\partial \boldsymbol{\sigma}}\right) + H} d\boldsymbol{\varepsilon} \tag{4.6}$$

By inserting 4.6 and 3.6 into the incremental stress-strain relation in 4.2 enables the stress increment to be expressed in terms of the total strain increment:

$$d\boldsymbol{\sigma} = \left[\mathbf{C} - \frac{\mathbf{C}\left(\frac{\partial g}{\partial \boldsymbol{\sigma}}\right)\left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{C}}{\left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{C}\left(\frac{\partial g}{\partial \boldsymbol{\sigma}}\right) + H}\right] d\boldsymbol{\varepsilon} = \left(\mathbf{C} - \mathbf{C}^p\right) d\boldsymbol{\varepsilon} = \mathbf{C}^{ep} d\boldsymbol{\varepsilon} \tag{4.7}$$

$\mathbf{C}^{ep}$ is the so-called elastoplastic stiffness matrix. If the material point is in the elastic domain the plastic stiffness matrix, $\mathbf{C}^p$, vanishes and $\mathbf{C}^{ep}$ and $\mathbf{C}$ becomes identical. It is noted that for non-associated flow $\mathbf{C}^p$ is asymmetric.

4.1.2 Elastoplastic stiffness matrix for the Single Hardening Model

The elastoplastic stiffness matrix for the Single Hardening Model can be established from the expression in 4.7. The elastic stiffness matrix is identical to the one given in Section 3.1, whereas the derivatives of the plastic potential function and yield function must be derived for establishment of the plastic constitutive matrix.

Derivatives of the plastic potential function

The plastic potential function, g, is given by (3.7). As the plastic potential function is expressed in terms of stress invariants the chain rule must be applied:

$$\frac{\partial g}{\partial \boldsymbol{\sigma}} = \frac{\partial g}{\partial I_1}\frac{\partial I_1}{\partial \boldsymbol{\sigma}} + \frac{\partial g}{\partial I_2}\frac{\partial I_2}{\partial \boldsymbol{\sigma}} + \frac{\partial g}{\partial I_3}\frac{\partial I_3}{\partial \boldsymbol{\sigma}} \tag{4.8}$$

in which

$$\frac{\partial g}{\partial I_1} = \left(\psi_1(\mu+3)\frac{I_1^2}{I_3} - (\mu+2)\frac{I_1}{I_2} + \frac{\mu\psi_2}{I_1}\right)\left(\frac{I_1}{p_a}\right)^\mu \tag{4.9}$$

$$\frac{\partial g}{\partial I_2} = \frac{I_1^2}{I_2^2}\left(\frac{I_1}{p_a}\right)^\mu \tag{4.10}$$

$$\frac{\partial g}{\partial I_3} = -\psi_1\frac{I_1^3}{I_3^2}\left(\frac{I_1}{p_a}\right)^\mu \tag{4.11}$$

and the derivatives of the stress invariants with respect to stresses are:

$$\frac{\partial I_1}{\partial \boldsymbol{\sigma}} = \begin{bmatrix} 1 \\ 1 \\ 1 \\ 0 \\ 0 \\ 0 \end{bmatrix} \tag{4.12}$$

$$\frac{\partial I_2}{\partial \boldsymbol{\sigma}} = \begin{bmatrix} -(\sigma_{22}+\sigma_{33}) \\ -(\sigma_{33}+\sigma_{11}) \\ -(\sigma_{11}+\sigma_{22}) \\ 2\sigma_{23} \\ 2\sigma_{31} \\ 2\sigma_{12} \end{bmatrix} \tag{4.13}$$

$$\frac{\partial I_3}{\partial \boldsymbol{\sigma}} = \begin{bmatrix} \sigma_{22}\sigma_{33} - \sigma_{23}^2 \\ \sigma_{33}\sigma_{11} - \sigma_{31}^2 \\ \sigma_{11}\sigma_{22} - \sigma_{12}^2 \\ 2(\sigma_{12}\sigma_{31} - \sigma_{11}\sigma_{23}) \\ 2(\sigma_{23}\sigma_{12} - \sigma_{22}\sigma_{31}) \\ 2(\sigma_{31}\sigma_{23} - \sigma_{33}\sigma_{12}) \end{bmatrix} \tag{4.14}$$

Derivatives of the yield function

The yield function, f', is given by (3.12). Using the chain rule, the derivatives of the yield function can be written as:

$$\frac{\partial f}{\partial \boldsymbol{\sigma}} = \frac{\partial f'}{\partial \boldsymbol{\sigma}} = \frac{\partial f'}{\partial I_1}\frac{\partial I_1}{\partial \boldsymbol{\sigma}} + \frac{\partial f'}{\partial I_2}\frac{\partial I_2}{\partial \boldsymbol{\sigma}} + \frac{\partial f'}{\partial I_3}\frac{\partial I_3}{\partial \boldsymbol{\sigma}} \tag{4.15}$$

in which the derivatives of the stress invariants, $\partial I_1/\partial \boldsymbol{\sigma}$, $\partial I_2/\partial \boldsymbol{\sigma}$ and $\partial I_3/\partial \boldsymbol{\sigma}$, are given by 4.12-4.14. The derivatives of f' with respect to the stress invariants are:

$$\frac{\partial f'}{\partial I_1} = \left(\frac{3+h}{I_1} + \frac{\partial q}{\partial I_1}\right) f' + \frac{I_1}{I_2}\left(\frac{I_1}{p_a}\right)^h e^q \tag{4.16}$$

$$\frac{\partial f'}{\partial I_2} = \frac{I_1^2}{I_2^2}\left(\frac{I_1}{p_a}\right)^h e^q \tag{4.17}$$

$$\frac{\partial f'}{\partial I_3} = f'\frac{\partial q}{\partial I_3} - \psi_1\frac{I_1^3}{I_3^2}\left(\frac{I_1}{p_a}\right)^h e^q \tag{4.18}$$

It is recalled that the exponent q varies with the actual stress level as defined by (3.13) and (3.14). Thus, the derivatives of q with respect to the stress invariants are:

$$\frac{\partial q}{\partial I_1} = \frac{\alpha}{\eta_1(1-(1-\alpha)S)^2}\left(\frac{mS\eta_1}{I_1} + \frac{3I_1^2}{I_3}\left(\frac{I_1}{p_a}\right)^m\right) \tag{4.19}$$

$$\frac{\partial q}{\partial I_3} = -\frac{\alpha}{\eta_1(1-(1-\alpha)S)^2}\frac{I_1^3}{I_3^2}\left(\frac{I_1}{p_a}\right)^m \tag{4.20}$$

Hardening modulus
The hardening modulus defined in 4.5 depends on the adopted hardening law. For the Single Hardening Model, which is of the work hardening type, the hardening modulus can be rewritten using the definition of plastic work and the flow rule. The hardening modulus is given by:

$$H = -\frac{1}{d\lambda_p}\frac{\partial f}{\partial W_p}dW_p \tag{4.21}$$

The increment of plastic work, dW_p, is derived from (3.6) and (3.18):

$$dW_p = d\lambda_p \left(\frac{\partial g}{\partial \boldsymbol{\sigma}}\right)^T \boldsymbol{\sigma} \tag{4.22}$$

Substitution of 4.22 in 4.21 leads to the final formulation of the hardening modulus for work hardening:

$$H = \frac{\partial f}{\partial W_p}\boldsymbol{\sigma}\frac{\partial g}{\partial \boldsymbol{\sigma}} = \frac{\partial f}{\partial W_p}\mu g \tag{4.23}$$

The latter formulation appears by use of Euler's theorem for a homogeneous function g of order μ. The derivative of the yield function with respect to the plastic work depends on the stress history. The derivative is given for both the hardening and softening laws in (3.15) and (3.19), respectively.

$$\frac{\partial f}{\partial W_p} = \frac{\partial f''}{\partial W_p} = \begin{cases} \frac{1}{\rho (D p_a)^{\frac{1}{\rho}}} W_p^{\frac{1}{\rho}-1} & \text{hardening} \\ -\frac{AB}{p_a} e^{-B\frac{W_p}{p_a}} & \text{softening} \end{cases} \tag{4.24}$$

4.2 Updating of stresses and hardening parameter
The objective is to determine the updated state of stress and hardening parameter as a total strain increment is imposed. A common used integration scheme for elastoplastic stress-strain relations is the forward Euler scheme, which is also used in the original version of the UMAT. In the following it is presumed that the initial state of stress is located on a yield surface and that the next stress increment causes elastoplastic deformations.

4.2.1 Forward Euler schemes
In the forward Euler schemes the stresses are updated by replacing the infinitesimal elastoplastic stress-strain relation in 4.7 by a finite incremental relation:

$$\Delta\boldsymbol{\sigma} = \mathbf{C}^{ep}(\boldsymbol{\sigma}_0, \kappa_0)\,\Delta\boldsymbol{\varepsilon} \tag{4.25}$$

where the elastoplastic constitutive matrix is evaluated at the initial state of stress. As the elastoplastic constitutive matrix depends on the stress and strain history this linear approximation is only accurate for very small strain increments. The method may be refined by a piecewise linear integration where the strain increment is subdivided into smaller subincrements (Zienkiewicz & Taylor 1991, Sloan 1987, Crisfield 1991, Chen & Mizuno 1990):

$$\delta\boldsymbol{\varepsilon} = \Delta T \Delta\boldsymbol{\varepsilon} = \frac{\Delta\boldsymbol{\varepsilon}}{m} \tag{4.26}$$

where ΔT is a dimensionless time step of fixed size and the finite stress increment is determined as the sum of the m stress subincrements, $\delta\boldsymbol{\sigma}_i$, each evaluated as a forward Euler step:

$$\delta\boldsymbol{\sigma}_i = \mathbf{C}^{ep}(\boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}_{i-1}, \kappa_0 + \Delta\kappa_{i-1})\,\delta\boldsymbol{\varepsilon} \tag{4.27}$$

$$\delta\kappa_i = \delta\lambda_p(\boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}_{i-1}, \kappa_0 + \Delta\kappa_{i-1}, \delta\boldsymbol{\varepsilon})(\boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}_{i-1})\frac{\partial g}{\partial(\boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}_{i-1})} \tag{4.28}$$

where

$$\Delta\boldsymbol{\sigma}_{i-1} = \sum_{j=1}^{i-1}\delta\boldsymbol{\sigma}_j \tag{4.29}$$

$$\Delta\kappa_{i-1} = \sum_{j=1}^{i-1}\delta\kappa_j \tag{4.30}$$

The given change to the hardening parameter is valid for work hardening and is computed by use of 4.6 and 4.23, with derivatives evaluated at the stress state $(\boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}_{i-1})$. The forward Euler scheme has, as illustrated in Figure 4.1, the advantage of being straight forward and easy to implement compared with other schemes .

Initial state $\boldsymbol{\sigma}_0, \kappa_0, \Delta\boldsymbol{\varepsilon}, \delta\boldsymbol{\varepsilon} = \frac{\Delta\boldsymbol{\varepsilon}}{m}$

Strain increments $i = 1, 2, ..., m$

$\delta\boldsymbol{\sigma}_i = \mathbf{C}^{ep}(\boldsymbol{\sigma}_{i-1}, \kappa_{i-1})\,\delta\boldsymbol{\varepsilon}$

$\delta\kappa_i = \delta\lambda_p(\boldsymbol{\sigma}_{i-1}, \kappa_{i-1}, \delta\boldsymbol{\varepsilon})\,\boldsymbol{\sigma}_{i-1}\frac{\partial g}{\partial\boldsymbol{\sigma}_{i-1}}$

$\boldsymbol{\sigma}_i = \boldsymbol{\sigma}_{i-1} + \delta\boldsymbol{\sigma}_i$

$\kappa_i = \kappa_{i-1} + \delta\kappa_i$

Stop strain subincrementation when $i = m$

Final state $\boldsymbol{\sigma}_i, \kappa_i, \mathbf{C}^{ep}(\boldsymbol{\sigma}_i, \kappa_i)$

Figure 4.1: *Subincremental forward Euler scheme.*

However, the yield criterion is not necessarily fulfilled and the stress increments tend to drift away from the yield surface, i.e. $f(\boldsymbol{\sigma}_0 + \Delta\boldsymbol{\sigma}, \kappa_0 + \Delta\kappa) \neq 0$. Even though the subdivision may reduce the yield surface drift the procedure may lead to unacceptable results as the error accumulates during subsequent load steps (e.g Potts & Gens 1985, Sloan 1987, Crisfield 1991, Krenk 1993). Moreover, the subincremental form, which was used in the original UMAT, has the disadvantage that it uses subincrements of equal size. This turns out to be computationally inefficient as the number of subincrements must be determined by trial an error so that the maximum error is within some close tolerance. The problem is discussed further in Chapter 5.

5 IMPROVEMENT OF THE SHM-MODULE

Initially ABAQUS was run with the original version of the user material module and this revealed that a number of revisions were necessary. The problems discovered were mainly related to the handling of elastic stress increments, but also the general procedure for elastoplastic stress updating were defective. The problems solved are listed below:

1. Improper handling of the transition from the purely elastic to the elastoplastic region.

2. Incorrect back-scaling of elastic trial stresses when these are located outside the positive octant of stress space.

3. Lack of fulfillment of the consistency condition in both the hardening and softening regime.

Besides the numerical improvement the SHM-modules prediction capabilities are enhanced by implementation of an alternative variation of Young's modulus and the possibility to control the material degradation in the softening regime. The numerical performance is furthermore improved by implementation of faster and more accurate integrations schemes and the use of the consistent tangent stiffness matrix instead of the normal elastoplastic constitutive matrix. Finally, the new SHM-module is capable of handling the effect of preshearing on the material strength.

5.1 Additional model features

5.1.1 Alternative variation of Young's modulus

Alternatively a more recent and reasonable relation proposed by Lade & Nelson (1987), that includes the effects of both mean effective stress and deviator stress can be used:

$$E = Mp_a \left[\left(\frac{I_1}{p_a} \right)^2 + 6 \left(\frac{1+\nu}{1-2\nu} \right) \frac{J_2}{p_a^2} \right]^\lambda \tag{5.1}$$

The dimensionless material parameters M and λ can be determined from unloading and reloading branches performed in triaxial tests.

5.1.2 Alteration of softening law parameters

Comparison of the Single Hardening Model with laboratory results indicates that the model generally exaggerates the material degradation after peak failure. It appeared from Section 3.4.3 that the model presumes that the slope of the hardening and softening curves are numerically identical at failure. A possible way to control the strength degradation is simply by changing the initial slope of the softening curve. In practice this is done by introduction of the softening parameter b that reduces the exponent B in 3.21:

$$B = b \left. \frac{df''}{d\left(\frac{W_p}{p_a}\right)} \frac{1}{f''} \right|_{S=1} \tag{5.2}$$

The parameter b is greater than or equal to zero, where the lower limit corresponds to that of a perfect plastic material.

5.2 Initial intersection of the yield surface

It has so far been presumed that the initial stress point is located on the yield surface and that any change in stresses would cause further elastoplastic loading. However, if a stress point changes from an elastic to an elastoplastic state, as it occurs for presheared or overconsolidated materials it is necessary to determine the portion of the stress increment that causes purely elastic deformations (see Figure 5.1).

Thus, the initial state of stress $\boldsymbol{\sigma}_a$ lies within the yield surface corresponding to:

$$f(\boldsymbol{\sigma}_a, \kappa) = f_a < 0 \tag{5.3}$$

where the hardening parameter κ remains constant as only elastic deformations occur. The elastic stress increment is calculated using Hooke's law:

$$\Delta\boldsymbol{\sigma}^e = \mathbf{C}(\boldsymbol{\sigma}_a)\,\Delta\boldsymbol{\varepsilon} \tag{5.4}$$

If the stress point changes from an elastic to an elastoplastic state the elastic trial stress $\boldsymbol{\sigma}_a + \Delta\boldsymbol{\sigma}^e$ may violate the yield criterion:

$$f(\boldsymbol{\sigma}_a + \Delta\boldsymbol{\sigma}^e, \kappa) = f(\boldsymbol{\sigma}_b, \kappa) = f_b > 0 \tag{5.5}$$

It is therefore necessary to determine a scalar α corresponding to the portion of the stress increment that lies within the yield surface so that the stress state $\boldsymbol{\sigma}_c$ fulfills the yield criterion:

$$f(\boldsymbol{\sigma}_a + \alpha\Delta\boldsymbol{\sigma}^e, \kappa) = f(\boldsymbol{\sigma}_c, \kappa) = f_c = 0 \qquad 0 < \alpha < 1 \tag{5.6}$$

Explicit expressions for the scalar α can be derived only for simple types of yield functions. A first estimate may be determined by a simple linear interpolation in f (Sloan 1987, Chen & Mizuno 1990):

$$\alpha_0 = -\frac{f_a}{f_b - f_a} \tag{5.7}$$

The yield function is, however, highly nonlinear and the scalar estimate determined by 5.7 will generally not satisfy the yield criterion:

$$f(\boldsymbol{\sigma}_a + \alpha_0\Delta\boldsymbol{\sigma}^e, \kappa) = f(\boldsymbol{\sigma}_d, \kappa) = f_d \neq 0 \tag{5.8}$$

A more accurate estimate for α may be obtained by a Taylor series expansion around $\boldsymbol{\sigma}_a + \alpha_0\Delta\boldsymbol{\sigma}^e$:

$$\alpha = \alpha_0 - \frac{f_d}{\left(\frac{\partial f}{\partial(\boldsymbol{\sigma}_a + \alpha_0\Delta\boldsymbol{\sigma}^e)}\right)^T \Delta\boldsymbol{\sigma}^e} \tag{5.9}$$

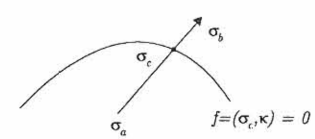

Figure 5.1: *Intersection of the yield surface.*

The original UMAT uses this estimate of the scalar α to determine the elastic portion of the stress increment. As for the forward Euler integration scheme this approach will be accurate for small strain increments only. In order to avoid any initial yield surface drift in the integration schemes and enhance the stress-strain relations prediction capabilities it is advisable to apply an iterative scheme (Sloan 1987). Using the Newton-Raphson technique, for example, $\boldsymbol{\sigma}$ and α are updated as outlined in Figure 5.2. The iterative procedure is started by assuming $\boldsymbol{\sigma}_0 = \boldsymbol{\sigma}_a$ and using α_1 from equation 5.7. The procedure is terminated when the stress norm $\|\boldsymbol{\sigma}_i - \boldsymbol{\sigma}_{i-1}\| \,/\, \|\boldsymbol{\sigma}_{i-1}\|$ is less than a specified tolerance. The obtained elastic stress increment $\alpha\Delta\boldsymbol{\sigma}^e$ corresponds to a elastic strain increment of $\alpha\Delta\boldsymbol{\varepsilon}$ and the strain increment used in the integration of the elastoplastic stress-strain relation equals $(1-\alpha)\,\Delta\boldsymbol{\varepsilon}$.

Initial state $\boldsymbol{\sigma}_0, \kappa$

If $f(\boldsymbol{\sigma}_0, \kappa) > 0$

$$\alpha_0 = -\frac{f_a}{f_b - f_a}$$

Iterations $i = 1, 2, \ldots, i_{\max}$

$$\boldsymbol{\sigma}_i = \boldsymbol{\sigma}_{i-1} + \alpha_{i-1}\Delta\boldsymbol{\sigma}^e$$

$$\alpha_i = \alpha_{i-1} + \Delta\alpha_i$$

$$\Delta\alpha_i = -\frac{f(\boldsymbol{\sigma}_i, \kappa)}{\left(\frac{\partial f}{\partial \boldsymbol{\sigma}_i}\right)^T \Delta\boldsymbol{\sigma}^e}$$

Stop iteration when $\|\boldsymbol{\sigma}_i - \boldsymbol{\sigma}_{i-1}\| \,/\, \|\boldsymbol{\sigma}_{i-1}\| \leq \epsilon$

Final state $\boldsymbol{\sigma}_c = \boldsymbol{\sigma}_i, \alpha_c = \alpha_i$

Figure 5.2: *Initial intersection of the yield surface.*

5.2.1 Handling the effect of preshearing on the material strength

The numerical problems that occur for presheared or over consolidated materials are generally handled by applying the procedure outlined in the previous section. However, in special cases the current yield surface, that indicates a previously experienced load level, extends beyond the failure criterion and actually strengthens the material. Thus, the material should not reach a state of failure until the yield surface is reached. This is numerically accomplished by calculating the intersection of the yield surface and subsequently by performing a check for failure. The stress level relative to failure, as defined by 3.14, will in such cases exceed unity, but is for consistency put equal to unity.

5.3 Back-scaling of elastic trial stresses

As a consequence of the fact that the Single Hardening Model is developed for frictional materials, only stress combinations in the pressure octant of stress space are allowed. This implies

that if the elastic trial stresses correspond to one (or more) negative or zero principal stresses, the stresses must be scaled back into the pressure octant, in order to facilitate the calculation of the elastoplastic stress-strain response. In the original version this back-scaling could result in a change of direction for the elastic stress increment, whereby erroneous results were produced. The scaling procedure has therefore been modified, so that the original direction is maintained. The problem most likely occurs for large strain increments and near failure in the extension region.

5.4 Correction for yield surface drift

When using explicit integrations schemes, as the forward Euler scheme presented in Section 4.2.1, for updating of stresses the stress state predicted at the end of the elastoplastic increment of loading may not lie on the current yield surface and the consistency condition is violated. The error will essentially depend on the size of the strain increment and number of subdivisions, but as the error is cumulative it is important to ensure that the stresses are corrected back to the yield surface during each increment of loading.

A method proposed by Potts & Gens (1985), that accounts for the changes in elastic strains which accompany any stress correction, is applied. The problem is illustrated schematically in Figure 5.3 where the material is subjected to loading which causes elastoplastic deformation.

