SVSOLID

2D/3D FINITE ELEMENT
STRESS DEFORMATION MODELING

Theory Manual

Written by:
The Bentley Systems Team

Last Updated: Tuesday, June 23, 2020

Bentley Systems Incorporated

1 INTRODUCTION

Stress-deformation problems in geotechnical and geo-environmental engineering involve the solution of a partial differential equation referred to as a PDE. The PDE must be solved for all “finite elements” which when combined form a “continuum” (or the geometry of the problem). The theory of stress and deformation expressed in mathematical form embraces the physical behavior of the material (e.g., soil or rock) and the conservative laws of physics (i.e., conservation of energy). The physical behavior or many materials, (particularly unsaturated soils), is nonlinear and as a consequence, the PDE becomes nonlinear in character. It is well-known that the solution of nonlinear PDEs can present a challenge to the numerical modeler.

The purpose of the theory manual is to provide the user with details regarding the theoretical formulation of the PDE as well as the numerical method used in the solution. The intent of the theory manual is not to provide an exhaustive summary of all theories associated with stress and deformation. Rather, the intent is to clearly describe details of the theory used in the SVSOLID software.

SVSOLID is a numerical analysis package capable of solving stress-strain models using the finite element method. SVSOLID embraces two-dimensional (2D) plane strain, and three-dimensional (3D) stress analysis models. A user-friendly and efficient interface allows the quick input of geometry, boundary conditions, material properties, sequence of loads, etc. The solver engine generates and automatically refines the finite element mesh, saving a great deal of modeling time and allowing better control of the solution accuracy. Three-dimensional models that were considered extremely challenging to model in the past can now be created in a short time, thanks to the geometry input and automatic mesh generation system used by SVSOLID.

A wide range of material types can be simulated using SVSOLID. Special focus is given to geotechnical models through the implementation of stress-strain laws typically used to represent material behavior. SVSOLID offers the following stress-strain models: Generalized Hooke’s Law, Anisotropic Hooke’s Law, Hyperbolic model, and other types of non-linear relationships. Several types of boundary conditions can be used, including displacements, concentrated loads, and spread loads. Construction stages can also be modeled.

SVSOLID uses the SOILVISION front-end interface for model setup, the SVCORE finite-element engine for the numerical analysis, and the SOILVISION back-end interface for results visualization.

SVSOLID is one module within the SOILVISION software and other modules (SVDESIGNER, SVSOILS, etc.) within SOILVISION can be used to develop the input for SVSOLID or can be used in combination with SOILVISION for various types of analysis (for example, with SVSLOPE). Geometry can be transferred between the modules within SOILVISION. The theory presented in this manual is for an uncoupled stress-strain model. The theory for the solution of coupled consolidation using SVSOLID/SVFLUX is not presented here. Pore-water pressure results from an SVFLUX groundwater seepage analysis can be used as input to SVSOLID in an uncoupled fashion.

This Technical Reference Manual provides a concise review of the theory and formulations on which SVSOLID is based. For details regarding the software operation and modeling guidelines consult the USER MANUAL and the SVSOLID TUTORIAL MANUAL, included with the software.

This documentation is organized in the following chapters:

- Chapter 1 presents an overview of SVSOLID
- Chapter 2 explains the basic principles and concepts involved in the formulation of stress-strain models
- Chapter 3 presents the stress-strain relationships available and provides information on the selection of appropriate material properties
- Chapter 4 presents the implementation of the software solutions
- Chapter 5 explains the use of initial conditions when performing staged analyses
- Chapter 6 presents a useful list of references and reading materials.

1.1 ANALYSIS TYPES

SVSOLID can handle the following types of analyses:

Stress variable: (a) Total stress;

(Note on Water Pressure: SVSOLID solver calculations are based on Total Stresses. Specifying Head or Pore-Water Pressure on the Initial Conditions dialog will allow the calculation of Effective Stresses only as a secondary calculation for output. The water will not influence the solution results.)

Stress/strain conditions: (a) 2D plane strain;
(b) General 3D stress conditions.

1.2 CONSTITUTIVE MODELS

Several stress-strain laws have been implemented in SVSOLID. The constitutive models presently available in SVSOLID are as follows:

1. Linear Elastic (Generalized Hooke's law),
2. Hyperbolic (Duncan-Chang model), and,
3. Elasto-plastic (Mohr-Coulomb).

1.3 NONLINEAR TECHNIQUES

The method of solution of nonlinear systems used by the SVSOLID engine is a Newton-type iteration procedure. This "descent" method tries to follow the gradient of the energy functional until a condition of minimum energy is achieved (i.e., the gradient of the functional goes to zero). If the functional is nearly quadratic, then the method converges quadratically (i.e., the relative error is squared on each iteration). This strategy implemented can determine a solution without user intervention is most cases.

1.4 BOUNDARY CONDITIONS

Several types of boundary conditions can be used, including displacements, concentrated loads, line loads, and distributed loads.

2 BASIC PRINCIPLES AND EQUATIONS

SVSOLID is based on the theory of elasticity, applied to 2D and 3D models (i.e., plane-strain and general 3D). Stress analysis models can be expressed in terms of partial differential equations, (PDE's), governing the static equilibrium of the forces acting on a representative elemental volume of material. The equilibrium equations are combined with constitutive relationships, expressing changes in stresses in terms of strains. The strains are written in terms of the displacements, assuming small displacements (Lagrangian formulation). This chapter presents the basic principles and concepts involved in the formulation of stress-strain models, including the final PDE's used in stress analysis models.

2.1 STRESS/STRAIN CONDITIONS AND COORDINATE SYSTEMS

SVSOLID embraces the two most common types of geometric conditions encountered in geotechnical engineering practice:

1. Two-dimensional (2D) plane strain,
2. General three-dimensional (3D).

2.1.1 2D Plane Strain

A 2D Cartesian coordinate system (*x, y*) is used to simulate 2D plane strain geometric conditions where the *y*-direction corresponds to the elevation:

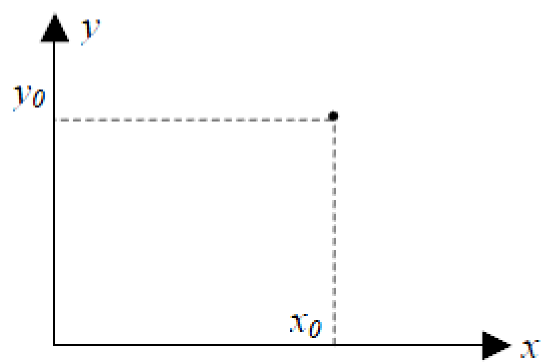

Figure 1 2D Plane Strain Coordinate Diagram

Two-dimensional plane strain problems are characterized by assuming that the strain in the z-direction is zero. Therefore, the stress and strain distributions are independent of the z-coordinates. In other words, there is a cross-section of the problem geometry that represents the entire problem. Many geotechnical problems can be approximated by a plane strain analysis. Examples of such problems include earth embankments, strip footings, highway embankments, and pipeline foundations.