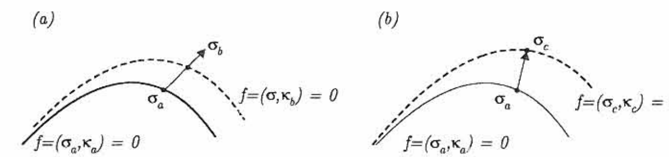

Figure 5.3: *Illustration and correction of yield surface drift. (a) Initial estimate on updated hardening parameter and stresses causing yield surface drift. (b) Corrected values of hardening parameter and stresses located on the yield surface.*

The material is at the initial state of stress $\boldsymbol{\sigma}_a$ located on the yield surface $f(\boldsymbol{\sigma}_a, \kappa_a) = 0$. Further loading involves elastoplastic deformation and a change in stresses as determined by the integrations schemes. This new stress state $\boldsymbol{\sigma}_b$ will due to the tendency for yield surface drift not necessarily be located on the new yield surface $f(\boldsymbol{\sigma}, \kappa_b) = 0$ and the objective is therefore to correct the stresses so that the yield criterion is fulfilled. During the correction process the total strain increment will remain constant which implies that any elastic strain change must be balanced by an equal and opposite change in plastic strain. These changes will affect the stresses and hardening parameter as well, wherefore the new and updated elastoplastic stress state is denoted $\boldsymbol{\sigma}_c$ and κ_c.

The requirement of an unaltered total strain increment throughout the correction process can be formulated as:

$$d\boldsymbol{\varepsilon}^p = -d\boldsymbol{\varepsilon}^e = -\mathbf{C}^{-1}(\boldsymbol{\sigma}_c - \boldsymbol{\sigma}_b) \tag{5.10}$$

The plastic strain increment is, moreover, proportional to the gradient of the plastic potential:

$$d\boldsymbol{\varepsilon}^p = \beta\frac{\partial g}{\partial \boldsymbol{\sigma}} \tag{5.11}$$

where β is a scalar quantity. The corrected stress state is obtained by substituting equation 5.11 into equation 5.10 and solving for $\boldsymbol{\sigma}_c$:

$$\boldsymbol{\sigma}_c = \boldsymbol{\sigma}_b - \beta \mathbf{C} \frac{\partial g}{\partial \boldsymbol{\sigma}_b} \tag{5.12}$$

The change in plastic strains will also affect the hardening parameter:

$$\kappa_c = \kappa_b + \Delta\kappa \tag{5.13}$$

where $\Delta\kappa$ for work hardening is given by:

$$\Delta\kappa = \beta \left(\frac{\partial g}{\partial \boldsymbol{\sigma}_b}\right)^T \boldsymbol{\sigma}_b \tag{5.14}$$

The corrected stress state must satisfy the yield criterion:

$$f(\boldsymbol{\sigma}_c, \kappa_c) = f\left(\boldsymbol{\sigma}_b - \beta \mathbf{C}\frac{\partial g}{\partial \boldsymbol{\sigma}_b}, \kappa_b + \beta\left(\frac{\partial g}{\partial \boldsymbol{\sigma}_b}\right)\boldsymbol{\sigma}_b\right) = 0 \tag{5.15}$$

A first order estimate of the scalar β can be obtained by a Taylor series expansion around $\boldsymbol{\sigma}_b$:

$$\beta = \frac{f(\boldsymbol{\sigma}_b, \kappa_b)}{\left(\frac{\partial f}{\partial \boldsymbol{\sigma}_b}\right)^T \mathbf{C}\left(\frac{\partial g}{\partial \boldsymbol{\sigma}_b}\right) - \left(\frac{\partial f}{\partial \kappa_b}\right)\left(\frac{\partial g}{\partial \boldsymbol{\sigma}_b}\right)^T \boldsymbol{\sigma}_b} \tag{5.16}$$

The above procedure may be of sufficient accuracy if the loading increment is small. However, it must in general be checked that the corrected stress state fulfills the yield criterion $f(\boldsymbol{\sigma}_c, \kappa_c) = 0$ to some close tolerance. Otherwise an iterative procedure as outlined in Figure 5.4 must be applied. The iterative procedure is started by assuming $\boldsymbol{\sigma}_0 = \boldsymbol{\sigma}_b$ and $\kappa_0 = \kappa_b$.

5.5 Stress updating

In the original UMAT a simple forward Euler scheme was used for updating of the stresses and the hardening parameter. The forward Euler scheme is, however, only accurate for small strain increments. Alternatively, it is possible to refine the scheme by subdividing the strain increment into a fixed number of subincrements. This approach will essentially improve the accuracy, but turns out to be computationally expensive (Sloan 1987). In order to improve the accuracy and the computational efficiency of the SHM-module two new schemes have been investigated and implemented.

The most widely used integration schemes used for elastoplastic stress-strain relations are of the backward and forward Euler type. The backward Euler scheme is an elastic predictor and plastic corrector type of method and is attractive because it does not require the initial intersection of the yield surface to be computed if the stress point passes from an elastic to an elastoplastic state. The plastic correction is obtained by solving a small system of nonlinear equations (Crisfield 1991, Krenk 1993) by an iterative procedure, which ensures that the consistency condition may be satisfied within a specified tolerance. The approach, however, has some major disadvantages. The establishment of the system of equations becomes laborious for more advanced models and convergence is not necessarily guaranteed. Furthermore many soil models have a separate failure criterion and the numerical handling may be obstructed near failure as the elastic trial stress intersects the failure criterion. Alternatively integration schemes of the forward Euler or Runge-Kutta type, that uses an explicit formulation, can be used. This type of schemes have the disadvantage that the initial intersection with the yield

Initial state $\boldsymbol{\sigma}_0, \kappa_0$

If $|f(\boldsymbol{\sigma}_0, \kappa_0)| > \epsilon$

Iterations $i = 1, 2, ..., i_{\max}$

$$\beta_{i-1} = \frac{f(\boldsymbol{\sigma}_{i-1}, \kappa_{i-1})}{\left(\frac{\partial f}{\partial \boldsymbol{\sigma}_{i-1}}\right)^T \mathbf{C}\left(\frac{\partial g}{\partial \boldsymbol{\sigma}_{i-1}}\right) - \left(\frac{\partial f}{\partial \kappa_{i-1}}\right)^T \boldsymbol{\sigma}_{i-1}\left(\frac{\partial g}{\partial \boldsymbol{\sigma}_{i-1}}\right)}$$

$$\boldsymbol{\sigma}_i = \boldsymbol{\sigma}_{i-1} - \beta_{i-1}\mathbf{C}\frac{\partial g}{\partial \boldsymbol{\sigma}_{i-1}}$$

$$\Delta\kappa_{i-1} = \beta_{i-1}\boldsymbol{\sigma}_{i-1}\frac{\partial g}{\partial \boldsymbol{\sigma}_{i-1}}$$

$$\kappa_i = \kappa_{i-1} + \Delta\kappa_{i-1}$$

Stop iteration when $|f(\boldsymbol{\sigma}_i, \kappa_i)| \leq \epsilon$

Final state $\boldsymbol{\sigma}_c = \boldsymbol{\sigma}_i, \kappa_c = \kappa_i$

Figure 5.4: *Correction for yield surface drift.*

surface must be computed if a stress point passes from an elastic to an elastoplastic state and they do not necessarily ensure that the consistency condition is fulfilled. The validity of this type of methods can, however, be enhanced and the disadvantages are balanced by their robustness.

On this basis it is found advisable to focus on two explicit integration schemes. This includes a refined subincremental version of the forward Euler scheme with active error control and an enhanced Runge-Kutta scheme.

The methods are applicable to nonlinear stress-strain relations in general, but the stress-strain relations given by the Single Hardening Model are applied as far as possible. The section is concluded by an example where the capabilities of the different methods are studied.

5.5.1 Modified forward Euler scheme with error control

In order to reduce the yield surface drift and computational costs of the forward Euler scheme a modified Euler scheme with active error control can be used (Sloan 1987, Sloan & Booker 1992). Instead of using a fixed number of subincrements of equal size, the size of the subincrements is varied throughout the integration process. Hence, the size of each subincrement is determined so that the new stress state fulfills the yield criterion to some close tolerance and only the absolutely necessary number of subdivisions are applied.

The modified scheme uses a pair of first and second order Euler formulas to estimate the error produced by the standard forward Euler scheme at the end of a strain increment, $\delta\boldsymbol{\varepsilon} = \Delta T \Delta\boldsymbol{\varepsilon}$. The first estimate of the updated stresses and the work hardening parameter at the end of the strain increment is given by:

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_0 + \delta\boldsymbol{\sigma}^I \tag{5.17}$$

$$\kappa = \kappa_0 + \delta\kappa^I \tag{5.18}$$

where

$$\delta\boldsymbol{\sigma}^I = \mathbf{C}^{ep}(\boldsymbol{\sigma}_0, \kappa_0)\,\delta\boldsymbol{\varepsilon} \tag{5.19}$$

$$\delta\kappa^I = \delta\lambda_p(\boldsymbol{\sigma}_0, \kappa_0, \delta\boldsymbol{\varepsilon})\,\boldsymbol{\sigma}_0\frac{\partial g}{\partial\boldsymbol{\sigma}_0} \tag{5.20}$$

A more accurate estimate of the updated stress state may be found by:

$$\hat{\boldsymbol{\sigma}} = \boldsymbol{\sigma}_0 + \frac{1}{2}\left(\delta\boldsymbol{\sigma}^I + \delta\boldsymbol{\sigma}^{II}\right) \tag{5.21}$$

$$\hat{\kappa} = \kappa_0 + \frac{1}{2}\left(\delta\kappa^I + \delta\kappa^{II}\right) \tag{5.22}$$

where

$$\delta\boldsymbol{\sigma}^{II} = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}_0 + \delta\boldsymbol{\sigma}^I, \kappa_0 + \delta\kappa^I\right)\delta\boldsymbol{\varepsilon} \tag{5.23}$$

$$\delta\kappa^{II} = \delta\lambda_p\left(\boldsymbol{\sigma}_0 + \delta\boldsymbol{\sigma}^I, \kappa_0 + \delta\kappa^I, \delta\boldsymbol{\varepsilon}\right)\left(\boldsymbol{\sigma}_0 + \delta\boldsymbol{\sigma}^I\right)\frac{\partial g}{\partial\left(\boldsymbol{\sigma}_0 + \delta\boldsymbol{\sigma}^I\right)} \tag{5.24}$$

The method is seen to employ two evaluations of the elastoplastic constitutive matrix in each subincrement. The difference between the stress states given by 5.17 and 5.21 can be seen as an estimate of the local error in $\boldsymbol{\sigma}$:

$$\delta\boldsymbol{\sigma}^{I-II} \approx \hat{\boldsymbol{\sigma}} - \boldsymbol{\sigma} = \frac{1}{2}\left(\delta\boldsymbol{\sigma}^{II} - \delta\boldsymbol{\sigma}^I\right) \tag{5.25}$$

This error estimate serves as a guide for selecting the size of the next time step, ΔT, when integrating over the total strain increment $\Delta\boldsymbol{\varepsilon}$. That is, the relative error for a subincrement is defined by the stress norm:

$$\xi = \frac{\left\|\delta\boldsymbol{\sigma}^{I-II}\right\|}{\left\|\hat{\boldsymbol{\sigma}}\right\|} \tag{5.26}$$

and the size of each step is continually adjusted until ξ is less than some specified tolerance, ϵ.

The integration is started by choosing a value of the dimensionless time step ΔT and computing $\delta\boldsymbol{\varepsilon}$, $\boldsymbol{\sigma}$, κ, $\hat{\boldsymbol{\sigma}}$, $\hat{\kappa}$, $\delta\boldsymbol{\sigma}^{I-II}$ and ξ using 5.17-5.26. If $\xi \leq \epsilon$ the new and updated stresses and hardening parameters are taken as $\hat{\boldsymbol{\sigma}}$ and $\hat{\kappa}$, otherwise it is necessary to reduce ΔT and repeat the calculation. The size of the next dimensionless time step is generally given by local extrapolation:

$$\Delta T = q\Delta T \tag{5.27}$$

where

$$q = 0.9\left(\frac{\epsilon}{\xi}\right)^{\frac{1}{2}} \tag{5.28}$$

The exponent 1/2 relates to the local truncation error $O\left(\Delta T^2\right)$ of the first order formulae, whereas the factor of 0.9 is introduced to reduce the number of subincrements that are likely to be rejected during the integration process. The size of the new increment is furthermore constrained as q must lie within the interval:

$$0.01 \leq q \leq 2 \tag{5.29}$$

The modified Euler scheme with active error control is summarized in Figure 5.5.

Initial state $\boldsymbol{\sigma}_0, \kappa_0, \Delta\boldsymbol{\varepsilon}, \delta\boldsymbol{\varepsilon}_0 = \Delta\boldsymbol{\varepsilon}, q = 1$

Strain increments $i = 1, 2, ..., n$

$$\delta\boldsymbol{\varepsilon}_i = \min\left[q\delta\boldsymbol{\varepsilon}_{i-1}, \Delta\boldsymbol{\varepsilon} - \sum_{j=1}^{i-1}\delta\boldsymbol{\varepsilon}_j\right]$$

Do

$$\delta\boldsymbol{\sigma}_i^I = \mathbf{C}^{ep}(\boldsymbol{\sigma}_{i-1}, \kappa_{i-1})\,\delta\boldsymbol{\varepsilon}_i$$

$$\delta\kappa_i^I = \delta\lambda_p(\boldsymbol{\sigma}_{i-1}, \kappa_{i-1}, \delta\boldsymbol{\varepsilon}_i)\,\boldsymbol{\sigma}_{i-1}\frac{\partial g}{\partial\boldsymbol{\sigma}_{i-1}}$$

$$\delta\boldsymbol{\sigma}_i^{II} = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}_{i-1} + \delta\boldsymbol{\sigma}_i^I, \kappa_{i-1} + \delta\kappa_i^I\right)\delta\boldsymbol{\varepsilon}_i$$

$$\delta\kappa_i^{II} = \delta\lambda_p\left(\boldsymbol{\sigma}_{i-1} + \delta\boldsymbol{\sigma}_i^I, \kappa_{i-1} + \delta\kappa_i^I, \delta\boldsymbol{\varepsilon}_i\right)\left(\boldsymbol{\sigma}_{i-1} + \delta\boldsymbol{\sigma}_i^I\right)\frac{\partial g}{\partial\left(\boldsymbol{\sigma}_{i-1} + \delta\boldsymbol{\sigma}_i^I\right)}$$

$$\delta\boldsymbol{\sigma}_i^{I-II} \approx \frac{1}{2}\left(\delta\boldsymbol{\sigma}_i^{II} - \delta\boldsymbol{\sigma}_i^I\right)$$

$$\hat{\boldsymbol{\sigma}}_i = \boldsymbol{\sigma}_{i-1} + \frac{1}{2}\left(\delta\boldsymbol{\sigma}_i^I + \delta\boldsymbol{\sigma}_i^{II}\right)$$

$$\hat{\kappa}_i = \kappa_{i-1} + \frac{1}{2}\left(\delta\kappa_i^I + \delta\kappa_i^{II}\right)$$

$$\xi_i = \frac{\left\|\delta\boldsymbol{\sigma}_i^{I-II}\right\|}{\left\|\hat{\boldsymbol{\sigma}}_i\right\|}$$

if $\xi_i > \epsilon$ *then* $q = \max\left[0.9\left(\frac{\epsilon}{\xi}\right)^{\frac{1}{2}}, 0.01\right]$; $\delta\boldsymbol{\varepsilon}_i = q\delta\boldsymbol{\varepsilon}_i$

Until $\xi_i \leq \epsilon$

$$q = \min\left[0.9\left(\frac{\epsilon}{\xi}\right)^{\frac{1}{2}}, 2\right]$$

Stop strain incrementation when $\sum_{i=1}^{n}\delta\boldsymbol{\varepsilon}_i = \Delta\boldsymbol{\varepsilon}$

Final state $\hat{\boldsymbol{\sigma}}_i, \hat{\kappa}_i, \mathbf{C}^{ep}(\hat{\boldsymbol{\sigma}}_i, \hat{\kappa}_i)$

Figure 5.5: *Modified Euler scheme with active error control.*

5.5.2 The Runge-Kutta-Dormand-Prince scheme with error control

Variations of the Runge-Kutta scheme is widely used for integration purposes. The classical Runge-Kutta scheme uses a fourth order integration scheme with only one strain increment for stress updating. Several higher order schemes with subincrementation have been proposed (e.g. England 1969, Fehlberg 1970, Dormand & Prince 1980, Sloan & Booker 1992). The Runge-Kutta scheme modified by Dormand & Prince (1980) is used in the following. The scheme uses a pair of fourth and fifth order formulas, where the coefficients have been chosen to estimate and control the error as accurately as possible, to estimate the new updated stresses and work hardening parameter:

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_0 + \frac{31}{540}\delta\boldsymbol{\sigma}^I + \frac{190}{297}\delta\boldsymbol{\sigma}^{III} - \frac{145}{108}\delta\boldsymbol{\sigma}^{IV} + \frac{351}{220}\delta\boldsymbol{\sigma}^V + \frac{1}{20}\delta\boldsymbol{\sigma}^{VI} \quad (5.30)$$

$$\kappa = \kappa_0 + \frac{31}{540}\delta\kappa^I + \frac{190}{297}\delta\kappa^{III} - \frac{145}{108}\delta\kappa^{IV} + \frac{351}{220}\delta\kappa^V + \frac{1}{20}\delta\kappa^{VI} \quad (5.31)$$

$$\hat{\boldsymbol{\sigma}} = \boldsymbol{\sigma}_0 + \frac{19}{216}\delta\boldsymbol{\sigma}^I + \frac{1000}{2079}\delta\boldsymbol{\sigma}^{III} - \frac{125}{216}\delta\boldsymbol{\sigma}^{IV} + \frac{81}{88}\delta\boldsymbol{\sigma}^V + \frac{5}{56}\delta\boldsymbol{\sigma}^{VI} \quad (5.32)$$

$$\hat{\kappa} = \kappa_0 + \frac{19}{216}\delta\kappa^I + \frac{1000}{2079}\delta\kappa^{III} - \frac{125}{216}\delta\kappa^{IV} + \frac{81}{88}\delta\kappa^V + \frac{5}{56}\delta\kappa^{VI} \quad (5.33)$$

where

$$\delta\boldsymbol{\sigma}^I = \mathbf{C}^{ep}(\boldsymbol{\sigma}_0, \kappa_0)\,\delta\boldsymbol{\varepsilon} \quad (5.34)$$

$$\delta\kappa^I = \delta\lambda_p(\boldsymbol{\sigma}_0, \kappa_0, \delta\boldsymbol{\varepsilon})\,\boldsymbol{\sigma}_0\frac{\partial g}{\partial\boldsymbol{\sigma}^0} \quad (5.35)$$

$$\delta\boldsymbol{\sigma}^{II} = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}^I, \kappa^I\right)\delta\boldsymbol{\varepsilon} \quad (5.36)$$

$$\delta\kappa^{II} = \delta\lambda_p\left(\boldsymbol{\sigma}^I, \kappa^I, \delta\boldsymbol{\varepsilon}\right)\boldsymbol{\sigma}^I\frac{\partial g}{\partial\boldsymbol{\sigma}^I} \quad (5.37)$$

$$\boldsymbol{\sigma}^I = \boldsymbol{\sigma}_0 + \frac{1}{5}\delta\boldsymbol{\sigma}^I$$

$$\kappa^I = \kappa_0 + \frac{1}{5}\delta\kappa^I$$

$$\delta\boldsymbol{\sigma}^{III} = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}^{II}, \kappa^{II}\right)\delta\boldsymbol{\varepsilon} \quad (5.38)$$

$$\delta\kappa^{III} = \delta\lambda_p\left(\boldsymbol{\sigma}^{II}, \kappa^{II}, \delta\boldsymbol{\varepsilon}\right)\boldsymbol{\sigma}^{II}\frac{\partial g}{\partial\boldsymbol{\sigma}^{II}} \quad (5.39)$$

$$\boldsymbol{\sigma}^{II} = \boldsymbol{\sigma}_0 + \frac{3}{40}\delta\boldsymbol{\sigma}^I + \frac{9}{40}\delta\boldsymbol{\sigma}^{II}$$

$$\kappa^{II} = \kappa_0 + \frac{3}{40}\delta\kappa^I + \frac{9}{40}\delta\kappa^{II}$$

$$\delta\boldsymbol{\sigma}^{IV} = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}^{III}, \kappa^{III}\right)\delta\boldsymbol{\varepsilon} \quad (5.40)$$

$$\delta\kappa^{IV} = \delta\lambda_p\left(\boldsymbol{\sigma}^{III}, \kappa^{III}, \delta\boldsymbol{\varepsilon}\right)\boldsymbol{\sigma}^{III}\frac{\partial g}{\partial\boldsymbol{\sigma}^{III}} \quad (5.41)$$

$$\boldsymbol{\sigma}^{III} = \boldsymbol{\sigma}_0 + \frac{3}{10}\delta\boldsymbol{\sigma}^I - \frac{9}{10}\delta\boldsymbol{\sigma}^{II} + \frac{6}{5}\delta\boldsymbol{\sigma}^{III}$$

$$\kappa^{III} = \kappa_0 + \frac{3}{10}\delta\kappa^I - \frac{9}{10}\delta\kappa^{II} + \frac{6}{5}\delta\kappa^{III}$$

$$\delta\boldsymbol{\sigma}^V = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}^{IV}, \kappa^{IV}\right)\delta\boldsymbol{\varepsilon} \quad (5.42)$$

$$\delta\kappa^V = \delta\lambda_p\left(\boldsymbol{\sigma}^{IV}, \kappa^{IV}, \delta\boldsymbol{\varepsilon}\right)\boldsymbol{\sigma}^{IV}\frac{\partial g}{\partial\boldsymbol{\sigma}^{IV}} \quad (5.43)$$

$$\boldsymbol{\sigma}^{IV} = \boldsymbol{\sigma}_0 + \frac{226}{729}\delta\boldsymbol{\sigma}^I - \frac{25}{27}\delta\boldsymbol{\sigma}^{II} + \frac{880}{729}\delta\boldsymbol{\sigma}^{III} + \frac{55}{729}\delta\boldsymbol{\sigma}^{IV}$$

$$\kappa^{IV} = \kappa_0 + \frac{226}{729}\delta\kappa^I - \frac{25}{27}\delta\kappa^{II} + \frac{880}{729}\delta\kappa^{III} + \frac{55}{729}\delta\kappa^{IV}$$

$$\delta\boldsymbol{\sigma}^{VI} = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}^V, \kappa^V\right)\delta\boldsymbol{\varepsilon} \quad (5.44)$$

$$\delta\kappa^{VI} = \delta\lambda_p\left(\boldsymbol{\sigma}^V, \kappa^V, \delta\boldsymbol{\varepsilon}\right)\boldsymbol{\sigma}^V\frac{\partial g}{\partial\boldsymbol{\sigma}^V} \quad (5.45)$$

$$\boldsymbol{\sigma}^V = \boldsymbol{\sigma}_0 - \frac{181}{270}\delta\boldsymbol{\sigma}^I + \frac{5}{2}\delta\boldsymbol{\sigma}^{II} - \frac{266}{297}\delta\boldsymbol{\sigma}^{III} - \frac{91}{27}\delta\boldsymbol{\sigma}^{IV} + \frac{189}{55}\delta\boldsymbol{\sigma}^V$$

$$\kappa^V = \kappa_0 - \frac{181}{270}\delta\kappa^I + \frac{5}{2}\delta\kappa^{II} - \frac{266}{297}\delta\kappa^{III} - \frac{91}{27}\delta\kappa^{IV} + \frac{189}{55}\delta\kappa^V$$

Even though the integration process requires six evaluations of the elastoplastic constitutive matrix, the scheme rapidly becomes competitive with the modified Euler scheme as the error tolerance is tightened. As for the modified Euler scheme the estimated relative error can be expressed as:

$$\xi = \frac{\|\hat{\boldsymbol{\sigma}} - \boldsymbol{\sigma}\|}{\|\hat{\boldsymbol{\sigma}}\|} \quad (5.46)$$

As the local truncation error in the fourth order formulae is $O\left(\Delta T^5\right)$ the exponent in 5.28 is replaced by 1/5 and the factor that controls the size of the next dimensionless time step is instead given by:

$$q = 0.9\left(\frac{\epsilon}{\xi}\right)^{\frac{1}{5}} \quad (5.47)$$

but the constraints in 5.29 still apply. The Runge-Kutta-Dormand-Prince scheme with active error control is summarized in Figure 5.6.