2.1.2 General 3D

For general 3D geometric considerations, a 3D Cartesian coordinate system (*x, y, z*) is used, where the *z*-direction corresponds to elevation:

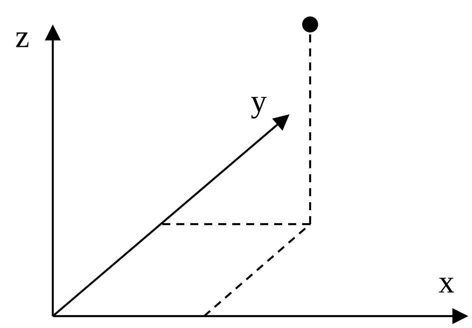

Figure 2 General 3D Coordinate Diagram

The geometry of a particular application will usually dictate the requirements of a 3D analysis. The general 3D system provides full 3D capabilities to situations where conditions cannot be represented accurately using the plane strain systems.

2.1.3 Vertical Axis

SVSOLID uses sign conventions that have differing vertical axis labels depending on the coordinate system. The sign conventions presented in this manual match the sign conventions used in the software. Note that the appropriate axis conversions are made when converting models from one coordinate system to another within the software.

Vertical Axis coordinate Label:

- 2D Plane Strain: Y
- 3D: Z

2.2 SIGN CONVENTIONS

SVSOLID uses the sign conventions that are most commonly used in geotechnical engineering. The sign conventions adopted are detailed in the following sections.

2.2.1 Load Displacement

The sign conventions for loads and displacements are defined according to the global coordinates, *x*, *y*, and *z* coordinate systems presented previously. A positive force acting in the *x*-direction will be acting towards the right, a force acting in the *y*-direction will act in the upward direction, a displacement in the *x*-direction directed to the right will be positive, and so on.

2.2.2 Stress and Pressure

SVSOLID follows the standard geotechnical engineering sign convention where compression is positive. All stress components presented in the figure below are positive. On a positive surface (i.e., the outward normal to a positive surface points in the positive coordinate direction), all stresses that act in negative coordinate directions are considered as positive. On a negative surface, all stresses acting in positive coordinate directions are positive. Therefore, compression is positive, and tension is negative. The above rule is also applied to shear stresses.

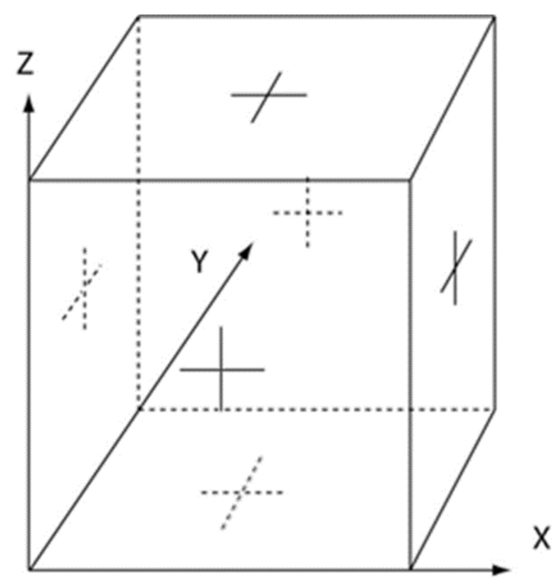

Figure 3 Illustration of the stress and pressure sign convention in SVSOLID

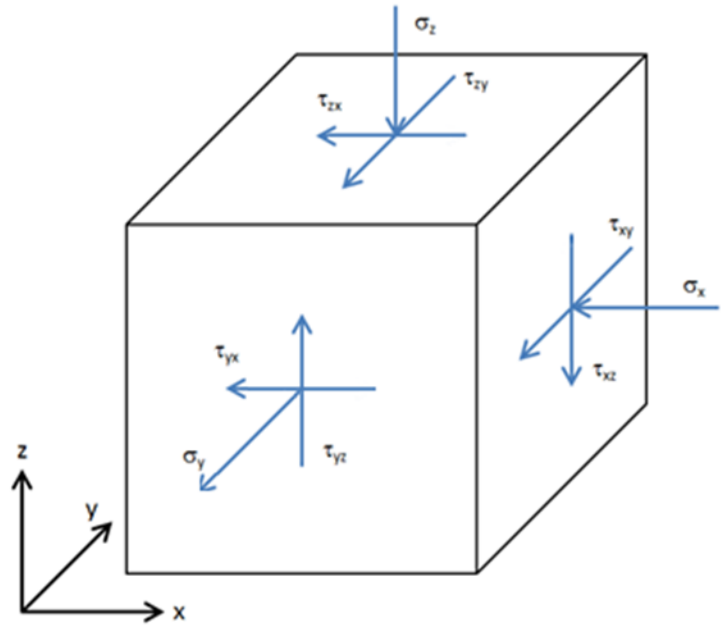

Figure 4 Illustration of components of stress tensors in SVSOLID

2.2.3 Strains

Normal strains are considered positive when a decrease in length is in the direction of the strain. Volumetric strains follow a similar convention, where a decrease in volume corresponds to a positive strain. If the shear strain is positive, the right angle between the negative extensions of the coordinate axes increases, while the angle between a positive axis and the negative extension of the outer axis decreases.

2.3 UNITS

SVSOLID analyses can be performed in Imperial or Metric units. The system of units selected dictates the units required when entering material properties, loads, and geometric dimensions.

The following table presents the model variables along with its units in both the Metric and Imperial systems:

Table 1 Model Variable Units

VARIABLE	METRIC SYSTEM	IMPERIAL SYSTEM
Distance	meters, *m*	foot, *ft*
Displacement	meters, *m*	foot, *ft*
Strain	%	%
Force	kilo Newton, *kN*	Pound force, *lbf*
Concentrated load	kilo Newton, *kN*	Pound force, *lbf*
Spread/line load	*kN/m*	*lbf/ft*
Stress	kN/m^2	lbf/ft^2 (*psf*)
Acceleration of Gravity	m/s^2	ft/s^2
Unit weight	kN/m^3	lbf/ft^3 (*pcf*)
Young modulus	*kPa*	lbf/ft^2 (*psf*)
Poisson Ratio	unit less	unit less

2.4 STRESS-STATE TENSORS

The stress-state tensor varies depending upon the stress-state variables used (i.e., total or effective stress) and the coordinate system adopted. Considering Cartesian coordinates, the stress state tensors for total and effective stresses are as follows:

i) Total stresses

$$\begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{xy} & \sigma_y & \tau_{yz} \\ \tau_{xz} & \tau_{yz} & \sigma_z \end{bmatrix}$$

Figure 5 Total stresses – cartesian coordinates

where:

σ_i = the normal stress acting on the *i*-plane, on the *i*-direction

τ_{ij} = normal stress acting on the *i*-plane, on the *j*-direction.

Figure 5 illustrates the stresses that form the stress state tensor acting upon a representative elemental volume of material.