Initial state $\boldsymbol{\sigma}_0, \kappa_0, \Delta\boldsymbol{\varepsilon}, \delta\boldsymbol{\varepsilon}_0 = \Delta\boldsymbol{\varepsilon}, q = 1$

Strain increments $i = 1, 2, ..., n$

$$\delta\boldsymbol{\varepsilon}_i = \min\left[q\delta\boldsymbol{\varepsilon}_{i-1}, \Delta\boldsymbol{\varepsilon} - \sum_{j=1}^{i-1}\delta\boldsymbol{\varepsilon}_j\right]$$

Do

Stress increments $m = I, II, ..., VI$

$$\delta\boldsymbol{\sigma}_i^m = \mathbf{C}^{ep}\left(\boldsymbol{\sigma}_i^{m-1}, \kappa_i^{m-1}\right)\delta\boldsymbol{\varepsilon}_i$$

$$\delta\kappa_i^m = \delta\lambda_p\left(\boldsymbol{\sigma}_i^{m-1}, \kappa_i^{m-1}, \delta\boldsymbol{\varepsilon}_i\right)\boldsymbol{\sigma}_i^{m-1}\frac{\partial g}{\partial\boldsymbol{\sigma}_i^{m-1}}$$

Stop stress incrementation when $j = VI$

$$\boldsymbol{\sigma}_i = \boldsymbol{\sigma}_{i-1} + \frac{31}{540}\delta\boldsymbol{\sigma}_i^I + \frac{190}{297}\delta\boldsymbol{\sigma}_i^{III} - \frac{145}{108}\delta\boldsymbol{\sigma}_i^{IV} + \frac{351}{220}\delta\boldsymbol{\sigma}_i^V + \frac{1}{20}\delta\boldsymbol{\sigma}_i^{VI}$$

$$\kappa_i = \kappa_{i-1} + \frac{31}{540}\delta\kappa_i^I + \frac{190}{297}\delta\kappa_i^{III} - \frac{145}{108}\delta\kappa_i^{IV} + \frac{351}{220}\delta\kappa_i^V + \frac{1}{20}\delta\kappa_i^{VI}$$

$$\hat{\boldsymbol{\sigma}}_i = \boldsymbol{\sigma}_{i-1} + \frac{19}{216}\delta\boldsymbol{\sigma}_i^I + \frac{1000}{2079}\delta\boldsymbol{\sigma}_i^{III} - \frac{125}{216}\delta\boldsymbol{\sigma}_i^{IV} + \frac{81}{88}\delta\boldsymbol{\sigma}_i^V + \frac{5}{56}\delta\boldsymbol{\sigma}_i^{VI}$$

$$\hat{\kappa}_i = \kappa_{i-1} + \frac{19}{216}\delta\kappa_i^I + \frac{1000}{2079}\delta\kappa_i^{III} - \frac{125}{216}\delta\kappa_i^{IV} + \frac{81}{88}\delta\kappa_i^V + \frac{5}{56}\delta\kappa_i^{VI}$$

$$\xi_i = \frac{\|\hat{\boldsymbol{\sigma}}_i - \boldsymbol{\sigma}_i\|}{\|\hat{\boldsymbol{\sigma}}_i\|}$$

if $\xi_i > \epsilon$ *then* $q = \max\left[0.9\left(\frac{\epsilon}{\xi}\right)^{\frac{1}{5}}, 0.01\right]$; $\delta\boldsymbol{\varepsilon}_i = q\delta\boldsymbol{\varepsilon}_i$

Until $\xi_i \le \epsilon$

$$q = \min\left[0.9\left(\frac{\epsilon}{\xi}\right)^{\frac{1}{5}}, 2\right]$$

Stop strain incrementation when $\sum_{i=1}^{n}\delta\boldsymbol{\varepsilon}_i = \Delta\boldsymbol{\varepsilon}$

Final state $\hat{\boldsymbol{\sigma}}_i, \hat{\kappa}_i, \mathbf{C}^{ep}\left(\hat{\boldsymbol{\sigma}}_i, \hat{\kappa}_i\right)$

Figure 5.6: *Runge-Kutta-Dormand-Prince scheme with error control.*

5.5.3 Evaluation of integration schemes

The effectiveness of the integration schemes presented in Sections 4.2.1, 5.5.1 and 5.5.2 is examined in the following. A single material point undergoing a constant volume compression is considered. The compression is started from an anisotropic stress state of $\boldsymbol{\sigma}_0^T = [\ 450\ \ 400\ \ 400\ \ 0\ \ 0\ \ 0\ ]$. The resulting stress path is somewhat simple as it does not involve any rotation of principal axes, but even a relatively small strain increment will nevertheless lead to a considerable change in the principal stresses. The simulation is conducted by imposing a number of strain increments of equal size $\Delta\boldsymbol{\varepsilon}^T = 10^{-4}\cdot[\ 5\ \ -2.5\ \ -2.5\ \ 0\ \ 0\ \ 0\ ]$ and using the three integration schemes for determination of the corresponding change in stresses. The resulting stress path and development in hardening parameter, shown in Figure 5.7, is composed of 40 sequential strain increments of equal size. All the shown simulations are performed without correction for yield surface drift. The material parameters used are identical to those used for the examples in Chapters 7 and 8.

As seen in graph a and c the modified Euler and Runge-Kutta-Dormand-Prince integration schemes yield similar results, whereas the forward Euler integration scheme deviates visibly for decreasing values of σ_3. This deviation becomes less distinct on the linear part of the stress path, but the forward Euler scheme generally overestimates the final stress state considerably. The applicability of the forward Euler scheme, however, improves as the number of subincrements in the integration process are increased. Hence, the discrepancy is more or less reduced by a factor of two as the number of subincrements in the integration process are doubled. Similar observations hold for the development in the hardening parameter, W_p, as indicated in Figure 5.7b.

The observations above illustrate the effect of rate dependent material behaviour and a reduction of the strain increment, $\Delta\boldsymbol{\varepsilon}$, may:

1. Change the appearance of the stress path for all the methods and lead to results that are in better agreement with the 'correct' solution.
2. Reduce the discrepancy between the integration schemes.
3. Decrease in computational efficiency.

The choice of the strain increment will, therefore, essentially depend on the required accuracy of the global solution and the purpose of the integration schemes is merely to provide an accurate stress update for a given strain increment. As already seen, it is possible to obtain similar results using the three different schemes, but so far no attempt was made to evaluate their computational efficiency. The computational cost is strongly related to the evaluation of the elastoplastic constitutive matrix and as described in Sections 4.2.1-5.5.2 the forward Euler, modified Euler and the Runge-Kutta-Dormand-Prince schemes require one, two and six evaluations per subincrement, respectively. The computational costs are evaluated by performing a number of runs, where the updated stresses must lie within some tolerance, ϵ, of a reference state. Since no analytical solution is available for integrating the relations of the Single Hardening Model exactly, the reference stresses are computed by using the Rung-Kutta-Dormand-Prince scheme with 250 subincrements of equal size. The error in each strain increment for the different schemes can then be expressed as:

$$E_i = \frac{\|\boldsymbol{\sigma}_i - \boldsymbol{\sigma}_{ref}\|}{\|\boldsymbol{\sigma}_{ref}\|} \tag{5.48}$$

and the average and maximum error along the stress path are given by:

$$E_{avg} = \frac{1}{40}\sum_{i=1}^{40}E_i \tag{5.49}$$

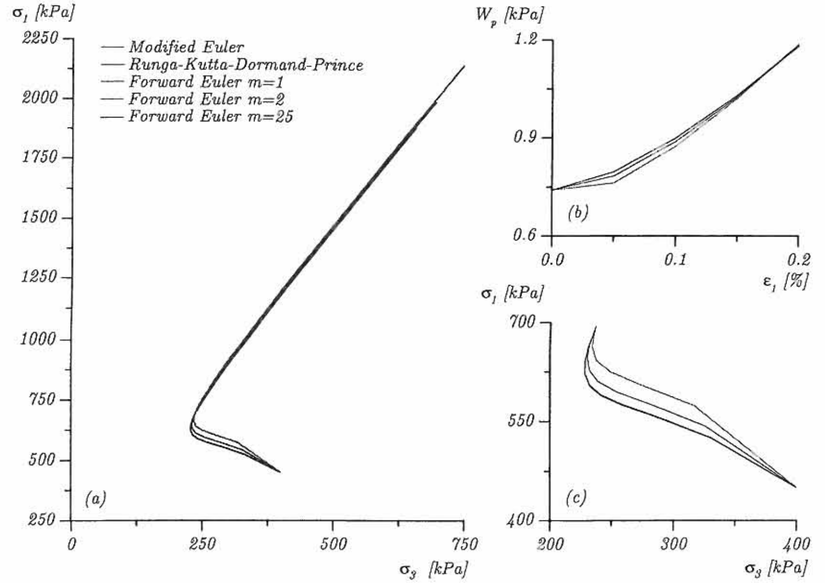

Figure 5.7: *Various integration schemes used for determination of constant volume stress path. (a) Stress path in $\sigma_3 - \sigma_1$ diagram. (b) Development of the hardening parameter, W_p, with the major principal strain. (c) Segment of stress path.*

$$E_{max} = \max_{1 \leq i \leq 40} E_i \tag{5.50}$$

The maximum number of subincrements within a single strain increment and the accumulated relative CPU time required for fulfillment of a given error tolerance along the stress path in Figure 5.7 are listed in Table 5.1. The CPU time is a measure of the computational costs due to the application of the user defined material module. For the forward Euler scheme the number of subincrements is adjusted until the average error equals the specified tolerance. The modified Euler and Runge-Kutta-Dormand-Prince schemes use the specified tolerance for adjustment of the size of the subincrements. The simulations have been performed with and without correction for yield surface drift.

As indicated in Table 5.1, the simulations reveal that the two higher order schemes have normalized errors that are substantially less than unity and are largely unaffected by correction for yield surface. However, the correction for yield surface drift becomes very important when using the forward Euler scheme as it reduces the maximum and average error in stresses with at least 70%. Moreover, if the subdivision of the strain increment is insufficient the correction for yield surface drift tends to diminish the overestimation of the final stress state shown in Figure 5.7.

As expected the forward Euler scheme shows a more or less proportional growth in the maximum number of subincrements and relative CPU time with the tightening of the error tolerance. For the modified Euler scheme the maximum number of subincrements and relative CPU time grows slightly with the reduction of the error tolerance, whereas the Runge-Kutta-Dormand-Prince scheme is barely affected. For all the schemes the first few increments (where

the stress path is changing direction) are decisive for the maximum number of subincrements needed for fulfillment of a given error tolerance. Thus, for the two higher order schemes only a few of the imposed strain increments are subjected to subincrementalization.

The Runge-Kutta-Dormand-Prince integration scheme is in general found to be superior to both the modified Euler and the forward Euler schemes in terms of accuracy and computational costs as the error tolerance is tightened. Even the modified Euler scheme is only competitive for large error tolerances.

Table 5.1: *Results for integration of the Single Hardening Model using various schemes.*

Method	E_{max}/ϵ			E_{avg}/ϵ			Maximum number of subincrements			Relative CPU time		
	$\epsilon = 10^{-3}$	$\epsilon = 10^{-4}$	$\epsilon = 10^{-5}$	$\epsilon = 10^{-3}$	$\epsilon = 10^{-4}$	$\epsilon = 10^{-5}$	$\epsilon = 10^{-3}$	$\epsilon = 10^{-4}$	$\epsilon = 10^{-5}$	$\epsilon = 10^{-3}$	$\epsilon = 10^{-4}$	$\epsilon = 10^{-5}$
FE	1.16	1.17	1.18	0.99	1.00	1.00	72	710	7091	10.8	98.5	913.9
	(0.30)	(0.34)	(0.31)	(0.22)	(0.22)	(0.23)						
ME	0.26	0.41	0.45	0.15	0.22	0.24	10	29	89	0.8	1.5	3.8
	(0.41)	(0.44)	(0.45)	(0.22)	(0.25)	(0.26)						
RKDP	0.04	0.04	0.13	0.02	0.03	0.10	1	2	3	1.0	1.0	1.0
	(0.04)	(0.04)	(0.11)	(0.01)	(0.01)	(0.08)						

Notes: Values denoted by () indicate errors in stresses after correction for yield surface drift
Forward Euler (FE), Modified Euler (ME), Rung-Kutta-Dormand-Prince (RKDP)

6 COMMUNICATION WITH ABAQUS

The finite element program ABAQUS provides an interface, whereby any constitutive relation can be added to the material library. To ensure proper functioning ABAQUS defines an interface containing a list of formal arguments for use with user defined material modules (ABAQUS 1995). The subroutine containing the constitutive relation (SHM-module) must, as already outlined in Chapter 4 and 5, provide information about the material behaviour at the end of each strain increment. The subroutine is called by ABAQUS at each Gauss point for calculation or updating of:

1. Stresses
2. Consistent tangent stiffness matrix
3. History information, i.e. values of path dependent parameters.

An introduction to the ABAQUS interface, the used syntax and some remarks on the call of user defined material modules are given in the following.

6.1 ABAQUS interface

The communication with ABAQUS is accomplished by use of a predefined interface written in FORTRAN 77 code:

```
      SUBROUTINE UMAT(STRESS,STATEV,DDSDDE,SSE,SPD,SCD,
     1RPL,DDSDDT,DRPLDE,DRPLDT,
     2STRAN,DSTRAN,TIME,DTIME,TEMP,DTEMP,PREDEF,DPRED,CMNAME,
     3NDI,NSHR,NTENS,NSTATV,PROPS,NPROPS,COORDS,DROT,PNEWDT,
     4CELENT,DFGRD0,DFGRD1,NOEL,NPT,LAYER,KSPT,KSTEP,KINC)
C
      CHARACTER*8 CNAME
      DIMENSION STRESS(NTENS),STATEV(NSTATV),
     1DDSDDE(NTENS,NTENS),
     2DDSDDT(NTENS),DRPLDE(NTENS),
     3STRAN(NTENS),DSTRAN(NTENS),TIME(2),PREDEF(1),DPRED(1),
     4PROPS(NPROPS),COORDS(3),DROT(3,3),DFGRD0(3,3),DFGRD1(3,3)
```

For temperature and time independent relations the user coding must calculate and update the variables STRESS, DDSDDE and STATEV. A description of all the variables can be found in the ABAQUS manual (ABAQUS 1995), whereas only the variables used in the present subroutine are described.

The main input variables are:

STRESS	Stress state
DSTRAN	Total strain increment
PROPS	Material and model properties
NPROPS	Number of material and model properties
STATEV	State variables
NSTATV	Number of state variables

The main output variables are:

STRESS	Updated stresses
DDSDDE	Consistent tangent stiffness matrix at the current state of stress
STATEV	Updated value of state variables

It should be noticed that the sequence of stress and strain differ from the convention normally used in continuum mechanics as the shear components are interchanged (see Chapter 2). The stresses and the total strain increment are given on vector form:

$$\boldsymbol{\sigma}^T = \left[\begin{array}{cccccc} \sigma_{11} & \sigma_{22} & \sigma_{33} & \sigma_{12} & \sigma_{13} & \sigma_{23} \end{array} \right] \tag{6.1}$$

$$\Delta\boldsymbol{\varepsilon}^T = \left[\begin{array}{cccccc} \Delta\varepsilon_{11} & \Delta\varepsilon_{22} & \Delta\varepsilon_{33} & \Delta\varepsilon_{12} & \Delta\varepsilon_{13} & \Delta\varepsilon_{23} \end{array} \right] \tag{6.2}$$

The material properties and various model control parameters are given by the property array, which is composed as follows:

PROPS(1)	Parameter a of failure criterion
PROPS(2)	Parameter m of failure criterion
PROPS(3)	Parameter η_1 of failure criterion
PROPS(4)	Parameter K or M for variation of Youngs Modulus
PROPS(5)	Parameter n or λ for variation of Youngs Modulus
PROPS(6)	Parameter ν Poisson's ratio
PROPS(7)	Parameter ψ_2 of plastic potential function
PROPS(8)	Parameter μ of plastic potential function
PROPS(9)	Parameter C of work hardening law
PROPS(10)	Parameter p of work hardening law
PROPS(11)	Parameter h of yield function
PROPS(12)	Parameter α of yield function
PROPS(13)	Parameter, p_a, atmospheric pressure
PROPS(14)	Integration scheme: (1) Modified Euler scheme (2) Runge-Kutta-Dormand-Prince scheme (3) Forward Euler scheme
PROPS(15)	Parameter for integration scheme: Stress tolerance for the Runge-Kutta-Dormand-Prince and modified Euler schemes Number of subdivisions in the forward Euler scheme
PROPS(16)	Variation of Youngs modulus (1) Relation defined by Janbu (1963) (2) Relation defined by Lade & Nelson (1987)
PROPS(17)	Correction for yield surface drift (0) Deactivated (1) Activated

PROPS(18)	Tolerance on yield criterion
PROPS(19)	Maximum number of iterations allowed for fulfillment yield criterion
PROPS(20)	Initial slope of softening branch

A number of state variables are used for storage of the materials local stress-strain history. The state variable array is composed as follows:

STATEV(1)	Current value of yield function, $f'(\boldsymbol{\sigma})$
STATEV(2)	Current yield surface, i.e. value of hardening or softening law, $f''(W_p)$
STATEV(3)	Plastic work
STATEV(4)	Flag indicating failure
STATEV(5)	Value of softening parameter, A, determined at failure
STATEV(6)	Value of softening parameter, B, determined at failure
STATEV(7)	Current stress level relative to failure, S.

6.2 Syntax for call of the SHM-module
When using the SHM-module it is necessary to define an element set that uses the specific material definition. Furthermore, the number of material and model properties, NPROPS, and the number of state variables, NSTATV, must be defined together with the material and model properties. An example of a material definition in an ABAQUS input file is given below:

```
*SOLID SECTION,ELSET=SOIL,MATERIAL=SAND
MATERIAL,NAME=SAND
USER MATERIAL,UNSYMM,CONSTANTS=20
 0.,0.2879,70.19,477.65,0.4142,0.20,-3.1375,1.9862,
 0.00013101,1.6188,0.6400,0.5548,101.4,3.,100,2.,
 1.,1.D-4,200.,1.D0
DEPVAR
 7,
USER SUBROUTINE,INPUT=shm3d.for
```

6.3 Initialization of state variables
For use with geomaterials the ABAQUS input file is composed of at least six parts:

1. Initial geometry of the problem
2. Type of elements used to approximate the displacement field
3. Specification of material model properties
4. Boundary conditions, i.e. prescribed displacements and pore pressure
5. User defined geostatic stress field
6. Load steps, i.e. loads and prescribed displacements

On execution of the ABAQUS job it is initially checked that the user defined geostatic stress field is in equilibrium. Thus, the material module is called once for each material point for establishment of the global stiffness matrix and correction of the stresses. However, the state variables, STATEV, are initially set to zero by ABAQUS and are not in accordance with the geostatic stress state. When the SHM-module is entered for the very first time the state variables must be initialized. This includes STATEV(1), STATEV(2), STATEV(3) and STATEV(7) of which the first three must be greater than zero (see page 35). The remaining three state varaibles are presumed to be zero as failure is not allowed at the initial state of stress.