For 2D plane strain conditions, the shear stresses acting on the *z*-direction vanish and the stress tensor reduces to the following:

i) Total stresses

$$\begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{xy} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix}$$

Figure 6 Total stresses – 2D plane strain

where:

σ_i = the normal stress acting on the *i*-plane, on the *i*-direction

τ_{ij} = shear stress acting on the *i*-plane, on the *j*-direction.

2.4.1 Effective and Total Stress, Drained and Undrained Conditions

The relationship between stresses and strains can be expressed in terms of total or effective stresses for a saturated soil. SVSOLID solver calculations are based on Total Stresses. Specifying Head or Pore-Water Pressure on the Initial Conditions dialog will allow the calculation of Effective Stresses only as a secondary calculation for output. The water will not influence the calculated solution of total stresses or deformations.

Pore-water pressures are not required when performing a total stress analysis. The stress-strain parameters must be given in terms of total stresses. A special feature available in the total stress analysis is the calculation of the buildup of pore-water pressures as a result of stresses applied on the material mass. The pore-water pressures are calculated based on the *A* and *B* pore pressure parameters. The pore-water pressures calculated in this manner correspond to an ideal undrained condition. If drained conditions are expected to occur, no excess pore-water pressures are built up by the total stresses applied to the material mass. Note that these two limiting conditions are idealized, and that the undrained pore-water pressures calculated using the *A* and *B* parameters is an approximation that does not take into consideration possible dissipation of pore pressures.

2.5 DISPLACEMENTS AND STRAINS

2.5.1 2D Plane Strain

The relationships between the components of strain and displacements can be written as follows for a Cartesian coordinate system and plane-strain conditions:

$$\varepsilon_x = \frac{\partial u}{\partial x}, \quad \varepsilon_y = \frac{\partial v}{\partial y}, \quad \gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \qquad \textbf{[1]}$$

where:

u and *v* = the displacements in the *x*- and *y*-directions, respectively.

These relationships are based on the assumption of small or infinitesimal strains. Note that the displacement, *w*, and the derivatives in the *z*-direction are zero. Therefore, ε_z, γ_{xz}, and γ_{yz} are zero.

2.5.2 General 3D

The relationships between the components of strain and displacements can be written as follows for a general three-dimensional Cartesian co-ordinate system.

$$\varepsilon_x = \frac{\partial u}{\partial x}, \quad \varepsilon_y = \frac{\partial v}{\partial y}, \quad \varepsilon_z = \frac{\partial w}{\partial z}$$
$$\gamma_{xy} = \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}, \quad \gamma_{xz} = \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}, \quad \gamma_{yz} = \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \qquad [2]$$

where:

u, *v*, and *w* are the displacements in the *x-, y-,* and *z*-directions, respectively. Once more, these relationships are based on the assumption of small strains.

2.6 EQUILIBRIUM EQUATIONS

2.6.1 2D Plane Strain

The differential equilibrium equations express the static equilibrium of forces acting upon a representative elemental volume, (REV). This REV is considered to be sufficiently small that the material properties and variables are continuous within the element. This represents the underlying assumption associated with continuum mechanics and is the approach taken in SVSOLID.

In accordance with the coordinate system adopted in SVSOLID, the direction with zero strain under plane-strain conditions is the *z*-direction. Therefore, the static equilibrium equations corresponding to plane-strain conditions can be written as follows:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + b_x = 0 \qquad [3]$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + b_y = 0 \qquad [4]$$

where:

σ_i and τ_{ij} = the total normal and shear stresses, respectively,
b_i = the body force.

2.6.2 General 3D

The static equilibrium for a general 3D condition may be written as follows:

$$\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + b_x = 0 \qquad [5]$$

$$\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + b_y = 0 \qquad [6]$$

$$\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + b_z = 0 \qquad [7]$$

where:

σ_i and τ_{ij} = the total normal and shear stresses, respectively.
b_i = the body force.

2.7 FINAL PARTIAL DIFFERNTIAL EQUATIONS

SVSOLID uses the displacement components as the primary variables describing the stress-strain field model. Therefore, in order to obtain the partial differential equations (PDE's) that govern the static equilibrium of forces throughout the continuum, the stresses present in the equilibrium equations must be replaced by the displacement components. The relationships used to re-write the PDE's are the constitutive stress-strain relations. The following equation is a representation of a generalized stress-strain relationship, which can be reduced/simplified to other specific conditions that can be simulated in SVSOLID:

$$d\boldsymbol{\varepsilon} = \mathbf{D}^{-1} d\boldsymbol{\sigma} \qquad [\,8\,]$$

where:

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} & D_{15} & D_{16} \\ D_{21} & D_{22} & D_{23} & D_{24} & D_{25} & D_{26} \\ D_{31} & D_{32} & D_{33} & D_{34} & D_{35} & D_{36} \\ D_{41} & D_{42} & D_{43} & D_{44} & D_{45} & D_{45} \\ D_{51} & D_{52} & D_{53} & D_{54} & D_{55} & D_{56} \\ D_{61} & D_{62} & D_{63} & D_{64} & D_{65} & D_{66} \end{bmatrix}$$

$$\boldsymbol{\sigma}^T = \left\{ \sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx} \right\}$$

$$\boldsymbol{\varepsilon}^T = \left\{ \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx} \right\}$$

The above generalized relationship is used in this section to present a general format for PDE’s implemented in SVSOLID. A review of the available stress-strain relationships available in SVSOLID is presented in Chapter 3. The strain variables in the stress-strain relationships are replaced by the displacement components using the strain-displacement relations presented in the previous sections.

2.7.1 2D Plane Strain

Combining the equilibrium equations for plane-strain conditions with the general stress-strain law provided above and using the relationships between the components of strain and displacements, the partial differential equations, PDE's, governing equilibrium under plane strain conditions can be written as follows:

$$\frac{\partial}{\partial x}\left[D_{11}\frac{\partial u}{\partial x} + D_{12}\frac{\partial v}{\partial y} + D_{14}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left[D_{41}\frac{\partial u}{\partial x} + D_{42}\frac{\partial v}{\partial y} + D_{44}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + b_x = 0 \qquad [\,9\,]$$

$$\frac{\partial}{\partial x}\left[D_{41}\frac{\partial u}{\partial x} + D_{42}\frac{\partial v}{\partial y} + D_{44}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left[D_{21}\frac{\partial u}{\partial x} + D_{22}\frac{\partial v}{\partial y} + D_{24}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + b_y = 0 \qquad [\,10\,]$$

All variables have been previously defined.