6.4 Routine call with zero strain increment

At the beginning of each new load increment ABAQUS calls the SHM-module at each material point with a zero strain increment for establishment of an estimate of the global stiffness matrix. ABAQUS uses the estimated stiffness to come up with a first guess on the corresponding displacement field and hence a new strain increment. The SHM-module recognizes being called with a zero strain increment and interpret this as a request for the current tangent stiffness only. In the present situation the direction of the next load increment is unknown and in order to avoid a gross exaggeration of the displacement field in case of unloading the maximum stiffness is returned.

7 VERIFICATION OF THE SHM-MODULES BY SINGLE ELEMENT TESTS

The SHM-modules ability to function is documented by performing numerous single element analyses. The validation of the 2 and 3D versions is performed by simulation of triaxial and true triaxial, respectively. The simulations are performed in both the compression and extension regime, following various stress paths. As these types of tests imply homogeneous stress and strain states, i.e. the stresses and strains attain constant values at all Gauss points, the simulations may seem superficial. Nevertheless, these simple tests makes it easy to interpret and validate the results. All simulations are performed with the basic material parameters listed in Table 7.1. The parameters are derived from conventional triaxial compression tests on Eastern Scheldt Sand (Jakobsen & Praastrup 1998).

Table 7.1: *Material parameters used for testing of the SHM-module.*

Parameter	Value	ABAQUS variable
	Failure criterion	
a	0	PROPS(1)
m	0.2879	PROPS(2)
η_1	70.19	PROPS(3)
	Elastic parameters	
$M^\dagger$	458.45	PROPS(4)
$\lambda^\dagger$	0.4142	PROPS(5)
ν	0.20	PROPS(6)
	Plastic potential function	
ψ_2	-3.1540	PROPS(7)
μ	2.0611	PROPS(8)
	Work hardening law	
C	$1.2748 \cdot 10^{-4}$	PROPS(9)
p	1.6078	PROPS(10)
	Yield function	
α	0.6166	PROPS(11)
h	0.5525	PROPS(12)
	Softening law	
$b^\ddagger$	0.5	PROPS(20)

Notes: † Variation of Youngs modulus as defined by Lade & Nelson (1987)
‡ This parameter is varied from zero to unity in Section 7.1.1

7.1 Triaxial tests

All the triaxial simulations are performed with a 4 node axisymmetric element - element type CAX4. During all simulations the nodes along the vertical symmetry line are fixed in the radial direction, whereas the bottom nodes are fixed in the vertical direction. A sketch of the used element and the principal directions are shown in Figure 7.1.

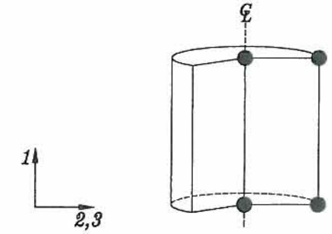

Figure 7.1: *Used element and definition of principal directions.*

The conditions of the remaining two boundaries are given during the respective load steps. The following simulations are performed for validation of the SHM-module:

- Variation of softening parameter
- Intersection of yield surface
- Back-scaling of elastic trial stresses
- Strengthening of a material due to preshearing

7.1.1 Variation of softening parameter

The effect of the softening parameter, b, is illustrated by simulation of several conventional drained triaxial compression tests. Each simulation consist of three load steps:

Load step 1: Establishment of an initial isotropic stress field of $20kPa$

Load step 2: Isotropic consolidation from 20 to $160kPa$

Load step 3: Drained compression at a constant confining pressure of $160kPa$

The simulations are performed with values of b equal to 0, 0.25, 0.5 and 1. The first corresponds to a perfectly plastic material behaviour at failure, whereas the remaining three correspond to various degrees of strength degradation after peak failure. The results of the four simulations are given in Figure 7.2.

As shown in the figure the value of b affects both the deviator stress and volumetric strain after peak failure. Thus, both the deviator stress and the rate of dilation decreases more rapidly as b increases. Whereas the effect on the stress-strain curves is distinct the effect on the volumetric behaviour is less pronounced.

The effect of b is most easily perceived by considering the value of the yield function, which is shown in Figure 7.2c. The expansion of the yield surfaces during hardening is seen to be identical for the four simulations, but as failure is reached, and the softening regime is entered, the curves starts to deviate as the yield surfaces diminish at different rates. However, in case of no softening ($b = 0$) the rate of dilation and plastic work rate remains constant throughout the simulation. As the softening parameter is increased the rate of dilation and plastic work rate decreases (see Figure 7.2d).

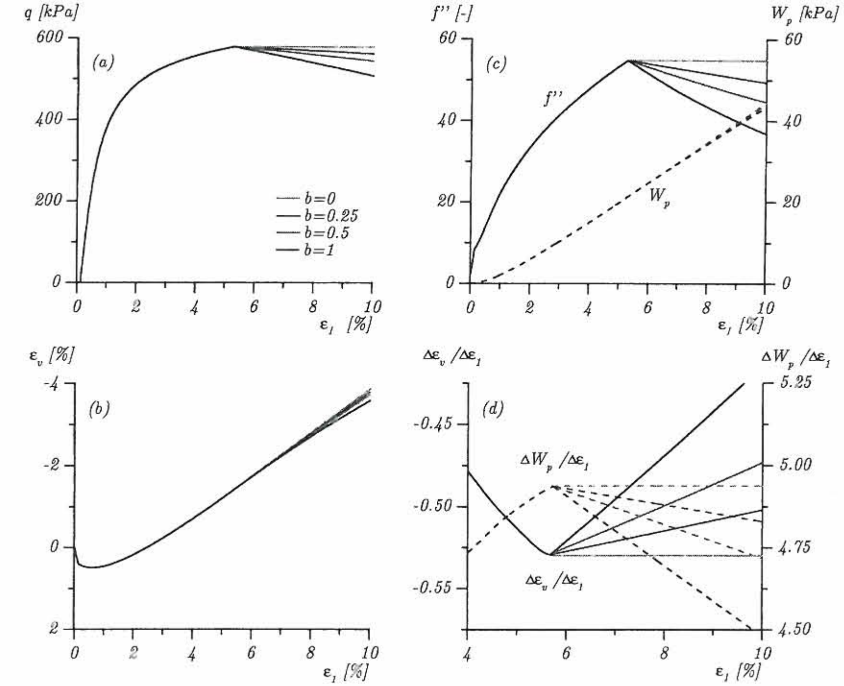

Figure 7.2: *Effect of the softening parameter b on the material degradation after peak failure. (a) Stress-strain curves. (b) Volumetric strain versus total axial strain. (c) Size of yield surface and plastic work versus total axial strain. (d) Magnification of post peak behaviour.*

7.1.2 Intersection of yield surface

The numerical procedure which deals with the intersection of a yield surface when a stress point changes from an elastic to an elastoplastic state is evaluated by simulation of a triaxial compression test with constant axial stress.

Load step 1: Establishment of an initial isotropic stress field of $20kPa$

Load step 2: Isotropic consolidation from 20 to $500kPa$

Load step 3: Drained compression at a constant axial pressure of $500kPa$

When load step 3 is started the material is subjected to an elastic loading. The material will exhibit elastic behaviour as it passes through elastic stress space until it intersects the current yield surface corresponding to the isotropic stress state applied in load step 2. The material then subsequently yields throughout the simulation. The model behaviour is shown in Figure 7.3, where the yield surface corresponding to the isotropic load step is indicated by f_2''.

The stress-strain curve in Figure 7.3a shows a pronounced change of stiffness when the yield surface is crossed. The stress level at which the stiffness change occurs corresponds accurately

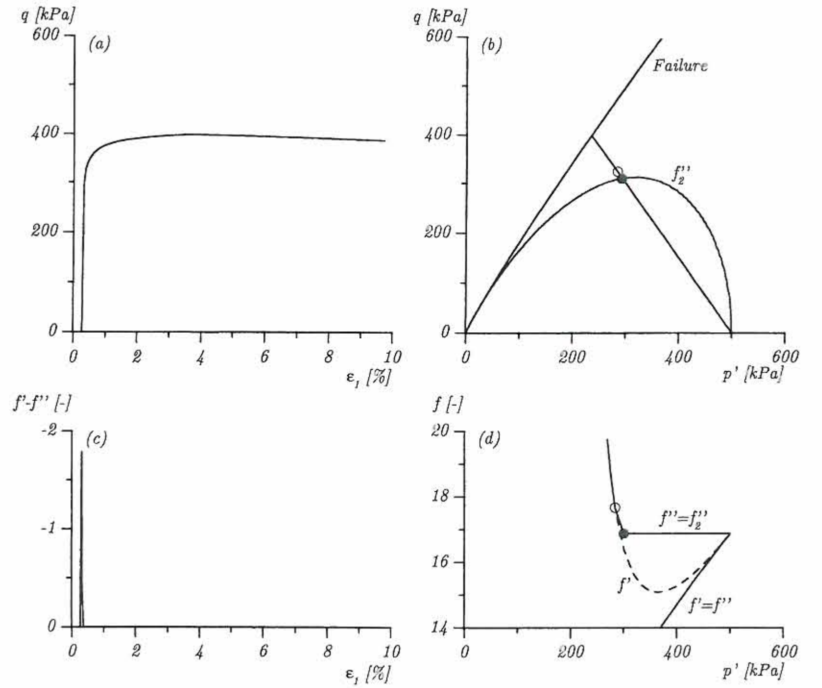

Figure 7.3: *Intersection of yield surface. (a) Stress-strain curve. (b) Stress path. (c) Value of yield criterion. (d) Size of yield surface and value of yield function.*

to the point of intersection between the stress path and the yield surface shown in 7.3b. The change in material stiffness is in 7.3c detected by the yield criterion, i.e. the condition $f < 0$ corresponds to elastic loading.

The abruptness with which the stiffness change occurs will essentially depend on the size of the strain increment. The models handling of the initial elastic unloading and subsequent elastoplastic loading is further illustrated in Figure 7.3d. During the isotropic loading step the values of the yield function and the hardening law are identical, i.e. the yield criterion is fulfilled. During load step 3 the yield function initially decreases as the elastic region is entered. As the stress point moves along the prescribed stress path and approaches the current yield surface the yield function increases, leading to a fulfillment of the yield criterion at the intersection point (closed circle) and the material subsequently yields throughout the simulation. However, as finite strain increments are used the current yield surface is intersected during an increment. The intersection is as outlined in Section 5.2 handled by performing a split into elastic and elastoplastic strain increments, but as only the final state is returned (open circle) the intersection point may not appear from the output. The intersection point can only be obtained from the output by reducing the size of the strain increment until the open and closed circles coincide.

7.1.3 Back-scaling of elastic trial stresses

The risk of attaining zero or negative elastic trial stresses is naturally greatest if one of the principal stresses at the initial stress state are close to zero. The back-scaling procedure is therefore verified by simulation of a triaxial extension test where the axial stress attains quite small values. The simulation is composed of the following three load steps.

Load step 1: Establishment of an initial isotropic stress field of $20kPa$

Load step 2: Isotropic compression from 20 to $160kPa$

Load step 3: Drained extension at a constant axial pressure of $160kPa$

The result of the simulation is shown in figure 7.4. The material will as shown in Figure

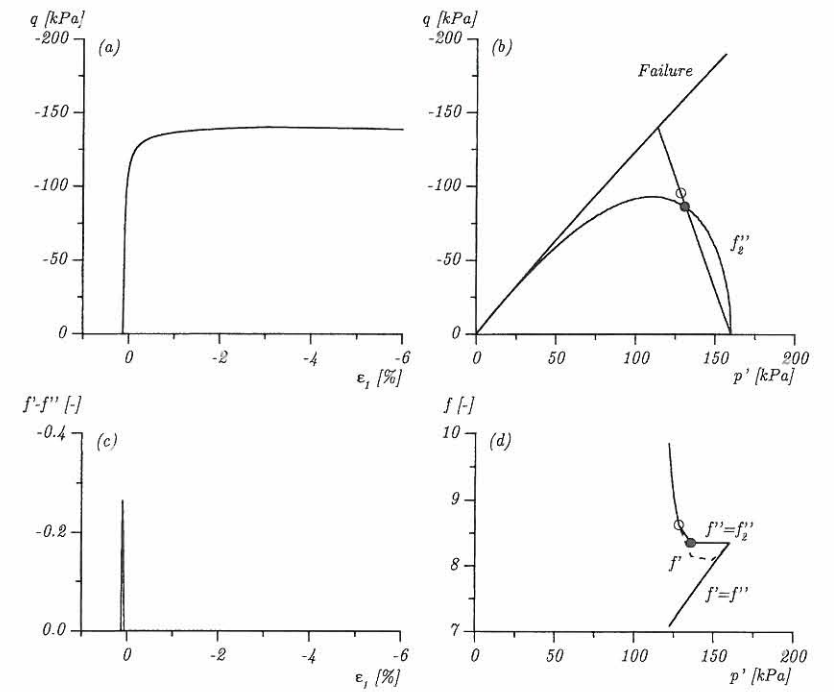

Figure 7.4: *Simulation of extension test. (a) Stress-strain curve. (b) Stress path. (c) Value of yield criterion. (d) Size of yield surface and value of yield function.*

7.4b,c and d exhibit elastic behaviour at the start of load step 3. Thus, the yield surface, denoted f_2'', is initially expanded due to the isotropic compression in load step 2. The yield surface maintains its size until it is intersected in the extension regime. The intersection and fulfillment of the yield criterion is as above-mentioned affected by the size of the imposed strain increment.

These considerations comply with the observed stress-strain behaviour shown in Figure 7.4a. During the elastoplastic loading the elastic trial stresses are repeatedly located outside the positive octant of stress space. The trial stresses are, however, scaled back properly and the theoretically calculated failure stresses are captured by the simulation.

7.1.4 Strengthening of a material due to preshearing

The problem of handling the increase in material strength due to preshearing is comparable with handling the intersection of the current yield surface. However, in the present case the current yield surface extends beyond the failure criterion and it must be ensured that failure is not detected before yielding occurs. The models capability to handle this effect is tested by simulation of a triaxial compression test, following a multi-leg stress path. The simulation is composed of the load steps given below:

Load step 1: Establishment of an initial isotropic stress field of $20kPa$

Load step 2: Isotropic consolidation from 20 to $640kPa$

Load step 3: Anisotropic consolidation by application of an axial stress of $2400kPa$

Load step 4: Unloading to an isotropic stress of $640kPa$

Load step 5: Reduction of confining pressure to $160kPa$

Load step 6: Conventional drained compression at a constant confining pressure of $160kPa$

The preshearing performed during load step 1-3 results, as shown in Figure 7.5b, in a considerable expansion of the yield surface. The value of the yield function subsequently decreases during the elastic unloading and attains its minimum value at the end of load step 5 (see Figure 7.5d.) In load step 6 the yield function increases during the elastic reloading until the current yield surface, which is located above the original failure criterion, is reached and failure is finally detected. The original and the new state of failure are marked by open and closed circles, respectively. The strengthening due to preshearing is distinct and the strength is increased from $592kPa$ to $912kPa$ corresponding to an increase in the effective friction angle from 40.4^o to 47.8^o.

The yield surface due to the preshearing has as shown in Figure 7.5b a characteristic bulge. The bulge occurs as the stress level, S, is restricted to values between zero and unity. Thus, a value of unity is obtained as the original failure criterion is fulfilled and the stress level remains constant upon further loading. This affects the value of the yield function greatly as the rate at which the the yield function changes with respect to stresses decreases (see Figure 7.5d).

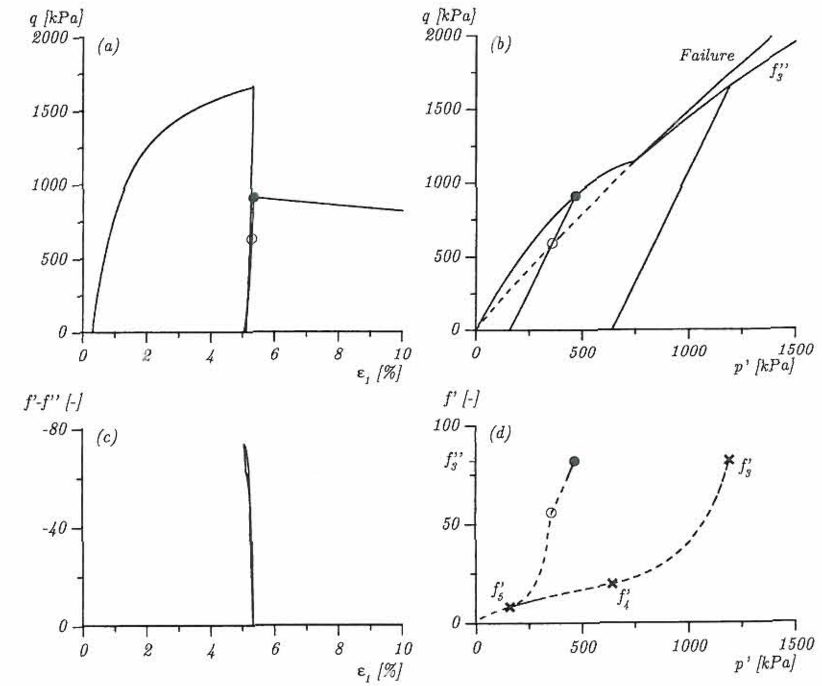

Figure 7.5: *Strengthening of material due to preshearing. (a) Stress-strain curve. (b) Stress path. (c) Value of yield criterion. (d) Value of yield function.*

7.2 *True triaxial tests*

All the true triaxial simulations are performed with a 8 node continuum element - element type C3D8. During all simulations the bottom nodes of the cube are fixed in the vertical direction. A sketch of the used element and the numbering of the three principal directions are shown in Figure 7.6.

The conditions of the remaining five boundaries are given during the respective load steps. The following simulations are performed for validation of the 3D version of the SHM-module:

- Conventional triaxial compression test
- Anisotropic consolidation and plain strain compression test

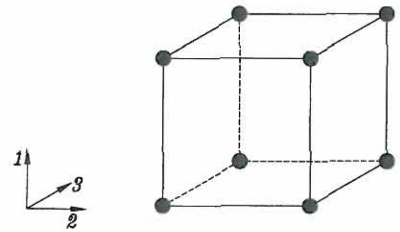

Figure 7.6: *Used element and definition of principal directions.*

7.2.1 Conventional triaxial compression test

To validate the 3D version of the SHM-module a conventional triaxial compression test similar to the one in Section 7.1.1 is performed. Hence, the simulation consist of three load steps:

Load step 1: Establishment of an initial isotropic stress field of $20kPa$

Load step 2: Isotropic consolidation from 20 to $160kPa$

Load step 3: Drained compression at a constant confining pressure of $160kPa$

The results of the true triaxial compression simulation, using the full 3D formulation are shown in Figure 7.7. The results are seen to be identical to the results from the conventional triaxial simulation based on axisymmetrical conditions given in Figure 7.2.

7.2.2 Anisotropic consolidation and plain strain compression

The use of the 3D version of the SHM-module is further illustrated by simulation of a true triaxial test, in which an anisotropic consolidation is followed by a plain strain compression.

Load step 1: Establishment of an initial isotropic stress field of $20kPa$

Load step 2: Isotropic consolidation from 20 to $160kPa$

Load step 3: Anisotropic consolidation
Direction 1: 160 to $480kPa$
Direction 2: 160 to $240kPa$
Direction 3: $160kPa$

Load step 4: Plain strain compression
Direction 2: Fixed boundaries
Direction 3: Constant axial stress of $160kPa$

The development in directional stresses and strains, plastic work etc. are shown in Figure 7.8. The stress-strain curves in Figure 7.8a shows how the imposed plain strain condition in direction 2 leads to an increase in stresses in both direction 1 and 2. The stresses subsequently decreases as failure is reached and the softening regime is entered. The corresponding stress path and volumetric strain curve are shown if Figure 7.8b and d. The material strength expressed in terms of the maximum stress difference, q, is as expected substantially higher

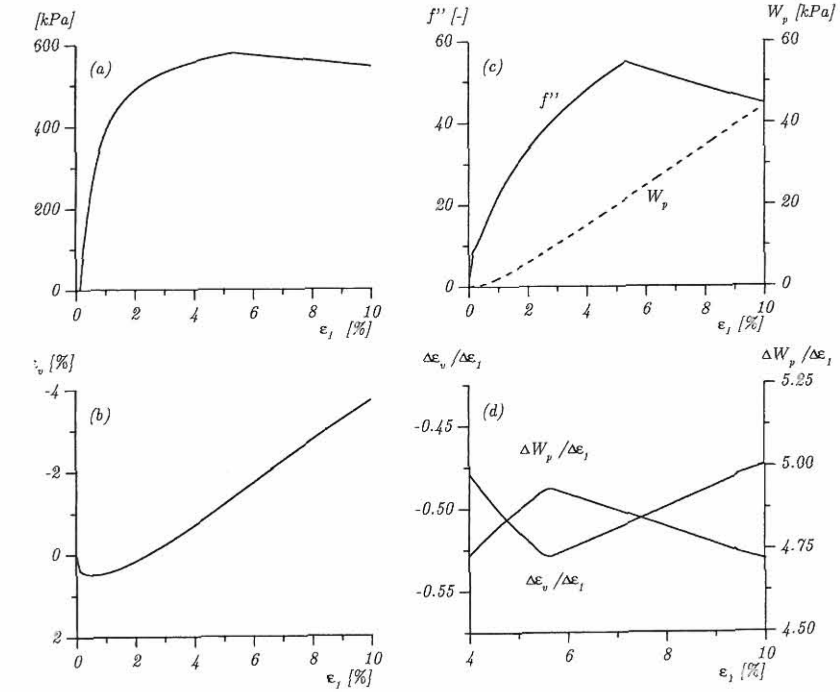

Figure 7.7: *True triaxial test. (a) Stress-strain curve. (b) Volumetric strain versus total axial strain. (c) Size of yield surface and plastic work versus total axial strain. (d) Magnification of post peak behaviour.*

than for the conventional triaxial test and the volume expansion is correspondingly reduced (see Figure 7.7).

The simulation is numerically straight forward as the material at all times is subjected to elastoplastic loading and the change from anisotropic consolidation to plain strain compression is hardly noticeable.

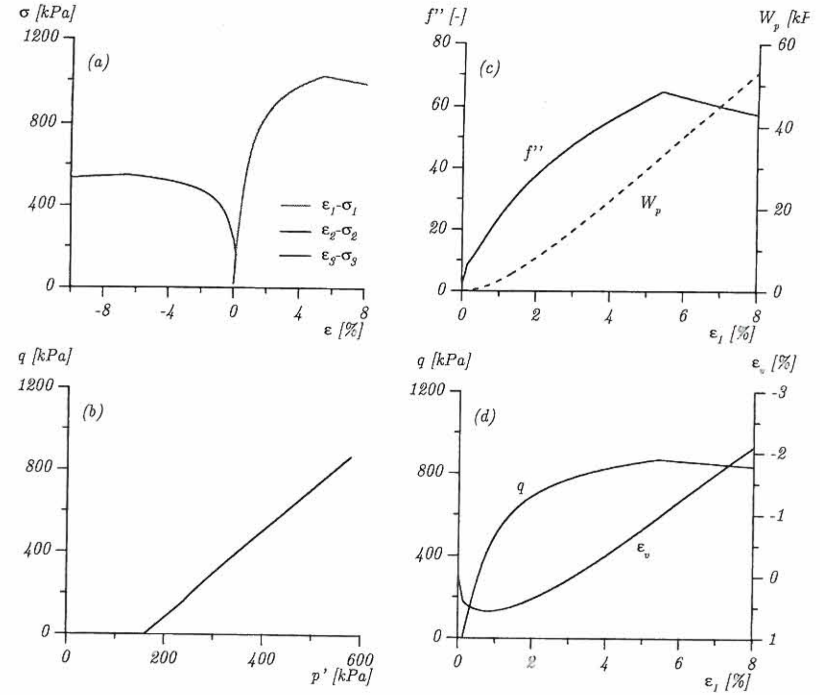

Figure 7.8: *Anisotropic consolidation and plain strain compression. (a) Stress-strain curves. (b) Stress path in the $p' - q$ plane. (c) Size of yield surface and plastic work versus major principal strain. (d) Deviator stress and volumetric strain versus major principal strain.*

8 CONCLUSIONS

The present work explains the working priciples of elastoplastic models and in particular addresses the problems involved in the implementation of the advanced Single Hardening constitutive model. The shortcommings of an earlier version of the user defined material module, UMAT, for the commercial finite element program ABAQUS are pinpointed.

The calculation strategy for a recoded version of the user defined material module, denoted the SHM-module, is presented and discussed, including the initial intersection of the yield surface, the techniques for updating of the stresses and hardening modulus, and correction for possible yield surface drift. Several integrations schemes are implemented in the module and their capabilities in relation to the Single Hardening constitutive model are evaluated. The forward Euler, modified Euler and Runge-Kutta-Dormand-Prince integration schemes have been compared in view of error tolerances and computational efficiency. The modified Euler and the Runge-Kutta-Dormand-Prince schemes, which includes active error control, posses the advantage of allowing greater strain increments to be imposed as subincrementalization automatically is performed if needed for fulfillment of a specific error tolerance. The traditional forward Euler schemes is, due to the principle of a fixed number of subincrements, not competitive with the higher order schemes, because too many subincrements are needed in order to obtain the required accuracy. However, the capability of the forward Euler scheme can be improved by adopting an iterative scheme correcting for a possible yiled surface drift. It is found that the most advantageous integration scheme for the model is the Runge-Kutta-Dormand-Prince integration scheme as it is superior in terms of both accuracy and computational costs. Eventually the SHM-modules ability to function is documented by performing numerous single element analyses.

8.1 Further improvements

The evaluation of the integration schemes illustrated how the use of finite strain increments affected the updating of stresses. It turned out that the application of integration schemes subdividing the originally imposed strain increment led to improved accuracy. The explanation of this improvement follows directly from the basis on which the elastoplastic stress-strain relation is derived. Thus, the relation is based on the assumption of infinitesimal increments in stresses and strains for which reason the path dependency of the elastoplastic tangent stiffness matrix becomes immaterial.

Both the original and the current versions of the user defined material modules returns the elastoplastic tangent stiffness evaluated from information at the end of the finite strain increment. However, to improve the overall rate of convergence of the overall equilibrium iterations the material or so-called consistent tangent stiffness matrix should be used (Simo & Taylor 1985, Crisfield 1991). Hence, the user defined material module should return a material stiffness consistent with the integration scheme used by ABAQUS for calculation of the non-linear finite strain increment.

REFERENCES

ABAQUS (1995), ABAQUS *Vers. 5.5 - Manuals*, Hibbit and Karlson and Soresen Inc.

Chen, W.F. & E. Mizuno (1990), *Non-Linear Analysis in Soil Mechanics*, Elsevier, New York.

Crisfield, M.A. (1991), *Non-Linear Finite Element Analysis of Solids and Structures*, Wiley, Chichester, U.K.

Dormand, J.R. & P.J. Prince (1980), 'A family of embedded Runge-Kutta formulae', *Journal of Computer Applied Mathematics* **6**, 19–26.

England, R. (1969), 'Error estimates for Runge-Kutta type solutions to systems of ordinary differential equations', *Computer Journal* **12**, 166–170.

Fehlberg, E. (1970), 'Klassische Runge-Kutta formeln vierter und niedrigerer ornung mit schrittweiten-kontrolle und ihre anwendungen auf wämleitungs-probleme', *Computing* **6**, 61–71.

Jakobsen, K.P. & U. Praastrup (1998), 'Drained triaxial tests on Eastern Scheldt sand', *AAU Geotechnical Engineering Papers* . ISSN 1398-6465 R9822.

Janbu, N. (1963), Soil compressibility as determined by odometer and triaxial tests, *in* 'Proceedings on the European Conference on Soil Mechanics and Foundation Engineering', Vol. 1, Wiesbaden, pp. 19–25.

Kim, M.K & P.V. Lade (1988), 'Single hardening constitutive model for frictional materials, I. Plastic potential function', *Computer and Geotechnics* (5), 307–324.

Krenk, S. (1993), *Non-Linear Analysis with Finite Elements*, Department of Building Technology and Structural Engineering, Aalborg University, Aalborg, Denmark. ISSN 0902-7513 R9333.

Lade, P.V. (1977), 'Elasto-plastic stress-strain theory for cohesionless soils with curved yield surfaces', *International Journal of Solids and Structures* **13**, 1019–1035.

Lade, P.V & M.K Kim (1988*a*), 'Single hardening constitutive model for frictional materials II. Yield criterion and plastic work contours', *Computer and Geotechnics* (6), 13–29.

Lade, P.V & M.K Kim (1988*b*), 'Single hardening constitutive model for frictional materials III. Comparisons with experimental data', *Computer and Geotechnics* (6), 31–47.

Lade, P.V. & R.B. Nelson (1984), 'Incrementalization procedure for elasto-plastic constitutive model with multiple, intersecting yield surfaces', *International Journal for Numerical and Analytical Methods in Geomechanics* **8**, 311–323.

Lade, P.V & R.B. Nelson (1987), 'Modelling the the elastic behaviour of granular materials', *International Journal for Numerical and Analytical Methods in Geomechanics* **11**, 521–542.

Malvern, L.E. (1969), *Introduction to the Mechanics of a Continuous Medium*, Prentice-Hall, Englewood Cliffs, N.J.

Potts, D.M. & A. Gens (1985), 'A critical assessement of methods of correcting for drift from the yield surface in elasto-plastic finite element analysis', *International Journal for Numerical and Analytical Methods in Geomechanics* **9**, 149–159.

Praastrup, U., K.P. Jakobsen & L.B. Ibsen (1999), 'Two theoretically consistent methods for analysing triaxial tests', *Computers and Geotechnics* (25), 157–170.

Praastrup, U., L.B. Ibsen & P.V. Lade (1999), Presentation of stress points in the customised octahedral plane, *in* 'Proceedings on the 13th ASCE Engineering Mechanics Division Conference', Baltimore.

Simo, J.C. & R.L. Taylor (1985), 'Consistent tangent operator for rate-independent elastoplasticity', *Computer Methods in Applied Mechanics and Engineering* **48**, 101–118.

Sloan, S.W. (1987), 'Substepping schemes for the numerical integration of elastoplatic stress-strain relations', *International Journal for Numerical Methods in Engineering* **24**, 893–911.

Sloan, S.W. & J.R. Booker (1992), 'Integration of Tresca and Mohr-Coulomb constitutive relations in plane stain elastoplaticity', *International Journal for Numerical Methods in Engineering* **33**, 163–196.

Spencer, A.J.M. (1980), *Continuum Mechanics*, Longmann Mathematical Texts, Essex, U.K.

Zienkiewicz, O.C. & R.L. Taylor (1991), *The Finite Element Method - Vol. 2: Solid and Fluid Dynamics and Non-Linearity*, McGraw-Hill, London, U.K.

A FLOW CHART FOR THE SHM-MODULES

The implementation of the user defined material module must ensure that ABAQUS recieves updated stresses, state variables and elastoplatic tangent stiffness that comply with the imposed strain increment.

The implementation of the SHM-module is based on the techniques described in Chapter 4 and 5. The description has to a great extent been related to the Single Hardening Model. However, the algorithms used for detection of intersection of the yield surface, correction for yield surface drift and the various methods for updating of stresses and hardening parameters haven been described in a more general manner. As the Single Hardening Model is of the work hardening type the term maximum plastic work is used in the following for description of the material hardening.

The code for the SHM-module follows the flow chart in Figure A.1 and A.2. The calculation strategy is explained in the following.

The initial state of stress, $\boldsymbol{\sigma}_0$, the maximum plastic work, $W_{p,0}$ and the imposed strain increment, $\Delta\boldsymbol{\varepsilon}$ are used to calculate a stress increment assuming the strain increment to be entirely elastic (Step 1). The assumed elastic state of stress, $\boldsymbol{\sigma}_e$, is used for calculation of $f'(\boldsymbol{\sigma}_e)$, being the value of the yield function corresponding to the new state of stress, and compared with the current yield function $f''(W_{p,0})$ calculated from the current maximum plastic work (Step 2). If the difference is less than or equal to zero, $f = f'(\boldsymbol{\sigma}_e) - f''(W_{p,0}) \leq 0$, then the new state of stress is located inside the yield surface and the imposed strain increment is indeed truly elastic. Hence, the calculation for the strain increment is completed and the SHM-module returns with the new state of stress $\boldsymbol{\sigma}_e$ and the elastic tangent stiffness, C (Step 3a).

If on the other hand $f = f'(\boldsymbol{\sigma}_e) - f''(W_{p,0}) > 0$ then the new stress state is outside the current yield surface and a portion of the strain increment is plastic. In this case it is determined whether the initial state of stress, $\boldsymbol{\sigma}_0$ is located inside or on the current yield surface. The initial stress state is used for calculation of $f'(\boldsymbol{\sigma}_0)$ and compared with the previously calculated $f''(W_{p,0})$ (Step 3b). If $f = f'(\boldsymbol{\sigma}_0) - f''(W_{p,0}) = 0$, then the initial stress state is located on the yield surface and the elastic loading ratio, α is equal to zero (Step 4a).

If $f = f'(\boldsymbol{\sigma}_0) - f''(W_{p,0}) < 0$ then the stress state is initially located inside the yield surface (Step 3b) and it it necessary to determine the elastic portion of the strain increment, expressed by the elastic loading ratio α (Step 4b). Having determined the elastic loading ratio the stress state located on the yield surface, $\boldsymbol{\sigma}_c$, and the remaining portion of the total strain increment are determined (Step 5) and used to update the stresses and the maximum plastic work (Step 6) by one of the three implemented integration schemes. 1: Forward Euler scheme with subincrementation, 2: Modified Euler scheme with error control or 3: Runge-Kutta-Dormand-Prince scheme with error control.

The updated stresses and maximum plastic work, $\boldsymbol{\sigma}_i$ and $W_{p,i}$, are used to calculate updated values for $f'(\boldsymbol{\sigma}_i)$ and $f''(W_{p,i})$ (Step 7). If the consistency condition is fulfilled to some close tolerance, $|f'(\boldsymbol{\sigma}_i) - f''(W_{p,i})| \leq \epsilon$, the SHM-module returns with the current updated stresses, $\boldsymbol{\sigma}_i$, hardening parameter, $W_{p,i}$ and elastoplastic tangent stiffness, C^{ep} (Step 8a). Otherwise the newly updated stresses and hardnig parameters must be corrected for yield surface drift and the SHM-module finally returns with corrected stresses, $\boldsymbol{\sigma}_c$, hardening parameter, $W_{p,c}$ and elastoplastic tangent stiffness, C^{ep} (Step 8b).