2.7.2 General 3D

Combining the equilibrium equations for general 3D conditions with the general stress-strain law provided above and using the relationships between the components of strain and displacements, the partial differential equations, PDE's, governing equilibrium for 3D conditions can be written as follows:

$$\frac{\partial}{\partial x}\left[D_{11}\frac{\partial u}{\partial x} + D_{12}\frac{\partial v}{\partial y} + D_{13}\frac{\partial w}{\partial z}\right] + \frac{\partial}{\partial y}\left[D_{44}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial z}\left[D_{66}\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right] + b_x = 0 \qquad [\,11\,]$$

$$\frac{\partial}{\partial x}\left[D_{44}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left[D_{21}\frac{\partial u}{\partial x} + D_{22}\frac{\partial v}{\partial y} + D_{23}\frac{\partial w}{\partial z}\right] + \frac{\partial}{\partial z}\left[D_{55}\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right)\right] + b_y = 0 \qquad [\,12\,]$$

$$\frac{\partial}{\partial x}\left[D_{66}\left(\frac{\partial w}{\partial x} + \frac{\partial v}{\partial z}\right)\right] + \frac{\partial}{\partial y}\left[D_{55}\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right)\right] + \frac{\partial}{\partial z}\left[D_{31}\frac{\partial u}{\partial x} + D_{32}\frac{\partial v}{\partial y} + D_{33}\frac{\partial w}{\partial z}\right] + b_z = 0 \qquad [\,13\,]$$

All variables have been previously defined.

3 CONSTITUTIVE RELATIONSHIPS

The User must decide on a suitable constitutive model for the problem being solved. The selection of the constitutive model will also involve the selection of appropriate material parameters.

3.1 GENERAL

Two decisions are required when deciding on the mathematical behavior for each region of material:

1. Selection of a constitutive model that is suitable for the type of behavior being modeled, and,
2. Entry of material behavior parameters applicable to the selected model.

3.2 ISOTROPIC LINEAR ELASTIC LAW

The most basic material behavior can be described by using a linear elastic model. The linear elastic model is a generalization of Hooke's law, for 3D conditions.

$$d\boldsymbol{\varepsilon} = \mathbf{D}^{-1} d\boldsymbol{\sigma} \qquad [\ 14\ ]$$

where:

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{12} & 0 & 0 & 0 \\ D_{12} & D_{11} & D_{12} & 0 & 0 & 0 \\ D_{12} & D_{12} & D_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & D_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{44} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{44} \end{bmatrix}$$

$$\boldsymbol{\sigma}^T = \left\{ \sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx} \right\}$$

$$\boldsymbol{\varepsilon}^T = \left\{ \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx} \right\}$$

$$D_{11} = E(1-\mu)/\left[(1+\mu)(1-2\mu)\right]$$

$$D_{12} = E\mu/\left[(1+\mu)(1-2\mu)\right]$$

$$D_{44} = E/\left[2(1+\mu)\right];$$

where:

E = Young's modulus
μ = Poisson ratio

Poisson ratio's ranges from values as low as 0.0 at as high as 0.499. The stress-strain model presented can be used for any of the geometric conditions previously mentioned.

3.3 HYPERBOLIC MODEL

The hyperbolic model (Duncan and Chang, 1970) is implemented in SVSOLID.

$$d\boldsymbol{\varepsilon} = \mathbf{D}^{-1} d\boldsymbol{\sigma} \qquad [\ 15\ ]$$

where:

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{12} & 0 & 0 & 0 \\ D_{12} & D_{11} & D_{12} & 0 & 0 & 0 \\ D_{12} & D_{12} & D_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & D_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{44} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{44} \end{bmatrix}$$

$$\boldsymbol{\sigma}^T = \{\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}\}$$

$$\boldsymbol{\varepsilon}^T = \{\varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}\};$$

$$D_{11} = E(1-\mu)/[(1+\mu)(1-2\mu)];$$

$$D_{12} = E\mu/[(1+\mu)(1-2\mu)];$$

$$D_{44} = E/[2(1+\mu)];$$

Young's Modulus is approximated as a tangent modulus. The tangent Young Modulus in the hyperbolic model is defined as a nonlinear function of the shear strength parameters.

$$E_t = \left[1 - \frac{R_f(1-\sin\phi)(\sigma_1-\sigma_3)}{2c\cos\phi + 2\sigma_3\sin\phi}\right]^2 KP_{atm}\left(\frac{\sigma_3}{P_{atm}}\right)^n \qquad [16]$$

where:

$R_f \quad = \dfrac{(\sigma_1-\sigma_3)_f}{(\sigma_1-\sigma_3)_{ult}}$

R_f = failure ratio, equal to the ratio of the stress difference at failure (shear resistance) and the actuating stress difference,

ϕ = friction angle,

c = cohesion,

σ_1 = major principal stress,

σ_3 = minor principal stress,

K = a modulus of elasticity of reference,

P_{atm} = atmospheric pressure, and

n = an experimental parameter that defines the ratio of increase of E with σ_3.

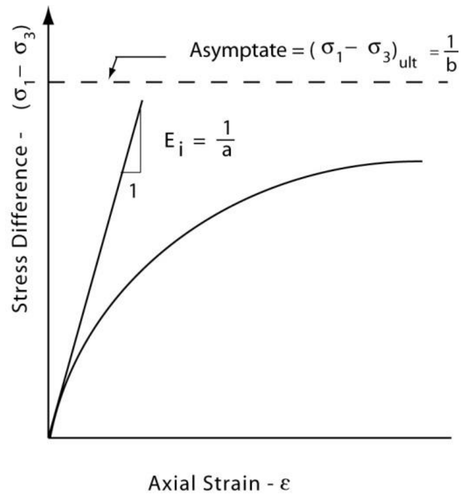

Figure 7 Characteristic stress versus strain representation for the hyperbolic model (Duncan and Chang, 1970)

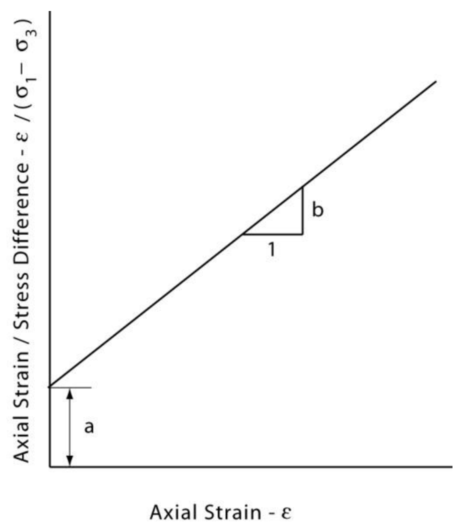

Figure 8 Transformed stress versus strain graphical representation for determining the hyperbolic model parameters (Duncan and Chang, 1970)

3.4 GENERIC ELASTO-PLASTIC RELATIONSHIP

This section presents a detailed description of how a generic elasto-plastic relationship can be established. The generic elasto-plastic relationship is established in a manner adequate for numerical modeling.