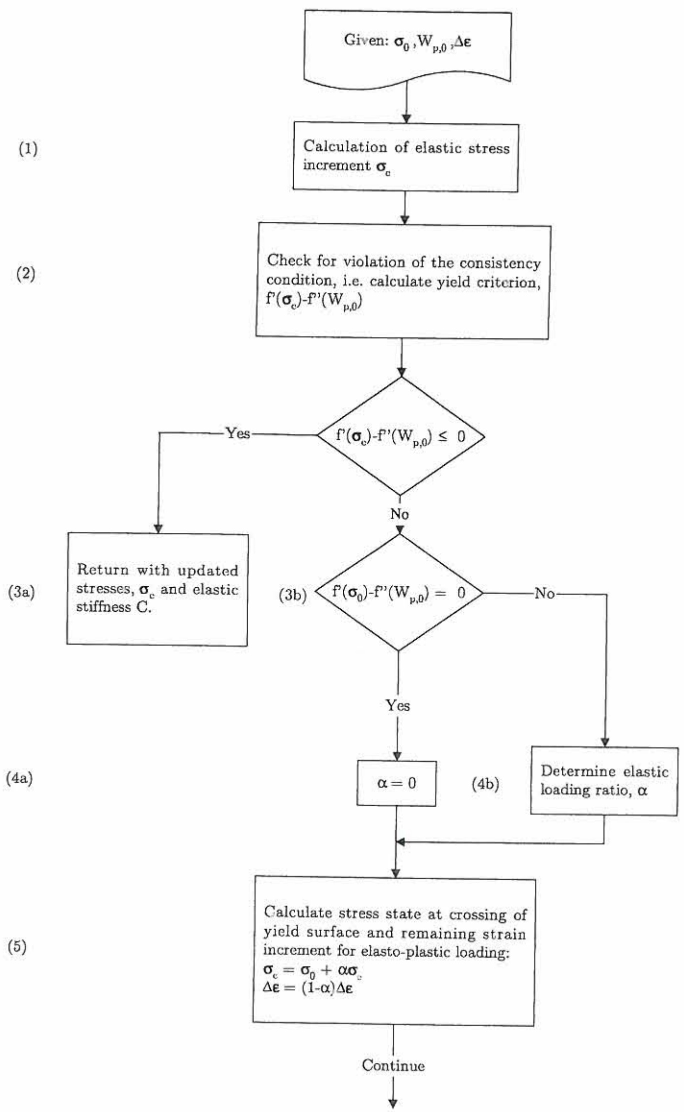

Figure A.1: *Flow chart for the SHM-module.*

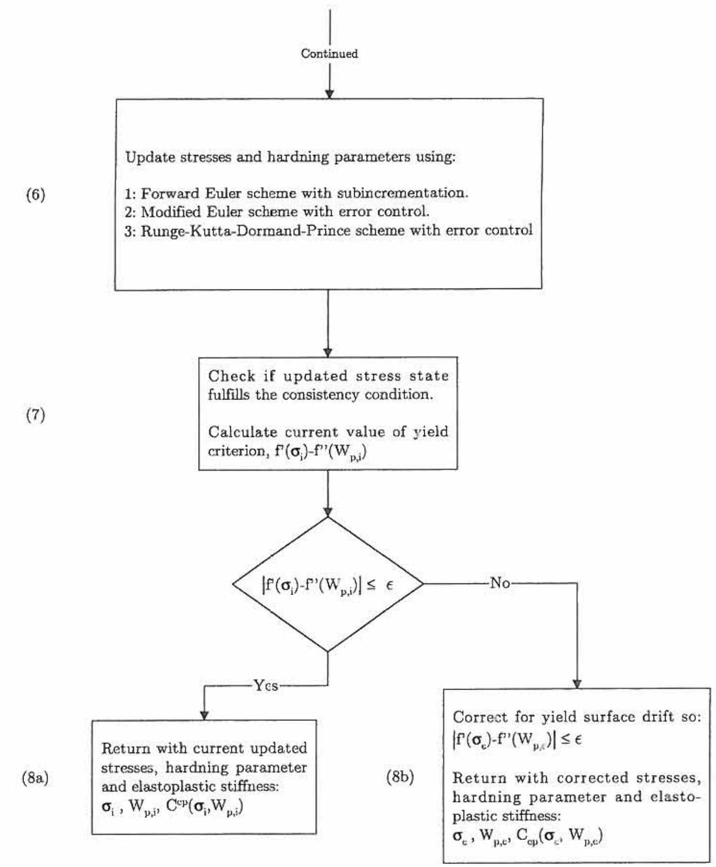

Figure A.2: *Flow chart for the SHM-module.*

B SOURCE CODE FOR THE 2D SHM-MODULE

```
      SUBROUTINE UMAT(STRESS,STATEV,DDSDDE,SSE,SPD,SCD,
     1RPL,DDSDDT,DRPLDE,DRPLDT,
     2STRAN,DSTRAN,TIME,DTIME,TEMP,DTEMP,PREDEF,DPRED,CMNAME,
     3NDI,NSHR,NTENS,NSTATV,PROPS,NPROPS,COORDS,DROT,PNEWDT,
     4CELENT,DFGRD0,DFGRD1,NOEL,NPT,LAYER,KSPT,KSTEP,KINC)

      IMPLICIT NONE
      INTEGER NPROPS,NSTATV,KSTEP,KINC,NOEL,NPT,NTENS,NDI,NSHR,
     1LAYER,KSPT
      CHARACTER*8 CMNAME
      REAL*8 STRESS(NTENS),STATEV(NSTATV),DDSDDE(NTENS,NTENS),
     1SSE,SPD,SCD,RPL,DDSDDT(NTENS),DRPLDE(NTENS),DRPLDT,
     2STRAN(NTENS),DSTRAN(NTENS),TIME(2),DTIME,TEMP,DTEMP,
     3PREDEF,DPRED,PROPS(NPROPS),COORDS(3),DROT(3,3),PNEWDT,CELENT,
     4DFGRD0(3,3),DFGRD1(3,3)


C The following predefined ABAQUS variables are used
C i:  input   o:  output   io:  in- and output
C
C DDSDDE:    Stiffness matrix (o)
C DSTRAN:    Total strain increment (i)
C KINC:      Load increment (i)
C KSTEP:     Load step (i)
C NPROPS:    Number of user defined properties (i)
C NTENS:     Number of stress and strain components (i)
C NDI:       Number of directional stress and strain components (i)
C NOEL:      Element (i)
C NPT:       Integration point (i)
C NSHR:      Number of shear stress and strain components (i)
C NSTATV:    Number of state dependent variables (i)
C PROPS:     Material and/or user defined properties (i)
C               (1) Parameter a of failure criterion
C               (2) Parameter m of failure criterion
C               (3) Parameter eta of failure criterion
C               (4) Parameter K or M for variation of Youngs Modulus
C               (5) Parameter n or lambda for variation of Youngs Modulus
C               (6) Parameter nu, Poisson's ratio
C               (7) Parameter psi2 of plastic potential function
C               (8) Parameter mu of plastic potential function
C               (9) Parameter C of work hardening law
C               (10) Parameter p of work hardening law
C               (11) Parameter h of yield function
C               (12) Parameter alpha of yield function
C               (13) Parameter, pa, athmosperic pressure
C               (14) User defined integration scheme
C               (14.1) Modified Euler (ME)
C               (14.2) Runge-Kutta-Dormand-Prince (RKDP)
C               (14.3) Forward Euler scheme (FE)
C               (15) Integration scheme parameter
C               (15.1) Stress tolerance for ME and RKDP
```

```
C               (15.2) Number of subdivisions for FE
C               (16) Variation of Youngs modulus
C               (16.1) Relation given by Janbu
C               (16.2) Relation given by Lade and Nelson
C               (17) Activate correction for yield surface drift
C               (17.0) No
C               (17.1) Yes
C               (18) Tolerance for yield surface drift
C               (19) Maximum number of iterations
C               (20) Initial slope of softening curve
C STATEV:    Initial and final state dependent variables (io)
C               (1) Current value of yield function
C               (2) Current value of hardening/softening function
C               (3) Current value of plastic work
C               (4) Failure flag
C               (5) Parameter for softening function, A
C               (6) Parameter for softening function, B
C               (7) Stress level relative to the failure function
C STRESS:    Initial and final state of stress (io)


C Function declarations
      REAL*8 CHECKSTRESS,FAILURE,NORM,YIELD,DHARD,DSOFT,POT
C CHECKSTRESS: Check for negative or zero principal stresses
C DHARD:       Calculate erivative of hardening function
C DSOFT:       Calculate derivative of softening function
C FAILURE:     Calculate stress level relative to failure
C NORM:        Calculate vector norm
C POT:         Calculate value of plastic potential function
C YIELD:       Calculate value of yield function


C Definition of variables defined within the subroutine
      INTEGER ACTIVEDRIFT,EP,I,IMAX,INTSTEP,ITRD,ITRI,J,M,METHOD,
     1REDUCTION
      REAL*8 CE(4,4),CP(4,4),DESTRESS(4),
     1DEPS(4),DF(4),DFDW,DG(4),DINVAR(4,3),DUMMY,
     2EF,EINVAR(4),ES,ESTRESS(4),FTOL,G,INVAR(4),ISTATEV(7),NSTRESS(4),
     3RATIO,STOL,TEMPSTRESS(4),TEMPSTATEV(7)


C ACTIVEDRIFT: Flag for correction for yield surface drift
C                 (0) Off
C                 (1) On
C EP:          Flag for elastic or elastoplastic loading
C                 (0) Elastic
C                 (1) Elastoplastic
C I:           Counter for loop over stress and strain arrays
C IMAX:        Maximum number of iterations allowed for correction of yield
C              surface drift and initial intersection of the yield surface
C INTSTEP:     Number of substep used in integration scheme
C ITRD:        Iterations performed for correction for yield surface drift
C ITRI:        Iterations performed for determination of yield surface intersection
```



```
C J:          Counter for loop over stress and strain arrays
C M:          Number of subdivisions in forward Euler integration scheme
C METHOD:     Selector for integration scheme
C             Modified Euler with error control
C             2) Runge-Kutta-Dormand-Prince with error control
C             3) Forward Euler with or without subincrements
C CE:         Elastic constitutive matrix
C CP:         Plastic constitutive matrix
C DF:         Derivative of yield function
C DFDW:       Derivative of hardening function
C DG:         Derivative of plastic potential function
C DINVAR:     Derivative of stress invariants
C DUMMY:      Dummy variable
C DESTRESS:   Elastic stress increment
C DEPS:       Strain increment
C EF:         Value of yield function based on the elastic trial stress
C EINVAR:     Stress invariants based on the elastic trial stress
C ES:         Stress level based on the elastic trial stress
C ESTRESS:    Elastic trial stress
C FTOL:       Tolerance for yield surface drift
C G:          Value of plastic potential surface
C INVAR:      Stress invariants
C NSTRESS:    New stress state
C RATIO:      Ratio of strain increment causing purely elastic deformations
C             or for reduction of elastic trial stress
C REDUCTION:  Number of corrections to size of substeps in the integration schemes
C RDEPS:      Elastoplastic strain increment (RDEPS=DEPS for RATIO=1)
C STOL:       Stress tolerance used with the modified Euler and the
C             Runge-Kutta-Dormand-Prince integration schemes
C TEMPSTRESS: Temporary stres vector used for establishment of the
C             material stiffness matrix
C TEMPSTATEV: Temporary state dependent variable vector used for establishment
C             of the material stiffness matrix


C Initialise user variables
      METHOD=IDNINT(PROPS(14))
      IF (METHOD.LT.3) THEN
        STOL=PROPS(15)
      ELSE
        M=IDNINT(PROPS(15))
      END IF
      ACTIVEDRIFT=IDNINT(PROPS(17))
      FTOL=PROPS(18)
      IMAX=IDNINT(PROPS(19))
C Change sign on stresses and strains
      DO I=1,4
        NSTRESS(I)=-STRESS(I)
        DEPS(I)=-DSTRAN(I)
      END DO
C Check for plane stress condition
      IF (NDI.EQ.2) THEN
        WRITE(*,*)'******************** ABAQUS RUN IS TERMINATED',
     1  '********************'
```

```
        WRITE(*,*)'A plane stress condition is not possible'
        STOP
      END IF
C Add true cohesion (shift co-ordinat system)
      IF (PROPS(1).GT.0) THEN
        DO I=1,3
          NSTRESS(I)=NSTRESS(I)+PROPS(1)*PROPS(13)
        END DO
      END IF
      CALL INVARIANTS(INVAR,NSTRESS)
C Check for zero strain increment.  If zero return with elastic stiffness
      IF (NORM(DEPS,4,1.DO).EQ.0) THEN
        CALL ELASTIC(DDSDDE,INVAR,PROPS,NPROPS)
        RETURN
      END IF
C Check whether state dependent variables have been initialised
      IF (STATEV(2).EQ.0) THEN
        CALL INITIALISE(STATEV,INVAR,PROPS,NSTATV,NPROPS)
      END IF
      CALL ELASTIC(DDSDDE,INVAR,PROPS,NPROPS)
C Perform elastic shooting
      DO I=1,4
        DESTRESS(I)=0
        DO J=1,4
          DESTRESS(I)=DESTRESS(I)+DDSDDE(I,J)*DEPS(J)
        END DO
      END DO
C Check that all principal stresses are positive
      RATIO=CHECKSTRESS(NSTRESS,DESTRESS)
      DO I=1,4
        ESTRESS(I)=NSTRESS(I)+RATIO*DESTRESS(I)
      END DO
      CALL INVARIANTS(EINVAR,ESTRESS)
      ES=FAILURE(EINVAR,PROPS,NPROPS)
      EF=YIELD(EINVAR,PROPS,ES,NPROPS)
      IF ((EF-STATEV(2)).LT.-FTOL) THEN
        DO I=1,4
          NSTRESS(I)=ESTRESS(I)
        END DO
        STATEV(1)=EF
        STATEV(7)=ES
        EP=0
      ELSE
C Check if the yield surface is crossed during loading
        EP=1
        ITRI=0
        RATIO=0.DO
        IF ((EF-STATEV(1)).GT.FTOL) THEN
          CALL INTERSECTION(NSTRESS,RATIO,STATEV,DESTRESS,
     1    EF,PROPS,FTOL,IMAX,ITRI,NSTATV,NPROPS)
          IF (ITRI.GE.IMAX) CALL DUMP(STATEV,NSTATV,1,NSTRESS,NOEL,
     1    NPT,KSTEP,KINC)
        END IF
```



```
        DO I=1,4
          DEPS(I)=(1-RATIO)*DEPS(I)
        END DO
        DO I=1,NSTATV
          ISTATEV(I)=STATEV(I)
        END DO
        INTSTEP=0
        REDUCTION=0
        IF (METHOD.EQ.1) THEN
          CALL MEULER(NSTRESS,STATEV,ISTATEV,DEPS,PROPS,STOL,INTSTEP,
     1    REDUCTION,NSTATV,NPROPS)
        ELSE IF (METHOD.EQ.2) THEN
          CALL RKDP(NSTRESS,STATEV,ISTATEV,DEPS,PROPS,STOL,INTSTEP,
     1    REDUCTION,NSTATV,NPROPS)
        ELSE IF (METHOD.EQ.3) THEN
          CALL FEULER(NSTRESS,STATEV,DEPS,PROPS,M,NSTATV,NPROPS)
        END IF
        IF (ACTIVEDRIFT.EQ.1) THEN
C The finite incremental form will generally cause yield surface drift and
C compatibility iterations are required
          IF ((ABS(STATEV(1)-STATEV(2))).GT.FTOL) THEN
            CALL INVARIANTS(INVAR,NSTRESS)
            CALL ELASTIC(CE,INVAR,PROPS,NPROPS)
            ITRD=0
            CALL DRIFT(NSTRESS,STATEV,CE,PROPS,FTOL,IMAX,ITRD,
     1      NSTATV,NPROPS)
            IF (ITRD.GE.IMAX) CALL DUMP(STATEV,NSTATV,2,STRESS,NOEL,
     1      NPT,KSTEP,KINC)
          END IF
        END IF

C Calculate elastoplastic constitutive matrix

        CALL INVARIANTS(INVAR,NSTRESS)
        CALL DINVARIANTS(DINVAR,NSTRESS)
        G=POT(INVAR,PROPS,NPROPS)
        CALL DYIELD(DF,INVAR,DINVAR,PROPS,STATEV(7),STATEV(1),NPROPS)
        CALL DPOT(DG,INVAR,DINVAR,PROPS,NPROPS)
        IF (STATEV(4).EQ.0) THEN
          DFDW=DHARD(STATEV(3),PROPS,NPROPS)
        ELSE
          DFDW=DSOFT(STATEV(5),STATEV(6),STATEV(3),PROPS(13))
        END IF

        CALL ELASTIC(CE,INVAR,PROPS,NPROPS)
        CALL PLASTIC(CP,DUMMY,CE,DF,DG,G,DFDW,PROPS(8),DEPS)

        DO I=1,4
          DO J=1,4
            DDSDDE(I,J)=CE(I,J)-CP(I,J)
          END DO
        END DO
      END IF
```

```
C Subtract an eventual added true cohesion (return to original co-ordinat system)
      IF (PROPS(1).GT.0) THEN
        DO I=1,3
          NSTRESS(I)=NSTRESS(I)-PROPS(1)*PROPS(13)
        END DO
      END IF
C Change sign on stresses and strains
      DO I=1,4
        STRESS(I)=-NSTRESS(I)
      END DO
      END


C SUBROUTINE INVARIANTS(inv,s)
C Calculate stress invariants
C
C INPUT
C   s:      Stresses [s11 s22 s33 s12]
C
C OUTPUT
C   inv:    Stress invariants [I1 I2 I3 J2]
C

      SUBROUTINE INVARIANTS(inv,s)
C Define primary variables
      IMPLICIT NONE
      REAL*8 s(4),inv(4)
      inv(1)=s(1)+s(2)+s(3)
      inv(2)=s(4)**2.D0-(s(1)*s(2)+s(2)*s(3)+s(1)*s(3))
      inv(3)=s(1)*s(2)*s(3)-s(3)*s(4)**2.D0
      inv(4)=((s(1)-s(2))**2.D0+(s(2)-s(3))**2.D0+(s(3)-s(1))**2.D0)/6
     1+s(4)**2.D0
      END


C SUBROUTINE DINVARIANTS(dinv,s)
C Calculate derivatives of stress invariants with respect to stresses.
C
C INPUT
C   s:      Stresses [s11 s22 s33 s12]
C
C OUTPUT
C   dinv:   Derivative of stress invariants [dI1 dI2 dI3]
C

      SUBROUTINE DINVARIANTS(dinv,s)
C Define primary variables
      IMPLICIT NONE
      REAL*8 dinv(4,3),s(4)
C Calculate derivative of the first stress invariant
```



```
      dinv(1,1)=1.D0
      dinv(2,1)=1.D0
      dinv(3,1)=1.D0
      dinv(4,1)=0.D0
C Calculate derivative of the second stress invariant
      dinv(1,2)=-(s(2)+s(3))
      dinv(2,2)=-(s(3)+s(1))
      dinv(3,2)=-(s(1)+s(2))
      dinv(4,2)=2.D0*s(4)
C Calculate derivative of the third stress invariant
      dinv(1,3)=s(2)*s(3)
      dinv(2,3)=s(3)*s(1)
      dinv(3,3)=s(1)*s(2)-(s(4)**2.D0)
      dinv(4,3)=-2.D0*s(3)*s(4)
      END


C SUBROUTINE INITIALISE(svar,invar,mat,nsv,np)
C Initialise state dependent variables.  The material point is initially located
C on a yield surface, wherefore the value of the stress term and hardening term
C in the yield criterion are equal
C
C INPUT
C   invar:  Stress invariants
C   mat:    Material properties
C   np:     Number of user defined properties
C   nsv:    Number of state dependent variables
C
C OUTPUT
C   svar:   State dependent variables corresponding to the initial stress state
C               (1) Value of yield function
C               (2) Value of hardening function
C               (3) Plastic work
C               (7) Stress level relative to failure
C

      SUBROUTINE INITIALISE(svar,inv,mat,nsv,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,np
      REAL*8 svar(nsv),inv(4),mat(np)
C Define secondary variables
      REAL*8 psi1,rho,d
C Function declarations
      REAL*8 FAILURE,YIELD
C Call failure function for initial value of stress level
      svar(7)=FAILURE(inv,mat,np)
C Call yield function for initial value
      svar(1)=YIELD(inv,mat,svar(7),np)
      svar(2)=svar(1)
C Calculate dependent material parameters for determination of equivalent plastic
C work
```

```
      psi1=0.00155D0*mat(2)**(-1.27D0)
      rho=mat(10)/mat(11)
      d=mat(9)/(27*psi1+3)**rho
C Calculate equivalent plastic work
      svar(3)=mat(13)*d*svar(2)**rho
      END


C FUNCTION HARD(wp,mat,np)
C Calculate the value of the hardening function for the current state of stress
C (Lade 1984, Lade & Kim 1988).
C
C INPUT
C  wp:      Plastic work
C  mat(2): Curvature parameter for failure criteria
C  mat(9): Coefficient for determination of plastic work
C  mat(10):Exponent for determination of plastic work
C  mat(12):Hardening parameter
C  mat(13):Athmospheric pressure
C  np:      Number of user defined properties
C
C OUTPUT
C  HARD:    Value of the hardening function
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C      Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C      materials II. Yield criterion and plastic work contours", Computers and
C      Geotechnics, (6), pp.  13-29.  C


      FUNCTION HARD(wp,mat,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 wp,mat(np),HARD
C Define secondary variables
      REAL*8 psi1,rho,d
C Calculate dependent parameters
      psi1=0.00155D0*mat(2)**(-1.27D0)
      rho=mat(10)/mat(11)
      d=mat(9)/(27.D0*psi1+3.D0)**rho
C Calculate value of hardening function
      HARD=(1.D0/d)**(1.D0/rho)*(wp/mat(13))**(1.D0/rho)
      END
```



```
C FUNCTION DHARD(wp,mat,np)
C Calculate the derivative of the hardening function with respect to plastic work
C at the current state of stress
C
C INPUT
C  wp:      Plastic work
C  mat(2): Curvature parameter for failure criteria
C  mat(9): Coefficient for determination of plastic work
C  mat(10):Exponent for determination of plastic work
C  mat(12):Hardening parameter
C  mat(13):Athmospheric pressure
C  np:      Number of user defined properties
C
C OUTPUT
C  DHARD:  Derivative of the hardening function
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C      Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C      materials II. Yield criterion and plastic work contours", Computers and
C      Geotechnics, (6), pp.  13-29.  C


      FUNCTION DHARD(wp,mat,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 wp,mat(np),DHARD
C Define secondary variables
      REAL*8 psi1,rho,d
C Calculate dependent parameters
      psi1=0.00155D0*mat(2)**(-1.27D0)
      rho=mat(10)/mat(11)
      d=mat(9)/(27.D0*psi1+3.D0)**rho
C Calculate derivative of the hardening function
      DHARD=(1.D0/(rho*(d*mat(13))**(1.D0/rho)))*wp**(1.D0/rho-1)
      END


C FUNCTION SOFT(a,b,wp,pa)
C Calculate the value of the softening function for the current state of stress
C after failure (Lade & Kim 1988).
C
C INPUT
C  a:       Coefficient for softening function
C  b:       Exponent for softening function
C  wp:      Plastic work
C  pa:      Athmospheric pressure
C
C OUTPUT
```

```
C  SOFT:   Value of the softening function
C
C REFERENCES
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C           materials II. Yield criterion and plastic work contours", Computers and
C           Geotechnics, (6), pp.  13-29.  C


      FUNCTION SOFT(a,b,wp,pa)
C Define primary variables
      IMPLICIT NONE
      REAL*8 a,b,wp,pa,SOFT
C Calculate value of softening function
      SOFT=a*DEXP(-b*(wp/pa))
      END


C FUNCTION DSOFT(a,b,wp,pa)
C Calculate the derivative of the softening function at the current state of stress
C (Lade & Kim 1988).
C
C INPUT
C  a:      Coefficient for softening function
C  b:      Exponent for softening function
C  wp:     Plastic work
C  pa:     Athmospheric pressure
C
C OUTPUT
C  DSOFT:  Value of the derivative of the softening function
C
C REFERENCES
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C           materials II. Yield criterion and plastic work contours", Computers and
C           Geotechnics, (6), pp.  13-29.  C


      FUNCTION DSOFT(a,b,wp,pa)
C Define primary variables
      IMPLICIT NONE
      REAL*8 a,b,wp,pa,DSOFT
C Calculate value of the derivative of the softening function
      DSOFT=-a*b/pa*DEXP(-b*(wp/pa))
      END


C FUNCTION FAILURE(inv,mat,np)
C Calculate the stress level relative to the failure criterion for the current
C state of stress (Lade 1984).
C The stress level equals unity and zero at failure and at the hydrostatic axis,
C respectively.
C
C INPUT
C  inv:    Stress invariants [I1 I2 I3 J2 S3]
```



```
C  mat(2): Curvature parameter for failure criteria
C  mat(3): Maximum function value of failure criterion
C  mat(13):  Athmospheric pressure
C  np:     Number of user defined properties
C
C OUTPUT
C  s:      Stress level relative to failure
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C     Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C


      FUNCTION FAILURE(inv,mat,n)
C Define primary variables
      IMPLICIT NONE
      INTEGER n
      REAL*8 inv(4),mat(n),FAILURE
C Calculate stress level relative to failure
      FAILURE=((inv(1)**3.D0/inv(3)-27.D0)*(inv(1)/mat(13))**mat(2))
     1/mat(3)
C The stress level must lie within the limits 0-1
      IF (FAILURE.GT.1) THEN
        FAILURE=1.D0
      ELSE IF (FAILURE.LT.0) THEN
        FAILURE=0.D0
      END IF
      END


C FUNCTION YIELD(inv,mat,s)
C Calculate the value of the yield function for the current state of stress
C (Lade 1984, Lade & Kim 1988).
C
C INPUT
C  inv:    Stress invariants [I1 I2 I3 J2]
C  mat(2): Curvature parameter for failure criteria
C  mat(3): Maximum function value of failure criterion
C  mat(11):Curvature of the yield criterion
C  mat(12):Hardening parameter
C  mat(13):Athmospheric pressure
C  s:      Stress level relative to failure
C  np:     Number of user defined properties
C
C OUTPUT
C  YIELD:  Value of the yield function
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C     Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C     materials II. Yield criterion and plastic work contours", Computers and
```

```
C      Geotechnics, (6), pp.  13-29.  C


      FUNCTION YIELD(inv,mat,s,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 inv(4),s,mat(np),YIELD
C Define secondary variables
      REAL*8 q,psi1
C Calculate dependent parameter
      q=mat(12)*s/(1.D0-(1.D0-mat(12))*s)
      psi1=0.00155D0*mat(2)**(-1.27D0)
C Calculate value of yield function
      YIELD=(psi1*inv(1)**3.D0/inv(3)-inv(1)**2.D0/inv(2))*(inv(1)/
     1mat(13))**mat(11)*DEXP(q)
      END


C SUBROUTINE DYIELD(df,inv,dinv,mat,s,f,np)
C Calculate the derivative of the yield function at the current state of stress.
C (Lade 1984, Lade & Kim 1988).
C
C INPUT
C  inv:     Stress invariants [I1 I2 I3 J2]
C  dinv:    Derivatives of stress invariants [dI1 dI2 dI3]
C  mat(2):  Curvature parameter for failure criteria
C  mat(3):  Maximum function value of failure criterion
C  mat(11): Curvature of the yield criterion
C  mat(12): Hardening parameter
C  mat(13): Athmospheric pressure
C  s:       Stress level relative to failure
C  f:       Value of yield function
C  np:      Number of user defined properties
C
C OUTPUT
C  df:      Derivative of the yield function
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C     Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C     materials II. Yield criterion and plastic work contours", Computers and
C     Geotechnics, (6), pp.  13-29.  C


      SUBROUTINE DYIELD(df,inv,dinv,mat,s,f,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 inv(4),dinv(4,3),mat(np),s,f,df(4)
C Define secondary variables
      INTEGER i
```



```
      REAL*8 q,psi1,dqdI1,dqdI3,dfdI1,dfdI2,dfdI3
C Calculate dependent parameters
      q=mat(12)*s/(1.D0-(1.D0-mat(12))*s)
      psi1=0.00155D0*mat(2)**(-1.27D0)
C Derivative of the exponent q with respect to the first stress invariant
      dqdI1=mat(12)/(mat(3)*(1.D0-(1.D0-mat(12))*s)**2.D0)*(mat(2)*s*
     1mat(3)/inv(1)+(3.D0*inv(1)**2.D0/inv(3))*(inv(1)/mat(13))**mat(2))
C Derivative of the exponent q with respect to the third stress invariant
      dqdI3=-mat(12)/(mat(3)*(1.D0-(1.D0-mat(12))*s)**2.D0)*inv(1)**3.D0
     1/inv(3)**2.D0*(inv(1)/mat(13))**mat(2)
C Derivative of the yield function with respect to the first stress invariant
      dfdI1=((3.D0+mat(11))/inv(1)+dqdI1)*f+inv(1)/inv(2)*(inv(1)/
     1mat(13))**mat(11)*DEXP(q)
C Derivative of the yield function with respect to the second stress invariant
      dfdI2=inv(1)**2.D0/inv(2)**2.D0*(inv(1)/mat(13))**mat(11)*DEXP(q)
C Derivative of the yield function with respect to the third stress invariant
      dfdI3=f*dqdI3-psi1*inv(1)**3.D0/inv(3)**2.D0*(inv(1)/mat(13))
     1**mat(11)*DEXP(q)
C The derivative of the yield function with respect to stress is obtained by use
C of the chain rule
      DO i=1,4
        df(i)=dfdI1*dinv(i,1)+dfdI2*dinv(i,2)+dfdI3*dinv(i,3)
      END DO
      END


C SUBROUTINE ELASTIC(C,inv,mat,np)
C Calculate the isotropic elastic stiffness matrix for the current state of stress
C (Janbu 1963, Lade & Nelson 1988).
C
C INPUT
C  inv:     Stress invariants [I1 I2 I3 J2]
C  mat(4):  Curvature parameter for variation of Youngs modulus
C  mat(5):  Coefficient for variation of youngs modulus
C  mat(6):  Poissons ratio
C  mat(13): Athmosperic pressure:  mat(13)
C  mat(16): User parameter determining the variation of Youngs moduls
C              (1) Variation given by Janbu (1963)
C              (2) Variation given by Lade & Nelson (1987)
C  np:      Number of user defined properties
C
C OUTPUT
C  C:       Isotropic elastic stiffness matrix
C
C REFERENCES
C  Janbu, N. (1963) "Soil compressibility as determined by odeometer and triaxial
C     tests" in Proceedings of European Conference on Soil Mechanics and
C     Foundation Engineering, Vol.  1, Wiesbaden, pp.  19-25.
C  Lade, P.V. & R.B. Nelson (1987) "Modelling the elastic behaviour of granular
C     materials",International Journal for Numerical and Analytical Methods in
C     Geomechanics, 11, pp.  521-542.
C
```

```
      SUBROUTINE ELASTIC(C,inv,mat,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 inv(4),mat(np),C(4,4)
C Define secondary variables
      INTEGER i,j
      REAL*8 E,R,lambda,mu,ps(3)
C Calculate Young's modulus at current state of stress
      IF (mat(16).EQ.1) THEN
        CALL PRINCE(ps,inv)
        E=mat(4)*mat(13)*(ps(3)/mat(13))**mat(5)
      ELSE
        R=6.D0*(1.D0+mat(6))/(1.D0-2.D0*mat(6))
        E=mat(4)*mat(13)*((inv(1)/mat(13))**2.D0+R*inv(4)/mat(13)**
     1  2.D0)**mat(5)
      END IF
C Calculate Lames constants
      lambda=E/(3.D0*(1.D0-2.D0*mat(6)))
      mu=E/(2.D0*(1.D0+mat(6)))
C Assemble elastic stiffness matrix
      DO i=1,4
        DO j=1,4
          IF (i.EQ.j) THEN
            IF (i.LE.3) THEN
              C(i,i)=lambda+2.D0*mu
            ELSE
              C(i,i)=mu
            END IF
          ELSE IF ((i.LE.3).AND.(j.LE.3)) THEN
            C(i,j)=lambda
          ELSE
            C(i,j)=0.D0
          END IF
        END DO
      END DO
      END


C SUBROUTINE PRINCE(ps,inv)
C Calculate principal stresses
C
C INPUT
C   inv:    Stress invariants [I1 I2 I3 J2]
C
C OUTPUT
C   ps:     Principal stresses
C


      SUBROUTINE PRINCE(ps,inv)
C Define primary variables
      IMPLICIT NONE
```



```
      REAL*8 inv(4),ps(3)
C Define secondary variables
      REAL*8 PI,y(3),z(2),alpha
      INTEGER j
      PI=4*DATAN(1.0D0)
      z(1)=-(inv(1)**2.D0)/3.D0-inv(2)
      z(2)=-2.D0*(inv(1)**3.D0)/27.D0-inv(1)*inv(2)/3.D0-inv(3)
      IF (z(1).GE.0) THEN
        DO j=1,3
          y(j)=0.D0
        END DO
      ELSE
        alpha=(-z(2)/2.D0)/(DSQRT((-z(1)/3.D0)**3.D0))
        IF(alpha.GT.1.D0) alpha=1.D0
        IF(alpha.LT.-1.D0) alpha=-1.D0
        alpha=DACOS(alpha)
        y(1)=2.D0*(DSQRT(-z(1)/3.D0))*DCOS(alpha/3.D0)
        y(2)=-2.D0*(DSQRT(-z(1)/3.D0))*DCOS((alpha+PI)/3.D0)
        y(3)=-2.D0*(DSQRT(-z(1)/3.D0))*DCOS((alpha-PI)/3.D0)
      END IF
      ps(1)=y(1)+inv(1)/3.D0
      ps(2)=y(2)+inv(1)/3.D0
      ps(3)=y(3)+inv(1)/3.D0
      END


C FUNCTION CHECKSTRESS(s0,dse)
C Check for negative or zero principal stresses.  If all principal stresses are
C greater than zero the elastic trial stress is accepted.  Otherwise, perform
C backscaling of stresses into the positive octant.
C
C INPUT
C   s0:     Initial stress state
C   dse:    Elastic stress increment due to elastic shooting
C
C OUTPUT
C   minratioRatio of elastic stress increment
C


      FUNCTION CHECKSTRESS(s0,dse)
C Define primary variables
      IMPLICIT NONE
      REAL*8 s0(4),dse(4),CHECKSTRESS
C Define secondary varaibles
      INTEGER i
      REAL*8 se(4),ps(3),pse(3),inv(4),ratio,minpse,minratio
C Calculate principal stresses for initial state of stress
      CALL INVARIANTS(inv,s0)
      CALL PRINCE(ps,inv)
C Calculate principal stresses for stress state obtained by elastic shooting
      DO i=1,4
        se(i)=s0(i)+dse(i)
```

```
      END DO
      CALL INVARIANTS(inv,se)
      CALL PRINCE(pse,inv)
C Check for negative principal stresses
      minpse=1
      DO i=1,3
        IF (pse(i).LT.minpse) minpse=pse(i)
      END DO
      IF (minpse.GT.0) THEN
C All principal stresses are positive.  Exit with minratio equal to unity
        CHECKSTRESS=1
      ELSE
        minratio=1
        DO i=1,3
          ratio=ps(i)/(ps(i)-pse(i))
          IF ((ratio.LT.CHECKSTRESS).AND.(ratio.GT.0))
     1      minratio=ratio
        END DO
        CHECKSTRESS=0.9999*minratio !  Scaleback stresses to the first octant
      END IF
      END


C FUNCTION POT(inv,mat,np)
C Calculate the value of the plastic potential function for the current state of
C stress (Lade 1984, Kim & Lade 1988).
C
C INPUT
C  inv:      Stress invariants [I1 I2 I3 J2 S3]
C  mat(2):   Curvature parameter for failure criteria
C  mat(7):   Plastic potential functions intersection with the hydrostatic axis
C  mat(8):   Curvature parameter for plastic potential function
C  mat(13):Athmospheric pressure
C  np:       Number of user defined properties
C
C OUTPUT
C  POT:      Value of the plastic potential function
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C       Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C       materials I. Plastic potential function", Computers and Geotechnics, (5),
C       pp.  307-324.
C


      FUNCTION POT(inv,mat,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 POT,inv(4),mat(np)
C Define secondary variables
```



```
      REAL*8 psi1
C Calculate dependent material parameter
      psi1=0.00155D0*mat(2)**(-1.27D0)
C Calculate value of the plastic potential function
      POT=(psi1*inv(1)**3.D0/inv(3)-inv(1)**2.D0/inv(2)+mat(7))*(inv(1)
     1/mat(13))**mat(8)
      END


C SUBROUTINE DPOT(dg,inv,dinv,mat,np)
C Calculate the derivative of the plastic potential function
C
C INPUT
C  inv:      Stress invariants [I1 I2 I3 J2 S3]
C  dinv:     Derivatives of stress invariants [dI1 dI2 dI3]
C  mat(2):   Curvature parameter for failure criteria
C  mat(7):   Plastic potential functions intersection with the hydrostatic axis
C  mat(8):   Curvature parameter for plastic potential function
C  mat(13):Athmospheric pressure
C  np:       Number of user defined properties
C
C OUTPUT
C  dg:       Derivative of the plastic potential function
C
C REFERENCES
C  P.V.  Lade (1984) "Failure criterion for friction materials", In Mechanics of
C       Engineering Materials, C.S. Desai & R.H. Gallagher (eds), Wiley.
C  M.K. Kim & P.V. Lade (1988) "Single hardening constitutive model for frictional
C       materials I. Plastic potential function", Computers and Geotechnics, (5),
C       pp.  307-324.
C


      SUBROUTINE DPOT(dg,inv,dinv,mat,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER np
      REAL*8 dg(4),inv(4),dinv(4,3),mat(np)
C Define secondary variables
      INTEGER i
      REAL*8 psi1,dgdI1,dgdI2,dgdI3
C Calculate dependent material parameter
      psi1=0.00155D0*mat(2)**(-1.27D0)
C Derivative of the potential function with respect to the first stress invariant
      dgdI1=(psi1*(mat(8)+3.D0)*inv(1)**2.D0/inv(3)-(mat(8)+2.D0)*inv(1)
     1/inv(2)+mat(8)*mat(7)/inv(1))*(inv(1)/mat(13))**mat(8)
C Derivative of the potential function with respect to the second stress invariant
      dgdI2=inv(1)**2.D0/inv(2)**2.D0*(inv(1)/mat(13))**mat(8)
C Derivative of the potential function with respect to the third stress invariant
      dgdI3=-psi1*inv(1)**3.D0/inv(3)**2.D0*(inv(1)/mat(13))**mat(8)
C The derivative of the potential function with respect to stress is obtained by
C use of the chain rule
      DO i=1,4
```

```
      dg(i)=dgdI1*dinv(i,1)+dgdI2*dinv(i,2)+dgdI3*dinv(i,3)
    END DO
    END


C SUBROUTINE PLASTIC(Cp,dl,Ce,df,dg,g,dfw,mu,deps)
C Calculate the plastic stiffness matrix and the plastic proportionality factor.
C
C INPUT
C  Ce:     Elastic constitutive matrix
C  df:     Derivative of yield function with respect to stresses
C  dg:     Derivative of the plastic potential function with respect to stresses
C  g:      Value of plastic potential function
C  dfw:    Derivative of hardening or softening law with respect to plastic work
C  mu:     Exponent in plastic potential function
C  deps:   Strain increment
C
C OUTPUT
C  Cp:     Plastic constitutive matrix
C  dl:     Plastic proportionality factor
C

      SUBROUTINE PLASTIC(Cp,dl,Ce,df,dg,g,dfw,mu,deps)
C Define primary variables
      IMPLICIT NONE
      REAL*8 Cp(4,4),dl,Ce(4,4),df(4),dg(4),g,dfw,mu,deps(4)
C Define secondary variables
      INTEGER i,j
      REAL*8 H,Cedg(4),Cedf(4),dfCedg,dfCedeps,Cedeps(4)
C Calculate hardening modulus
      H=dfw*mu*g
C Calculate tensor product (Ce)(dg)
      DO i=1,4
        Cedg(i)=0.D0
      END DO
      DO i=1,4
        DO j=1,4
          Cedg(i)=Cedg(i)+Ce(i,j)*dg(j)
        END DO
      END DO
C Calculate scalar (df)(Ce)(dg)
      dfCedg=0.D0
      DO i=1,4
        dfCedg=dfCedg+Cedg(i)*df(i)
      END DO
C Calculate tensor product (Ce)(df)
      DO i=1,4
        Cedf(i)=0.D0
      END DO
      DO i=1,4
        DO j=1,4
          Cedf(i)=Cedf(i)+Ce(i,j)*df(j)
```



```
        END DO
      END DO
C Assemble the plastic constitutive matrix
      DO i=1,4
        DO j=1,4
          Cp(i,j)=Cedg(i)*Cedf(j)/(dfCedg+H)
        END DO
      END DO
C Calculate tensor product (Ce)(deps)
      DO i=1,4
        Cedeps(i)=0.D0
      END DO
      DO i=1,4
        DO j=1,4
          Cedeps(i)=Cedeps(i)+Ce(i,j)*deps(j)
        END DO
      END DO
C Calculate scalar (df)(Ce)(deps)
      dfCedeps=0.D0
      DO i=1,4
        dfCedeps=dfCedeps+Cedeps(i)*df(i)
      END DO
C Calculate plastic proportionality factor
      dl=DMAX1(0.D0,dfCedeps/(dfCedg+H))
      END


C SUBROUTINE INTERSECTION(s,alpha,svar,dse,fe,mat,tol,im,i,np, nsv)
C Determine ratio of strain increment causing purely elastic deformations,
C whenever the current yield surface is crossed during loading (Sloan 1987,
C Chen & Mizuno 1990, Jakobsen 2001).
C
C INPUT
C  svar:   State dependent variables
C             (1) Current value of yield function
C             (2) Current value of hardening or softening function
C             (7) Stress level relative to failure
C  s:      Initial stress state
C  dse:    Elastic stress increment due to elastic shooting
C  fe:     Value of yield function after elastic shooting (initial value)
C  mat:    Material properties
C  tol:    Tolerance
C  im:     Maximum iterations allowed
C  np:     Number of user defined properties
C  nsv:    Number of state dependent variables
C
C OUTPUT:
C  s:      Final stress state
C  alpha:  Ratio of strain increment causing purely elastic deformation
C  svar:   Updated values of state dependent variables
C             (1):  Final value of yield function (this should equal svar(2))
C             (7):  Final stress level
C  i:      Iterations performed
```

```
C
C REFERENCES:
C  Sloan, S.W. (1987) "Substepping schemes for the numerical integration of
C     elastoplastic stress-strain relations", International Journal for Numerical
C     methods in Engineering, 24, pp.  893-911.
C  Chen, W.F. & E. Mizuno (1990) "Non-linear Analysis in Soil Mechanics",
C     Elsevier, New York.
C


      SUBROUTINE INTERSECTION(s,alpha,svar,dse,fe,mat,tol,im,i,np,nsv)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,np,im,i
      REAL*8 s(4),alpha,svar(nsv),s0(4),dse(4),fe,mat(np),tol
C Define secondary variables
      INTEGER j
      REAL*8 inv(4),dinv(4,3),dfds(4),dalpha,error,rho,psi1,d,
     1dfdse,FAILURE,YIELD,HARD
C Calculate first estimate of elastic strain increment
      alpha=-(svar(1)-svar(2))/(fe-svar(1))
C Perform compatibility check for first estimat
      DO j=1,4
        s0(j)=s(j)
        s(j)=s0(j)+alpha*dse(j)
      END DO
      CALL INVARIANTS(inv,s)
      svar(7)=FAILURE(inv,mat,np)
      svar(1)=YIELD(inv,mat,svar(7),np)
      error=svar(1)-svar(2)
C Compatibility not obtained, perform iterations
      i=0
      DO WHILE ((abs(error).GT.tol).AND.(i.LT.im))
        i=i+1
C Calculate derivatives of stress invariants
        CALL DINVARIANTS(dinv,s)
C Calculate derivatives of the yield function with respect to stresses
        CALL DYIELD(dfds,inv,dinv,mat,svar(7),svar(1),np)
C Calculate vector product dfds dse
        dfdse=0.D0
        DO j=1,4
          dfdse=dfdse+dfds(j)*dse(j)
        END DO
        dalpha=-error/dfdse
        alpha=alpha+dalpha
C Calculate updated stresses and perform tolerance check
        DO j=1,4
          s(j)=s0(j)+alpha*dse(j)
        END DO
        CALL INVARIANTS(inv,s)
        svar(7)=FAILURE(inv,mat,np)
        svar(1)=YIELD(inv,mat,svar(7),np)
        error=svar(1)-svar(2)
      END DO
```



```
C Check for failure, this could be the effect of preshearing
      IF ((svar(7).EQ.1.D0).AND.(IDNINT(svar(4)).EQ.0)) THEN
        svar(2)=HARD(svar(3),mat,np)
        svar(4)=1.D0
        psi1=0.00155D0*mat(2)**(-1.27D0)
        rho=mat(10)/mat(11)
        d=mat(9)/(27.D0*psi1+3)**rho
        svar(6)=mat(20)/rho*(1.D0/d)**(1.D0/rho)*(svar(3)/mat(13))
     1  **(1.D0/rho-1.D0)*1.D0/svar(2)
        svar(5)=svar(2)*DEXP(svar(6)*svar(3)/mat(13))
      END IF
      END



C SUBROUTINE DRIFT(s,svar,C,mat,tol,im,i,nsv,np)
C Correction for yield surface drift.  The function uses the correction method
C given by Potts & Gens (1985), which is further described in Jakobsen (2001).
C
C INPUT
C   s:      Initial stress state
C   svar:   Initial value of state dependent variables
C               (1) Initial value of yield function
C               (2) Initial value of hardening function
C               (3) Initial plastic work
C               (4) Failure indicator
C               (5-6) Parameters for softening law
C               (7) Initial stress level
C   C:      Elastic constitutive matrix
C   mat:    Material properties
C   tol:    Tolerance
C   im:     Maximum number of iterations
C   np:     Number of user defined properties
C   nsv:    Number of state dependent variables
C
C OUTPUT
C   s:      Final stress state
C   svar:   Updated state dependent variables
C               (1) Final value of yield function
C               (2) Final value of hardening function
C               (3) Final plastic work
C               (7) Final stress level
C   i:      Iterations performed
C
C REFERENCES:
C  Potts, D.M. & A. Gens (1985) "A critical assessement of methods of correcting
C     for drift from the yield surface in elasto-plastic finite element analysis",
C     International Journal for Numerical an Analytical Methods in Geomechanics, 9,
C     pp.  149-159.
C  Jakobsen, K.P (2001) "Application of the Single Hardening model in ABAQUS"
C     AGEP0000
C
```

```
      SUBROUTINE DRIFT(s,svar,C,mat,tol,im,i,nsv,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,np,i,im
      REAL*8 s(4),svar(nsv),mat(np),tol,C(4,4)
C Define secondary variables
      INTEGER j,r,cutback
      REAL*8 inv(4),dinv(4,3),ds(4),dfds(4),dgds(4),dfdw,beta1,
     1beta2,error1,error2,dwp,Cdg(4),dfCdg,sdg,depsp(4)
C Function declarations
      REAL*8 FAILURE,YIELD,SOFT,HARD,DHARD,DSOFT
C Initialise local variables
      i=0
      cutback=0
      error1=0.D0
      error2=0.D0
      CALL INVARIANTS(inv,s)
      error1=svar(1)-svar(2)
      DO WHILE ((abs(error1).GT.tol).AND.(i.LE.im))
        i=i+1
C Calculate derivatives of stress invariants
        CALL DINVARIANTS(dinv,s)
C Calculate derivatives of yield and plastic potential function with respect to
C stresses
        CALL DYIELD(dfds,inv,dinv,mat,svar(7),svar(1),np)
        CALL DPOT(dgds,inv,dinv,mat,np)
C Calculate derivative of hardening or softening law
        IF (IDNINT(svar(4)).GT.0) THEN
          dfdw=DSOFT(svar(5),svar(6),svar(3),mat(13))
        ELSE
          dfdw=DHARD(svar(3),mat,np)
        END IF
C Calculate vector product (s dgds)
        sdg=0.D0
        DO j=1,4
          sdg=sdg+s(j)*dgds(j)
        END DO
C Calculate tensor product (C dgds)
        DO j=1,4
          Cdg(j)=0.D0
          DO r=1,4
            Cdg(j)=Cdg(j)+C(j,r)*dgds(r)
          END DO
        END DO
C Calculate vector product (dfds Cdg)
        dfCdg=0.D0
        DO j=1,4
          dfCdg=dfCdg+dfds(j)*Cdg(j)
        END DO
C Calculate correction factor
        beta1=error1/(dfCdg+dfdw*sdg)
C Reduction of correction factor due to lack of convergence
        IF (((abs(error2)-abs(error1)).LT.0).AND.((abs(error2)).GT.0)
```



```
     1  .AND.(cutback.EQ.0)) THEN
          beta1=(beta1+beta2)/2
          cutback=1
        ELSEIF (cutback.EQ.1) THEN
          beta1=(beta1+beta2)/2.D0
        END IF
C Update stresses and plastic work
        dwp=0.D0
        DO j=1,4
          ds(j)=-beta1*Cdg(j)
          s(j)=s(j)+ds(j)
          depsp(j)=beta1*dgds(j)
          dwp=dwp+s(j)*depsp(j)
        END DO
        svar(3)=svar(3)+dwp