It is convenient to write the elasto-plastic relationship in the following manner:

$$\begin{cases} \text{if } F < 0 & d\boldsymbol{\sigma} = \mathbf{D}^{\mathrm{e}} d\boldsymbol{\varepsilon} \\ \text{if } F \geq 0 & d\boldsymbol{\sigma} = \mathbf{D}^{\mathrm{ep}} d\boldsymbol{\varepsilon} \end{cases} \qquad [17]$$

where:

F = the *yield function*, which divides elastic and elasto-plastic behavior.
d = an increment of stress or strain
σ = the vector of stress components: $\boldsymbol{\sigma}^T = \{\sigma_x \quad \sigma_y \quad \sigma_z \quad \tau_{xy} \quad \tau_{yz} \quad \tau_{zx}\}$
ε = the vector of strain components: $\boldsymbol{\varepsilon}^T = \{\varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx}\}$
D^e = the elastic constitutive matrix, which is a function of the Young's modulus, *E* and Poisson's ratio, μ. It is defined as the follow

$$\mathbf{D}^e = \begin{bmatrix} D_{11}^e & D_{12}^e & D_{12}^e & 0 & 0 & 0 \\ D_{12}^e & D_{11}^e & D_{12}^e & 0 & 0 & 0 \\ D_{12}^e & D_{12}^e & D_{11}^e & 0 & 0 & 0 \\ 0 & 0 & 0 & D_{44}^e & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{44}^e & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{44}^e \end{bmatrix} \qquad [\ 18\ ]$$

$$D_{11}^e = \frac{(1-\mu)E}{(1+\mu)(1-2\mu)}, \quad D_{12}^e = \frac{\mu E}{(1+\mu)(1-2\mu)}, \quad D_{44}^e = \frac{E}{2(1+\mu)} \qquad [\ 19\ ]$$

$\mathbf{D}^{ep}$ = the elasto-plastic constitutive matrix, to be defined.

The *yield function* defines the conditions that result in elastic and plastic strains. Experimental evidences show that the *yield locus* can often be expressed as a function of the stress state and a function of a hardening parameter, Γ, as follows:

$$F(\boldsymbol{\sigma},\ \Gamma) = 0 \qquad [\ 20\ ]$$

In order to obtain the constitutive matrix $\mathbf{D}^{ep}$, several fundamental principles related to elasto-plastic constitutive models must be employed. First, it is assumed that when the soil is yielding (i.e., when $F \geq 0$) strain increments, $d\varepsilon$, can be divided into an elastic portion, $d\varepsilon^e$, and into a plastic portion, $d\varepsilon^p$ (i.e., principle of additivity). The elastic strains, $d\varepsilon^e$, can be obtained using the elastic constitutive matrix, $\mathbf{D}^e$, given by Eq. [18]. Plastic strains are postulated to be proportional to the plastic potential, G:

$$d\boldsymbol{\varepsilon}^p = \Lambda \frac{\partial G}{\partial \boldsymbol{\sigma}} \qquad [\ 21\ ]$$

where:

Λ = a constant of proportionality to be determined.

The *principle of additivity* mentioned previously states that total strain increments are equal to the sum of elastic and plastic strain increments:

$$d\boldsymbol{\varepsilon} = d\boldsymbol{\varepsilon}^{\mathrm{e}} + d\boldsymbol{\varepsilon}^p = \mathbf{D}^{e^{-1}} d\boldsymbol{\sigma} + \Lambda \frac{\partial G}{\partial \boldsymbol{\sigma}} \qquad [\ 22\ ]$$

The term Λ must be obtained in order to make Eq. [22] determined. An expression for Λ can be determined by considering the incremental form of the yield function, at yield:

$$\left(\frac{\partial F}{\partial \boldsymbol{\sigma}}\right)^T d\boldsymbol{\sigma} + \frac{\partial F}{\partial \Gamma} d\Gamma = 0 \qquad [\ 23\ ]$$

or, similarly:

$$\left(\frac{\partial F}{\partial \boldsymbol{\sigma}}\right)^T d\boldsymbol{\sigma} - \Lambda A = 0 \qquad [\ 24\ ]$$

where:

$$A = -\frac{1}{\Lambda}\frac{\partial F}{\partial \Gamma} d\Gamma = -\frac{1}{\Lambda}\frac{\partial F}{\partial \Gamma}\left(\frac{\partial \Gamma}{\partial \boldsymbol{\varepsilon}^p}\right)^T d\boldsymbol{\varepsilon}^p = -\frac{\partial F}{\partial \Gamma}\left(\frac{\partial \Gamma}{\partial \boldsymbol{\varepsilon}^p}\right)^T \frac{\partial G}{\partial \boldsymbol{\sigma}} \qquad [\ 25\ ]$$

The term A is zero for models without hardening (i.e., perfectly plastic models). The next step is to multiply Eq. [22] by $(\partial F / \partial \boldsymbol{\sigma})^T \mathbf{D}^e$:

$$\left(\frac{\partial F}{\partial \boldsymbol{\sigma}}\right)^T d\boldsymbol{\sigma} = \left(\frac{\partial F}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{D}^e d\boldsymbol{\varepsilon} - \left(\frac{\partial F}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} \Lambda \qquad [\ 26\ ]$$

An expression for Λ is obtained by combining Eqs. [24] and [26] and rearranging:

$$\Lambda = \frac{1}{A - A_{cr}} \left(\frac{\partial F}{\partial \boldsymbol{\sigma}} \right)^T \mathbf{D}^e d\boldsymbol{\varepsilon} \qquad [\ 27\ ]$$

where:

$$A_{cr} = -\left(\frac{\partial F}{\partial \boldsymbol{\sigma}} \right)^T \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} \qquad [\ 28\ ]$$

Finally, the relationship between elasto-plastic strain and stress increments can be obtained by combining Eqs. [22] and [27], as follows:

$$d\boldsymbol{\varepsilon} = \mathbf{D}^{e^{-1}} d\boldsymbol{\sigma} + \left[\frac{1}{A - A_{cr}} \left(\frac{\partial F}{\partial \boldsymbol{\sigma}} \right)^T \mathbf{D}^e d\boldsymbol{\varepsilon} \right] \frac{\partial G}{\partial \boldsymbol{\sigma}} \qquad [\ 29\ ]$$

or, rearranging:

$$\mathbf{D}^{ep} = \frac{d\boldsymbol{\sigma}}{d\boldsymbol{\varepsilon}} = \left[\mathbf{D}^e - \frac{1}{A - A_{cr}} \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} \left(\frac{\partial F}{\partial \boldsymbol{\sigma}} \right)^T \mathbf{D}^e \right] \qquad [\ 30\ ]$$

Elasto-plastic model with no hardening (i.e., perfect plastic model with A = 0) is a particular case of the more general equation [30]:

$$\mathbf{D}^{ep} = \frac{d\boldsymbol{\sigma}}{d\boldsymbol{\varepsilon}} = \left[\mathbf{D}^e - \frac{\mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} \left(\frac{\partial F}{\partial \boldsymbol{\sigma}} \right)^T \mathbf{D}^e}{\left(\frac{\partial F}{\partial \boldsymbol{\sigma}} \right)^T \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}}} \right] \qquad [\ 31\ ]$$

It's important to point out that Eqs. [29] and [30] provide considerably general expressions for the elasto-plastic constitutive matrix. Equations [29] and [30] require the establishment of F and G and the derivatives of F and G with respect to the stress vector. A variety of equations for F and G are available in the literature or can be established based on experimental observation of soil behavior.

3.5 GENERIC IMPLICIT INTEGRATION SCHEME

In the SVCORE finite element solver, the following implicit integration algorithm of return mapping is adopted, and the consistent tangent matrix is evaluated in order to achieve quadratic convergence in the global Newton-Raphson iteration. For elasto-plastic problems, one will eventually need to integrate the equation as below:

$$\Delta\boldsymbol{\sigma} = \int_t^{t+\Delta t} \mathbf{D}^{ep} \dot{\boldsymbol{\varepsilon}}\ dt \qquad [\ 32\ ]$$

The implicit return mapping integration scheme is implemented for equation [32] as follows.

Based on Hooke's law, the updating stress on the yield locus can be expressed as

$$\boldsymbol{\sigma}_{t+\Delta t} = \boldsymbol{\sigma}^{tr} - \Delta\lambda \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} \qquad [\ 33\ ]$$

where

$\boldsymbol{\sigma}^{tr}$ = the stress estimate at time $t + \Delta t$.

The new stress on the yield locus can be solved using equations [29] and [33] using Newton method.

In order to obtain the consistent constitutive matrix $\mathbf{D}^{ep}$, taking the derivatives with respect to ε in equation [30]:

$$\frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}} = \frac{\partial \boldsymbol{\sigma}^{tr}}{\partial \boldsymbol{\varepsilon}} - \frac{\partial \Delta\lambda}{\partial \boldsymbol{\varepsilon}} \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} - \Delta\lambda \mathbf{D}^e \frac{\partial^2 G}{\partial \boldsymbol{\sigma}^2} \frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}} \quad [\ 34\ ]$$

Rearranging equation [34], yields

$$\left(\mathbf{I} + \Delta\lambda \mathbf{D}^e \frac{\partial^2 G}{\partial \boldsymbol{\sigma}^2} \right) \frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}} + \mathbf{D}^e \frac{\partial G}{\partial \boldsymbol{\sigma}} \frac{\partial \Delta\lambda}{\partial \boldsymbol{\varepsilon}} = \mathbf{D}^e \quad [\ 35\ ]$$

Then, taking derivatives with respect to $\boldsymbol{\varepsilon}$ in the yield function Eq. [29], gives

$$\frac{\partial F}{\partial \boldsymbol{\sigma}} \frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}} + \frac{\partial F}{\partial \Gamma} \left(\frac{\partial \Gamma}{\partial \boldsymbol{\sigma}} \frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}} + \frac{\partial \Gamma}{\partial \Delta\lambda} \frac{\partial \Delta\lambda}{\partial \boldsymbol{\varepsilon}} \right) = 0 \quad [\ 36\ ]$$

where the derivatives with respect to the stress in Eqs. [35] and [36] are evaluated at the updated stress on the yield locus.

Solving Eqs. [35] and [36], the consistent constitutive relationship $\frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}}$ can be obtained.

3.5.1 The Mohr-Coulomb Model

This model is perfectly plastic there is no hardening/softening law required. The yield function of the Mohr-Coulomb model is

$$F = R_{mc} q - p \tan\phi - c \quad [\ 37\ ]$$

where:

R_{mc} = the Mohr-Coulomb deviatoric measure
q = the Mises equivalent stress,
p = the mean stress,
ϕ = the friction angel and
c = the soil cohesion.

R_{mc} is expressed as below:

$$R_{mc} = \frac{1}{\sqrt{3}\cos\phi} \sin\left(\theta + \frac{\pi}{3}\right) + \frac{1}{3}\cos\left(\theta + \frac{\pi}{3}\right)\tan\phi \quad [\ 38\ ]$$

where:

ϕ = the friction angle of the material in the meridional stress plane
θ = the deviatoric polar angle defined as

$$\cos(3\theta) = \left(\frac{r}{q}\right)^3 \quad [\ 39\ ]$$

where:

r = the third invariant of deviatoric stress and it is

$$r = \left(\frac{9}{2} S \cdot S : S\right)^{1/3} \quad [\ 40\ ]$$

where:

S = the stress deviator.

The potential function G of Mohr-Coulomb model is defined as

$$G = \sqrt{(\varepsilon c_0 \tan\psi)^2 + (R_{mw} q)^2} - p \tan\psi \quad [\ 41\ ]$$

Where

ψ = the dilation angle measured in the p-$R_{mw}q$ plane at high confining pressure
c_0 is the initial yield cohesion stress
ε is a parameter, referred as the eccentricity, that defines the rate at which the function approaches the asymptote.

The flow potential function is continuous and smooth in the deviatoric stress plane; we adopt the deviatoric elliptic function used by Menetrey and Willam (1995)

$$R_{mw}(\theta, e) = \frac{4(1-e^2)\cos^2\theta + (2e-1)^2}{2(1-e^2)\cos\theta + (2e-1)\sqrt{4(1-e^2)\cos^2\theta + 5e^2 - 4e}} R_{mc}\left(\frac{\pi}{3}, \phi\right) \qquad [\ 42\ ]$$

where:

$R_{mc}(\pi/3, \phi) = (3 - \sin\phi)/(6 \cos\phi)$
e = a parameter that describes the "out-of-roundedness" of the deviatoric section in terms of the ratio between the shear stress along the extension meridian ($\theta = 0$) and the shear stress along the compression meridian ($\theta = \pi/3$).

By default, the out-of-roundedness parameter, e, is dependent on the friction angle f; it is calculated by matching the flow potential to the yield surface in both triaxial tension and compression in the deviatoric plane:

$$e = \frac{3 - \sin\phi}{3 + \sin\phi} \qquad [\ 43\ ]$$

4 NEWTON RAPHSON METHOD AND CONVERGENCE CRITERION

The nonlinear solver (e.g., Newton-Raphson method) associated with stress analysis is outlined in this section.

It is assumed that a system is in equilibrium state at time *t*. In the subsequent time increment *Δt*, an external load is applied {*Δ**F***}. Substitution of the incremental stress and strain relationship $\frac{\partial \boldsymbol{\sigma}}{\partial \boldsymbol{\varepsilon}}$ into equilibrium equations leads to:

$$[\mathbf{K}]\{\Delta\mathbf{u}\}=\{\Delta\mathbf{F}\} \qquad [\,44\,]$$

where

[**K**] = a stiffness matrix,
{Δ**u**} = the incremental vector of displacement,
{Δ**F**} = the incremental vector of equivalent external force.

However, the relation between stress and strain increments is generally nonlinear; the aforementioned stiffness matrix cannot be formulated exactly beforehand. Therefore, a global iterative procedure is required to satisfy both the equilibrium and the incremental stress and strain relation. The global iteration process can be written as:

$$[\mathbf{K}]\{\delta\mathbf{u}\}=\{\mathbf{b}\} \qquad [\,45\,]$$

where:

[**K**] = a stiffness matrix,
{δ**u**} = the correction vector of displacement,
{**b**} = the vector of unbalanced force.