C Perform compatibility check
        CALL INVARIANTS(inv,s)
        svar(7)=FAILURE(inv,mat,np)
        svar(1)=YIELD(inv,mat,svar(7),np)
        IF (IDNINT(svar(4)).GT.0) THEN
          svar(2)=SOFT(svar(5),svar(6),svar(3),mat(13))
        ELSE
          svar(2)=HARD(svar(3),mat,np)
        END IF
        beta2=beta1
        error2=error1
        error1=svar(1)-svar(2)
      END DO
      END


C SUBROUTINE MEULER(s,svar,isvar,deps,mat,tol,r,j,nsv,np)
C Update stresses by the modified Euler scheme with active error control proposed
C by Sloan (1987). The scheme is furthermore described in Jakobsen (2001)
C
C INPUT
C   s:      Initial stress state
C   isvar:  Initial values of state dependent variables
C   deps:   Strain increment
C   mat:    Material properties
C   tol:    Tolerance
C   nsv:    Number of state dependent variables
C   np:     Number of user defined properties
C
C OUTPUT
C   s:      Final stress state
C   svar:   Updated values of state dependent variables
C   r:      Number of substeps used for integration
C   j:      Number of corrections to the substep size
C
C REFERENCES
C   Sloan, S.W. (1987) "Substepping schemes for the numerical integration of
C     elastoplastic stress-strain relations", International Journal for Numerical
```

```
C       methods in Engineering, 24, pp.  893-911.
C     Jakobsen, K.P (2001) "Application of the Single Hardening model in ABAQUS"
C       AGEP0000
C

      SUBROUTINE MEULER(s,svar,isvar,deps,mat,tol,r,j,nsv,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,np,r,j
      REAL*8 s(4),s0(4),svar(nsv),isvar(nsv),deps(4),mat(np),tol
C Define secondary variables
      INTEGER p,t
      REAL*8 q,dds(4,2),ddwp(2),sumddeps(4),ddeps(4),error,
     1inv(4),dinv(4,3),df(4),dg(4),g,dfdw,mindeps,resdeps(4),
     2dl,s1(4),ddepsp(4),dwp,ddds(4),psi1,rho,d,
     3Cp(4,4),Ce(4,4)
C Function declarations
      REAL*8 NORM,POT,YIELD,FAILURE,SOFT,HARD,DHARD,DSOFT
C Initialise local variables
      r=0
      j=0
      q=1.0D0
      DO p=1,4
        DO t=1,2
          dds(p,t)=0.D0
        END DO
        s0(p)=s(p)
        sumddeps(p)=0.D0
        ddeps(p)=deps(p)
        resdeps(p)=deps(p)
      END DO
      error=1
      DO WHILE (NORM(resdeps,4,1.0D0).GT.0)
        r=r+1
C Determined size of next strain increment
        mindeps=NORM(ddeps,4,q)
        IF (NORM(resdeps,4,1.0D0).LT.mindeps) THEN
          DO p=1,4
            ddeps(p)=resdeps(p)
          END DO
        ELSE
          DO p=1,4
            ddeps(p)=q*ddeps(p)
          END DO
        END IF
        DO WHILE (error.GT.tol)
          DO p=1,nsv
            svar(p)=isvar(p)
          END DO
C Calculate first estimate of stress increment
C
C First calculate derivatives of yield and plastic potential function
C and value of potential function
```



```
          CALL INVARIANTS(inv,s0)
          CALL DINVARIANTS(dinv,s0)
          CALL DYIELD(df,inv,dinv,mat,svar(7),svar(1),np)
          CALL DPOT(dg,inv,dinv,mat,np)
          g=POT(inv,mat,np)
C Calculate derivative of hardening or softening law
          IF (IDNINT(svar(4)).EQ.0) THEN
            dfdw=DHARD(svar(3),mat,np)
          ELSE
            dfdw=DSOFT(svar(5),svar(6),svar(3),mat(13))
          END IF
          CALL ELASTIC(Ce,inv,mat,np)
          CALL PLASTIC(Cp,dl,Ce,df,dg,g,dfdw,mat(8),ddeps)
C Calculate first estimate of stress increment, plastic strain and plastic work
          ddwp(1)=0.0D0
          DO p=1,4
            dds(p,1)=0.D0
            DO t=1,4
              dds(p,1)=dds(p,1)+(Ce(p,t)-Cp(p,t))*ddeps(t)
            END DO
            ddepsp(p)=dl*dg(p)
            ddwp(1)=ddwp(1)+s0(p)*ddepsp(p)
            s1(p)=s0(p)+dds(p,1)
          END DO
C Update state variables due to first estimate of stress increment
          CALL INVARIANTS(inv,s1)
          svar(7)=FAILURE(inv,mat,np)
          svar(1)=YIELD(inv,mat,svar(7),np)
C Calculate second estimate of stress increment
C First calculate derivatives of yield and plastic potential function
          CALL DINVARIANTS(dinv,s1)
          CALL DYIELD(df,inv,dinv,mat,svar(7),svar(1),np)
          CALL DPOT(dg,inv,dinv,mat,np)
          g=POT(inv,mat,np)
C Calculate derivative of hardening or softening law
          IF (IDNINT(svar(4)).EQ.0) THEN
            dfdw=DHARD((svar(3)+ddwp(1)),mat,np)
          ELSE
            dfdw=DSOFT(svar(5),svar(6),(svar(3)+ddwp(1)),mat(13))
          END IF
          CALL ELASTIC(Ce,inv,mat,np)
          CALL PLASTIC(Cp,dl,Ce,df,dg,g,dfdw,mat(8),ddeps)
          ddwp(2)=0.0D0
          DO p=1,4
            dds(p,2)=0.D0
            DO t=1,4
              dds(p,2)=dds(p,2)+(Ce(p,t)-Cp(p,t))*ddeps(t)
            END DO
            ddepsp(p)=dl*dg(p)
            ddwp(2)=ddwp(2)+s1(p)*ddepsp(p)
            s(p)=s0(p)+0.5*(dds(p,1)+dds(p,2))
          END DO
          dwp=0.5*(ddwp(1)+ddwp(2))
```

```
          svar(3)=svar(3)+dwp
          DO p=1,4
            ddds(p)=dds(p,2)-dds(p,1)
          END DO
          error=NORM(ddds,4,0.5D0)/NORM(s,4,1.0D0)
          IF (error.GT.tol) THEN
            q=DMAX1((0.9D0*DSQRT(tol/error)),0.01D0)
            DO p=1,4
              ddeps(p)=ddeps(p)*q
            END DO
            j=j+1
          ELSE
C Perform failure check and update state variables due to updated stresses
            CALL INVARIANTS(inv,s)
            svar(7)=FAILURE(inv,mat,np)
            svar(1)=YIELD(inv,mat,svar(7),np)
            IF ((svar(7).EQ.1.D0).AND.(IDNINT(svar(4)).EQ.0)) THEN
              svar(2)=HARD(svar(3),mat,np)
              svar(4)=1
              psi1=0.00155D0*mat(2)**(-1.27D0)
              rho=mat(10)/mat(11)
              d=mat(9)/(27.D0*psi1+3.D0)**rho
              svar(6)=mat(20)/rho*(1.D0/d)**(1.D0/rho)*(svar(3)
     1        /mat(13))**(1.D0/rho-1.D0)*1.D0/svar(2)
              svar(5)=svar(2)*DEXP(svar(6)*svar(3)/mat(13))
            END IF
          END IF
        END DO
        q=DMIN1(0.9D0*DSQRT(tol/error),2.D0)
        error=1
        DO p=1,4
          sumddeps(p)=sumddeps(p)+ddeps(p)
          s0(p)=s(p)
          resdeps(p)=deps(p)-sumddeps(p)
        END DO
      END DO
      IF (IDNINT(svar(4)).EQ.0) THEN
        svar(2)=HARD(svar(3),mat,np)
      ELSE
        svar(2)=SOFT(svar(5),svar(6),svar(3),mat(13))
      END IF
      END


C SUBROUTINE RKDP(s,svar,isvar,deps,mat,tol,r,j,nsv,np)
C Update stresses by the Runge-Kutta-Dormand-Prince integration scheme (Dormand &
C Prince 1980, Sloan & Booker 1992). The scheme is furthermore described in
C Jakobsen (2001)
C
C INPUT
C  s:     Initial stress state
C  isvar: Initial values of state dependent varibles
C  deps:  Strain increment
```



```
C  mat:   Material properties
C  tol:   Tolerance
C  nsv:   Number of state dependent variables
C  np:    Number of user defined properties
C
C OUTPUT
C  s:     Final stress state
C  svar:  Updated values of state dependent variables
C  r:     Number of substeps used for integration
C  j:     Number of corrections to the substep size
C
C REFERENCES
C  Dormand, J.R. & P.J. Prince (1980) "A family of embedded Runge-Kutta formulae",
C     Journal of Computer Applied Mathematics, 6, pp. 19-26.
C  Sloan, S.W. & J.R. Booker (1992) "Integration of Tresca amd Mohr-Coulomb
C     constitutive relations in plane strain elastoplasticity", International
C     Journal for Numerical Methods in Engineering, 33, pp. 163-196.
C  Jakobsen, K.P (2001) "Application of the Single Hardening model in ABAQUS"
C     AGEP0000
C


      SUBROUTINE RKDP(s,svar,isvar,deps,mat,tol,r,j,nsv,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,np,r,j
      REAL*8 s(4),s0(4),svar(nsv),isvar(nsv),deps(4),mat(np),tol
C Define secondary variables
      INTEGER i,p,t
      REAL*8 ddeps(4),ddepsp(4),sumddeps(4),resdeps(4),minddeps,q,
     1s1(4),s2(4),ds(4,6),dds(4),swei(4),inv(4),dinv(4,3),error,
     2ddwp(6),dwp,wpwei,psi1,rho,d,dss(4),
     3df(4),dg(4),g,dfdw,Cp(4,4),Ce(4,4),dl,coef(6),coef1(6),coef2(6)
C Function declarations
      REAL*8 NORM,POT,YIELD,FAILURE,SOFT,HARD,DHARD,DSOFT
C Initialise local variables
      r=0
      j=0
      q=1.0D0
      DO p=1,4
        DO t=1,6
          ds(p,t)=0.D0
        END DO
        s0(p)=s(p)
        sumddeps(p)=0.D0
        ddeps(p)=deps(p)
        resdeps(p)=deps(p)
      END DO
      error=1
      DO WHILE (NORM(resdeps,4,1.0D0).GT.0)
        r=r+1
C Determined size of next strain increment
        minddeps=NORM(ddeps,4,q)
        IF (NORM(resdeps,4,1.0D0).LT.minddeps) THEN
```

```
          DO p=1,4
            ddeps(p)=resdeps(p)
          END DO
        ELSE
          DO p=1,4
            ddeps(p)=q*ddeps(p)
          END DO
        END IF
        DO WHILE (error.GT.tol)
          DO p=1,nsv
            svar(p)=isvar(p)
          END DO
C Calculate estimates of stress increments 1-6
          DO i=1,6
C First establish coefficient vectors depending on the stress increment
            IF (i.EQ.1) THEN
              DO p=1,6
                coef(p)=0.D0
              END DO
            ELSEIF (i.EQ.2) THEN
              coef(1)=1.D0/5.D0
              coef(2)=0.D0
              coef(3)=0.D0
              coef(4)=0.D0
              coef(5)=0.D0
              coef(6)=0.D0
            ELSEIF (i.EQ.3) THEN
              coef(1)=3.D0/40.D0
              coef(2)=9.D0/40.D0
              coef(3)=0.D0
              coef(4)=0.D0
              coef(5)=0.D0
              coef(6)=0.D0
            ELSEIF (i.EQ.4) THEN
              coef(1)=3.D0/10.D0
              coef(2)=-9.D0/10.D0
              coef(3)=6.D0/5.D0
              coef(4)=0.D0
              coef(5)=0.D0
              coef(6)=0.D0
            ELSEIF (i.EQ.5) THEN
              coef(1)=226.D0/729.D0
              coef(2)=-25.D0/27.D0
              coef(3)=880.D0/729.D0
              coef(4)=55.D0/729.D0
              coef(5)=0.D0
              coef(6)=0.D0
            ELSEIF (i.EQ.6) THEN
              coef(1)=-181.D0/270.D0
              coef(2)=5.D0/2.D0
              coef(3)=-266.D0/297.D0
              coef(4)=-91.D0/27.D0
              coef(5)=189.D0/55.D0
```



```
              coef(6)=0.D0
            END IF
C Calculate stress increments and temporary weighted update of stresses and plastic
C work
            DO p=1,4
              swei(p)=s0(p)
              DO t=1,6
                swei(p)=swei(p)+ds(p,t)*coef(t)
              END DO
            END DO
            wpwei=svar(3)
            DO p=1,6
              wpwei=wpwei+ddwp(p)*coef(p)
            END DO
C Calculate stress invariants and derivatives of stress invariants
            CALL INVARIANTS(inv,swei)
            CALL DINVARIANTS(dinv,swei)
C Calculate values of yield and plastic potential functions
            svar(7)=FAILURE(inv,mat,np)
            svar(1)=YIELD(inv,mat,svar(7),np)
            g=POT(inv,mat,np)
C Calculate derivatives of yield, plastic potential and hardening functions
            CALL DYIELD(df,inv,dinv,mat,svar(7),svar(1),np)
            CALL DPOT(dg,inv,dinv,mat,np)
            IF (IDNINT(svar(4)).EQ.0) THEN
              dfdw=DHARD(wpwei,mat,np)
            ELSE
              dfdw=DSOFT(svar(5),svar(6),wpwei,mat(13))
            END IF
            CALL ELASTIC(Ce,inv,mat,np)
            CALL PLASTIC(Cp,dl,Ce,df,dg,g,dfdw,mat(8),ddeps)
            ddwp(i)=0.D0
            DO p=1,4
              ds(p,i)=0.D0
              DO t=1,4
                ds(p,i)=ds(p,i)+(Ce(p,t)-Cp(p,t))*ddeps(t)
              END DO
              ddepsp(p)=dl*dg(p)
              ddwp(i)=ddwp(i)+swei(p)*ddepsp(p)
            END DO
          END DO
C Weights for first and second estimate of stresses
          coef1(1)=31.D0/540.D0
          coef1(2)=0.D0
          coef1(3)=190.D0/297.D0
          coef1(4)=-145.D0/108.D0
          coef1(5)=351.D0/220.D0
          coef1(6)=1.D0/20.D0
          coef2(1)=19.D0/216.D0
          coef2(2)=0.D0
          coef2(3)=1000.D0/2079.D0
          coef2(4)=-125.D0/216.D0
          coef2(5)=81.D0/88.D0
```

```
          coef2(6)=5.D0/56.D0
C First and second estimate of stresses and plastic work
          dwp=0.D0
          DO p=1,4
            s1(p)=s0(p)
            s2(p)=s0(p)
            DO t=1,6
              s1(p)=s1(p)+ds(p,t)*coef1(t)
              s2(p)=s2(p)+ds(p,t)*coef2(t)
            END DO
            dds(p)=s2(p)-s1(p)
          END DO
          DO p=1,6
            dwp=dwp+ddwp(p)*coef2(p)
          END DO
          svar(3)=svar(3)+dwp
          error=NORM(dds,4,1.D0)/NORM(s2,4,1.D0)
          IF (error.GT.tol) THEN
            q=DMAX1((0.9D0*(tol/error)**(1.D0/5.D0)),0.01D0)
            DO p=1,4
              ddeps(p)=ddeps(p)*q
            END DO
            j=j+1
          ELSE
C Perform failure check and update state variables due to updated stresses
C and plastic work.
            CALL INVARIANTS(inv,s2)
            svar(7)=FAILURE(inv,mat,np)
            svar(1)=YIELD(inv,mat,svar(7),np)
            IF ((svar(7).EQ.1.D0).AND.(IDNINT(svar(4)).EQ.0)) THEN
              svar(2)=HARD(svar(3),mat,np)
              svar(4)=1
              psi1=0.00155D0*mat(2)**(-1.27D0)
              rho=mat(10)/mat(11)
              d=mat(9)/(27.D0*psi1+3.D0)**rho
              svar(6)=mat(20)/rho*(1.D0/d)**(1.D0/rho)*(svar(3)
     1        /mat(13))**(1.D0/rho-1.D0)*1.D0/svar(2)
              svar(5)=svar(2)*DEXP(svar(6)*svar(3)/mat(13))
            END IF
          END IF
        END DO
        q=DMIN1(0.9D0*(tol/error)**(1.D0/5.D0),2.D0)
        error=1
        CALL INVARIANTS(inv,s0)
        CALL DINVARIANTS(dinv,s0)
        CALL DPOT(dg,inv,dinv,mat,np)
        DO p=1,4
          dss(p)=s2(p)-s0(p)
          sumddeps(p)=sumddeps(p)+ddeps(p)
          s0(p)=s2(p)
          resdeps(p)=deps(p)-sumddeps(p)
        END DO
      END DO
```



```
      IF (IDNINT(svar(4)).EQ.0) THEN
        svar(2)=HARD(svar(3),mat,np)
      ELSE
        svar(2)=SOFT(svar(5),svar(6),svar(3),mat(13))
      END IF
      DO p=1,4
        s(p)=s2(p)
      END DO
      END


C SUBROUTINE FEULER(s,svar,deps,mat,m,nsv,np)
C Update stresses by the forward Euler scheme (Sloan 1987, Chen & Mizuno 1990,
C Jakobsen 2001)
C
C INPUT
C   s:      Initial stress state
C   svar:   Initial values of state dependent varibles
C   dl:     Plastic multiplier
C   deps:   Strain increment
C   mat:    Material properties
C   m:      Number of subincrements
C   nsv:    Number of state dependent variables
C   np:     Number of user defined properties
C
C OUTPUT
C   s:      Final stress state
C   svar:   Updated values of state dependent variables
C
C REFERENCES
C   Sloan, S.W. (1987) "Substepping schemes for the numerical integration of
C      elastoplastic stress-strain relations", International Journal for Numerical
C      methods in Engineering, 24, pp.  893-911.
C   Chen, W.F. & E. Mizuno (1990) "Non-linear Analysis in Soil Mechanics",
C      Elsevier, New York.
C   Jakobsen, K.P (2001) "Application of the Single Hardening model in ABAQUS"
C      AGEP0000
C


      SUBROUTINE FEULER(s,svar,deps,mat,m,nsv,np)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,np,m
      REAL*8 s(4),svar(nsv),mat(np),deps(4),dl
C Function declarations
      REAL*8 POT,YIELD,FAILURE,SOFT,HARD,DHARD,DSOFT
C Define secondary variables
      INTEGER i,p,t
      REAL*8 ddeps(4),ddepsp(4),
     1s0(4),ds(4),inv(4),dinv(4,3),
     2df(4),dg(4),g,dfdw,Ce(4,4),Cp(4,4),
     3ddwp,psi1,rho,d
```

```
C Calculate size of subincrements
      DO p=1,4
        ddeps(p)=deps(p)/DFLOAT(m)
      END DO
      CALL INVARIANTS(inv,s)
      CALL ELASTIC(Ce,inv,mat,np)
C Perform m subincrements
      DO i=1,m
C Define initial stress state for current substep
        DO p=1,4
         s0(p)=s(p)
        END DO
C Calculate stress invariants and derivative of stress invariants
        CALL INVARIANTS(inv,s0)
        CALL DINVARIANTS(dinv,s0)
C Calculate derivatives of yield and palstic potential functions
        CALL DYIELD(df,inv,dinv,mat,svar(7),svar(1),np)
        CALL DPOT(dg,inv,dinv,mat,np)
C Calculate value of potential function
        g=POT(inv,mat,np)
C Calculate derivative of hardening or softening function
        IF (IDNINT(svar(4)).EQ.0) THEN
          dfdw=DHARD(svar(3),mat,np)
        ELSE
          dfdw=DSOFT(svar(5),svar(6),svar(3),mat(13))
        END IF
C CALL ELASTIC(Ce,inv,mat,np)
        CALL PLASTIC(Cp,dl,Ce,df,dg,g,dfdw,mat(8),ddeps)
        ddwp=0.D0
        DO p=1,4
          ds(p)=0.D0
          DO t=1,4
            ds(p)=ds(p)+(Ce(p,t)-Cp(p,t))*ddeps(t)
          END DO
          s(p)=s0(p)+ds(p)
          ddepsp(p)=dl*dg(p)
          ddwp=ddwp+s0(p)*ddepsp(p)
        END DO
        svar(3)=svar(3)+ddwp
C Perform failure check and update state variables
        CALL INVARIANTS(inv,s)
        svar(7)=FAILURE(inv,mat,np)
        svar(1)=YIELD(inv,mat,svar(7),np)
        IF ((svar(7).EQ.1.D0).AND.(IDNINT(svar(4)).EQ.0)) THEN
          svar(2)=HARD(svar(3),mat,np)
          svar(4)=1.D0
          psi1=0.00155D0*mat(2)**(-1.27D0)
          rho=mat(10)/mat(11)
          d=mat(9)/(27.D0*psi1+3.D0)**rho
          svar(6)=mat(20)/rho*(1.D0/d)**(1.D0/rho)*(svar(3)/mat(13))
     1    **(1.D0/rho-1.D0)*1.D0/svar(2)
          svar(5)=svar(2)*DEXP(svar(6)*svar(3)/mat(13))
        END IF
```



```
      END DO
      IF (IDNINT(svar(4)).EQ.0) THEN
        svar(2)=HARD(svar(3),mat,np)
      ELSE
        svar(2)=SOFT(svar(5),svar(6),svar(3),mat(13))
      END IF
      END


C FUNCTION NORM(vec,n,q)
C Calculate the norm of a vector
C
C INPUT
C   vec:    Vector
C   n:      Number of elements in vector
C   q:      Scalar multiplier
C
C OUTPUT
C   NORM:   Norm of vector
C

      FUNCTION NORM(vec,n,q)
C Define primary variables
      IMPLICIT NONE
      INTEGER n
      REAL*8 vec(n),q,NORM
C Define secondary variables
      INTEGER i
C Calculate length
      NORM=0.0D0
      DO i=1,n
        NORM=NORM+q**2.D0*vec(i)**2.D0
      END DO
      END


C SUBROUTINE DUMP(svar,nsv,problem,s,el,gp,n,i)
C Termination of ABAQUS in case of numerical problems
C
C INPUT
C   svar:   State dependent variables
C   nsv:    Number of state dependent variables
C   problem:Problem identifier
C              (1) Problem encountered during intersection of yield surface
C              (2) Problem encountered during correction for yield surface drift
C   s:      Current stress state
C   el:     Element number
C   gp:     Gauss point
C   n:      Current load step
C   i:      Increment in load step
C
```

```
      SUBROUTINE DUMP(svar,nsv,problem,s,el,gp,n,i)
C Define primary variables
      IMPLICIT NONE
      INTEGER nsv,problem,el,gp,n,i
      REAL*8 svar(nsv),s(4)
C Define secondary variables
      INTEGER j
      OPEN(15,file='shm2d.dmp',status='new')
      WRITE(*,*)'******************** ABAQUS RUN IS TERMINATED ',
     1'********************'
      IF (problem.EQ.1) THEN
        WRITE(*,*)'Problem encountered during intersection of yield ',
     1   'surface'
      ELSE
        WRITE(*,*)'Problems encountered during correction for yield ',
     1   'surface drift'
      END IF
      WRITE(15,10) n
   10 FORMAT(' Load step:  ',I2)
      WRITE(15,20) i
   20 FORMAT(' Increment in load step:  ',I4)
      WRITE(15,30) el
   30 FORMAT(' Element number:  ',I5)
      WRITE(15,40) gp
   40 FORMAT(' Gauss point:  ',I1)
      WRITE(15,*) ' '
      WRITE(15,*)'Current state of stress'
      DO j=1,4
        WRITE(15,50) j,s(j)
      END DO
   50 FORMAT('  Stress component no.  ',I1,':',3D24.16)
      WRITE(15,*) ' '
      WRITE(15,*)'State dependent variables at current state of stress'
      DO j=1,nsv
        WRITE(15,60) j,svar(j)
      END DO
   60 FORMAT('  State dependent variable no.  ',I1,':',3D24.16)
      CLOSE(15)
      CALL XIT
      END
```

ISSN 1398-6465 R2002-1