Thus, the displacement after correction can be written as:

$$\{\Delta\mathbf{u}\}=\{\Delta\mathbf{u}\}+\{\delta\mathbf{u}\} \qquad [\,46\,]$$

After one iteration, we need to check the convergence. It can stop to run if it is converged by the following absolute energy criterion:

$$\left\| \frac{\{\delta\mathbf{u}\}^T\{\mathbf{b}\}}{\{\Delta\mathbf{u}\}^T\{\Delta\mathbf{F}\}} \right\| < Tol \qquad [\,47\,]$$

where:

Tol = the tolerance.

If the above convergence criterion is not satisfied, the solver will compute unbalance force and loop again to correct displacement until maximum iteration number is reached.

5 INITIAL CONDITIONS

The initial conditions associated with a stress analysis, available for specification in SVSOLID are outlined in this section.

5.1 INITIAL STRESSES

The options available to define initial stress conditions are outlined in this section.

5.1.1 Introduction

The import of initial *in situ* stresses is important for many geotechnical engineering analyses. Initial stress conditions must be input whenever the stress analysis is nonlinear. Initial stress conditions are required in order to obtain a reasonable representation of existing *in situ* stresses at the start of an analysis.

5.1.2 Body Loads

Initial *in situ* stresses are often calculated through the application of a body load simulating gravity. The deformations are then discarded, and the stresses are imported as the initial stresses in subsequent analysis.

5.1.3 Horizontal or Vertical

Initial stresses can be imported as computed from a previous analysis or the vertical stresses can be imported, and the horizontal stresses calculated using an earth pressure coefficient at-rest. It is up to the User to select the scenario that produces the most realistic initial stresses.

5.1.4 Linear Variation

The current version of SVSOLID does not allow for the definition of linearly varying stresses. Linearly varying stresses with depth can be generated by specifying body loads for each material region and running an initial analysis.

5.1.5 Pore-Water Pressures

Pore-water pressures can be imported from a previous stress analysis or from a SVFLUX analysis. The water pressures are imported at each node.

6 SUPPORTS

This section outlines the supports theory used in SVSOLID.

6.1 GENERAL

Support systems are modeled by using one-dimensional (1D) bar finite elements (Figure 9). A bar element is a mechanical device capable of supporting axial loading only, and the elongation or contraction of the bar element is directly proportional to the applied axial load.

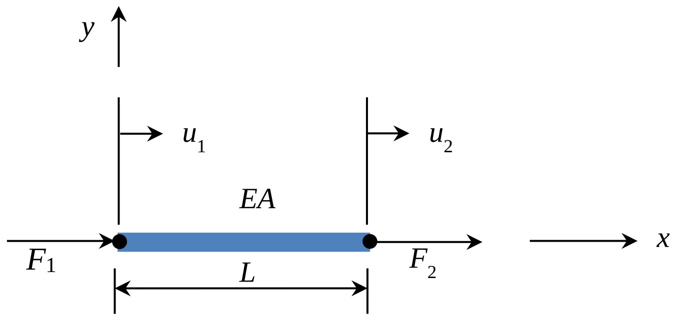

Figure 9. Line element with nodes, nodal displacements, and nodal forces.

From the elementary strength of materials that the deflection Δu of an elastic bar of length L and uniform cross-sectional area A when subjected to axial load F :

$$\Delta u = \frac{FL}{EA} \quad [48]$$

where E is the modulus of elasticity of the material and the product EA is known as axial rigidity. The constant of proportionality between deformation and load is referred to as stiffness (K) of the element.

$$K = \frac{F}{\Delta u} = \frac{EA}{L} \quad [49]$$

Assuming that both the nodal displacements are zero initially, the net deformation is given by

$$\Delta u = u_2 - u_1 \quad [50]$$

The resultant force in the element is

$$F = K\Delta u = K(u_2 - u_1) \quad [51]$$

For equilibrium,

$$F_1 + F_2 = 0 \text{ or, } F_1 = -F_2 \quad [52]$$

Then, in terms of the applied nodal forces as

$$F_1 = -K(u_2 - u_1)$$
$$F_2 = K(u_2 - u_1) \quad [53]$$

which can be expressed in the matrix form as

$$\begin{bmatrix} K & -K \\ -K & K \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \text{ or } [K_e]\{u\} = \{f\} \quad [54]$$

where:

$[K_e] = \begin{bmatrix} K & -K \\ -K & K \end{bmatrix} = K \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$ is the stiffness matrix for a one bar element,

$\{u\}$ = vector of nodal displacements, and

$\{f\}$ = vector of element nodal forces.

After assembling all the elements, the above equation becomes

$$[K]\{U\} = \{F\} \quad [55]$$

where:

$[K]$ = Global stiffness matrix,

$\{U\}$ = Array of displacements, and

$\{F\}$ = Array of external forces.

6.2 SWELLEX / SPLIT SETS

The Swellex /Split Sets bolts can be called Friction or Shear Bolts. The bolt is discretized into multiple finite elements at the intersection points with other finite element mesh, but the entire bolt behaves as a single element because each segment of the bolt has a direct effect on adjacent segments. The Swellex / Split Set bolt fails in two ways: if either the Tensile Capacity is exceeded, or the Bond Strength is exceeded.

The Swellex / Split Sets bolt considers the shear stress due to a relative movement between the bolt and surrounding soil/rock. For example, Farmer (1975) experimentally studied the stress distribution along a resin grouted rock anchor in concrete, limestone, and chalk and proposed a theoretical equation to approximate the shear stress distribution along a typical resin anchor. Figure 10a shows a uniform bolt member with linearly varying distributed the axial load (shear force) of intensity $q(x)$. Thus, the potential energy functional for the bolt can be written as:

$$\Pi_p = \frac{EA}{2} \int_0^L \left(\frac{du}{dx} \right)^2 dx - \int_0^L qu \, dx \quad [56]$$

and the governing differential equation can be written as (Cook et al., 1989):

$$EA \frac{d^2u}{dx^2} + q = 0 \quad [57]$$

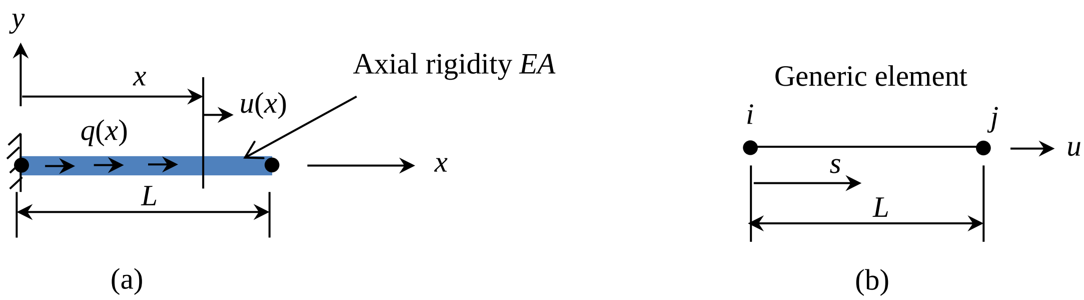

Figure 10. (a) A uniform bolt under linearly varying distributed axial load of intensity *q*, and (b) A generic element.

If the shear force is assumed to be a linear function of the relative movement between the surrounding material and the bolt then,

$$q = k(u_r - u_x) \quad \textbf{[58]}$$

where k is the shear stiffness of the bolt-grout interface measured in laboratory pull-out tests, u_r is displacement of surrounding material, and u_x is the displacement of the bolt.

The finite element weak form can be written by substituting the Equation [58] into [57] as follows:

$$\begin{aligned} \delta\Pi &= \int \left(EA \frac{d^2 u_x}{dx^2} - ku_x + ku_r \right) \delta u \, dx \\ &= \int \left\{ EA \left[\frac{d}{dx} \left(\frac{du_x}{dx} \delta u \right) - \frac{du_x}{dx} \frac{d\delta u}{dx} \right] - \left(ku_x - ku_r \right) \delta u \right\} dx \\ &= EA \, \delta u \frac{du_x}{dx} \Big|_0^L - \int \left(EA \frac{du_x}{dx} \frac{d\delta u}{dx} + ku_x \delta u \right) dx + \int \left(ku_r \delta u \right) dx \end{aligned} \quad \textbf{[59]}$$

Let's consider a generic element as shown in Figure 10b. Its axial displacement u is to be linear in the axial coordinate s (Cook et al., 1989). The displacement field must yield $u = u_i$ at one end and $u = u_j$ at the other. Then, the displacement at any point within the element can be written as:

$$\begin{aligned} u &= \frac{L-s}{L} u_i + \frac{s}{L} u_j \\ &= [N]\{d\} \end{aligned} \quad \textbf{[60]}$$

where $[N] = \begin{bmatrix} \frac{L-s}{L} & \frac{s}{L} \end{bmatrix}$ is called a shape function matrix, and $\{d\} = \{u_i \quad u_j\}^T$ is called an array of displacements.

The axial strain is $\varepsilon_x = du/dx = du/ds$. From Equation [60],

$$\varepsilon_x = [B]\{d\}, \quad \text{where} \quad [B] = \frac{d}{ds}[N] = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} \quad \textbf{[61]}$$

Matrix $[B]$ is called the *strain-displacement* matrix.

For two displacement fields, the Equation [60] can be written as:

$$u = \begin{Bmatrix} u_x \\ u_r \end{Bmatrix} = \begin{bmatrix} N_1 & N_2 & 0 & 0 \\ 0 & 0 & N_1 & N_2 \end{bmatrix} \begin{Bmatrix} u_{x1} \\ u_{x2} \\ u_{r1} \\ u_{r2} \end{Bmatrix} \quad [62]$$

Now, the stiffness matrix from the Equation [59] can be written as:

$$-\int \left(EA \frac{du_x}{dx} \frac{d\delta u}{dx} + ku_x \delta u \right) dx + \int \left(ku_r \delta u \right) dx =$$

$$-\begin{bmatrix} u_{x1} & u_{x2} & u_{r1} & u_{r2} \end{bmatrix} \begin{bmatrix} K_b & 0 \\ 0 & -K_r \end{bmatrix} \delta \begin{Bmatrix} u_{x1} \\ u_{x2} \\ u_{r1} \\ u_{r2} \end{Bmatrix} \quad [63]$$

where,

$$\left[K_b \right] = \int_0^L \left\{ EA \begin{bmatrix} \frac{dN_1}{dx}\frac{dN_1}{dx} & \frac{dN_1}{dx}\frac{dN_2}{dx} \\ \frac{dN_2}{dx}\frac{dN_1}{dx} & \frac{dN_2}{dx}\frac{dN_2}{dx} \end{bmatrix} + k \begin{bmatrix} N_1 N_1 & N_1 N_2 \\ N_2 N_1 & N_2 N_2 \end{bmatrix} \right\} dx$$

$$= \frac{EA}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} + \frac{kL}{3} \begin{bmatrix} 1 & 0.5 \\ 0.5 & 1 \end{bmatrix} \quad [64]$$

And

$$\left[K_r \right] = k \begin{bmatrix} N_1 N_1 & N_1 N_2 \\ N_2 N_1 & N_2 N_2 \end{bmatrix} = \frac{kL}{3} \begin{bmatrix} 1 & 0.5 \\ 0.5 & 1 \end{bmatrix} \quad [65]$$

Equations [64] and [65] are used to assemble the stiffness for the bolt elements.

7 SHEAR STRENGTH REDUCTION

The Shear Strength Reduction Method (SSR) uses the Finite Element Method (FEM) to determine the Factor of Safety (FOS) for a slope. In general, the method involves reducing the shear strength envelope of a material until deformations are large or convergence cannot be achieved.

7.1 SHEAR STRENGTH REDUCTION EQUATIONS

The Factor of Safety FOS of a slope is defined as the factor by which the original shear strength parameter is divided to bring the slope to the critical equilibrium state.

$$C_f = \frac{C}{FOS} \qquad tg\phi_f = \frac{tg\phi}{FOS} \qquad [66]$$

where:

C_f = the strength parameter of cohesion in the critical equilibrium state

ϕ_f = the strength parameter of friction angle in the critical equilibrium state

C = the actual strength parameter of cohesion

ϕ = the actual strength parameter of friction angle

FOS = the factor of safety in the critical equilibrium state

When the strength reduction technique is used to calculate the *FOS*, it is usually necessary to solve a series of elastic-perfectly plastic problems with the following strength parameters:

$$C_i = \frac{C}{FOS^i} \qquad tg\phi_i = \frac{tg\phi}{FOS^i} \qquad [67]$$

where:

FOS^i = the strength reduction factors (*i*=1, 2, 3, ……).

In the case where the elasto-perfectly plastic Mohr-Coulomb constitutive model is used for geomaterials, we take the FOS^i corresponding to the critical equilibrium state as the FOS.

8 REFERENCES

Cook, R. D., Malkus, D. S., & Plesha, M. E. (1989). Concepts and Applications of Finite Element Analysis (3rd ed.). John Wiley & Sons, Inc.

Duncan, J.M., Chang, C. Y., (1970). Nonlinear analysis of stress and strain in soil. ASCE J. Of the Soil Mech. and Found. Div. Vol. 96, pp. 1629-1653.

Farmer, I. W. (1975). Stress distribution along a resin grouted rock anchor. International Journal of Rock Mechanics and Mining Science & Geomechanics Abstracts, 12, 347–351.

Menetrey, P., and Willam, K. J. (1995). Triaxial failure criterion for concrete and its generalization. ACI structural Journal, 92(3), 311-318.