PLAXIS
3D FOUNDATION
Scientific Manual
version 1.5

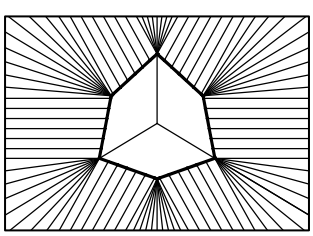

TABLE OF CONTENTS

1 INTRODUCTION

In this part of the manual some scientific background is given of the theories and numerical methods on which the PLAXIS 3D FOUNDATION program is based. The manual contains a general chapter on deformation theory and a chapter on finite element formulations and integration rules for the various types of elements used in the 3D FOUNDATION program. In the Appendix a global calculation scheme is provided for a plastic deformation analysis.

In addition to the specific information given in this part of the manual, more information on backgrounds of theory and numerical methods can be found in the literature, as a.o. referred to in Chapter 5. For detailed information on stresses, strains, constitutive modelling and the types of soil models used in the PLAXIS 3D FOUNDATION program, the reader is referred to the Material Models Manual.

2 DEFORMATION THEORY

In this chapter the basic equations for the static deformation of a soil body are formulated within the framework of continuum mechanics. A restriction is made in the sense that deformations are considered to be small. This enables a formulation with reference to the original undeformed geometry. The continuum description is discretised according to the finite element method.

2.1 BASIC EQUATIONS OF CONTINUUM DEFORMATION

The static equilibrium of a continuum can be formulated as:

$$\underline{\underline{L}}^T \underline{\sigma} + \underline{p} = \underline{0} \tag{2.1}$$

This equation relates the spatial derivatives of the six stress components, assembled in vector $\underline{\sigma}$, to the three components of the body forces, assembled in vector $\underline{p}$. $\underline{\underline{L}}^T$ is the transpose of a differential operator, defined as:

$$\underline{\underline{L}}^T = \begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial z} \\ 0 & \frac{\partial}{\partial y} & 0 & \frac{\partial}{\partial x} & \frac{\partial}{\partial z} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix} \tag{2.2}$$

In addition to the equilibrium equation, the kinematic relation can be formulated as:

$$\underline{\varepsilon} = \underline{\underline{L}}\, \underline{u} \tag{2.3}$$

This equation expresses the six strain components, assembled in vector $\underline{\varepsilon}$, as the spatial derivatives of the three displacement components, assembled in vector $\underline{u}$, using the previously defined differential operator $\underline{\underline{L}}$. The link between Eq. (2.1) and (2.3) is formed by a constitutive relation representing the material behaviour. Constitutive relations, i.e. relations between rates of stress and strain, are extensively discussed in the Material Models Manual. The general relation is repeated here for completeness:

$$\dot{\underline{\sigma}} = \underline{\underline{\underline{M}}}\, \dot{\underline{\varepsilon}} \tag{2.4}$$

The combination of Eqs. (2.1), (2.3) and (2.4) would lead to a second-order partial differential equation in the displacements $\underline{u}$.

However, instead of a direct combination, the equilibrium equation is reformulated in a weak form according to Galerkin's variation principle (see among others Zienkiewicz, 1967):

$$\int \delta \underline{u}^T \left(\underline{\underline{L}}^T \underline{\sigma} + \underline{p} \right) dV = 0 \tag{2.5}$$

In this formulation $\delta\underline{u}$ represents a kinematically admissible variation of displacements. Applying Green's theorem for partial integration to the first term in Eq. (2.5) leads to:

$$\int \delta \underline{\varepsilon}^T \underline{\sigma}\, dV = \int \delta \underline{u}^T \underline{p}\, dV + \int \delta \underline{u}^T \underline{t}\, dS \tag{2.6}$$

This introduces a boundary integral in which the boundary traction appears. The three components of the boundary traction are assembled in the vector $\underline{t}$. Eq. (2.6) is referred to as the virtual work equation.

The development of the stress state $\underline{\sigma}$ can be regarded as an incremental process:

$$\underline{\sigma}^i = \underline{\sigma}^{i\text{-}1} + \Delta\underline{\sigma} \qquad\qquad \Delta\underline{\sigma} = \int \dot{\underline{\sigma}}\, d\, t \tag{2.7}$$

In this relation $\underline{\sigma}^i$ represents the actual state of stress which is unknown and $\underline{\sigma}^{i\text{-}1}$ represents the previous state of stress which is known. The stress increment $\Delta\underline{\sigma}$ is the stress rate integrated over a small time increment.

If Eq. (2.6) is considered for the actual state *i*, the unknown stresses $\underline{\sigma}^i$ can be eliminated using Eq. (2.7):

$$\int \delta \underline{\varepsilon}^T\ \Delta\underline{\sigma}\, dV = \int \delta \underline{u}^T\, \underline{p}^i\ dV + \int \delta \underline{u}^T \underline{t}^{\,i}\, dS - \int \delta \underline{\varepsilon}^T \underline{\sigma}^{i\text{-}1}\, dV \tag{2.8}$$

It should be noted that all quantities appearing in Eqs. (2.1) to (2.8) are functions of the position in the three-dimensional space.

2.2 FINITE ELEMENT DISCRETISATION

According to the finite element method a continuum is divided into a number of (volume) elements. Each element consists of a number of nodes. Each node has a number of degrees of freedom that correspond to discrete values of the unknowns in the boundary value problem to be solved. In the present case of deformation theory the degrees of freedom correspond to the displacement components. Within an element the displacement field $\underline{u}$ is obtained from the discrete nodal values in a vector $\underline{v}$ using interpolation functions assembled in matrix $\underline{\underline{N}}$:

$$\underline{u} = \underline{\underline{N}}\,\underline{v} \tag{2.9}$$

The interpolation functions in matrix $\underline{\underline{N}}$ are often denoted as shape functions. Substitution of Eq. (2.9) in the kinematic relation (2.3) gives:

$$\underline{\varepsilon} = \underline{\underline{L}}\,\underline{\underline{N}}\,\underline{v} = \underline{\underline{B}}\,\underline{v} \tag{2.10}$$

In this relation $\underline{\underline{B}}$ is the strain interpolation matrix, which contains the spatial derivatives of the interpolation functions. Eqs. (2.9) and (2.10) can be used in variational, incremental and rate form as well.

Eq. (2.8) can now be reformulated in discretised form as:

$$\int \left(\underline{\underline{B}}\,\delta\underline{v}\right)^T \Delta\underline{\sigma}\, dV = \int \left(\underline{\underline{N}}\,\delta\underline{v}\right)^T \underline{p}^i\, dV + \int \left(\underline{\underline{N}}\,\delta\underline{v}\right)^T \underline{t}^{\,i}\, dS - \int \left(\underline{\underline{B}}\,\delta\underline{v}\right)^T \underline{\sigma}^{i-1}\, dV \tag{2.11}$$

The discrete displacements can be placed outside the integral:

$$\delta\,\underline{v}^T \int \underline{\underline{B}}^T \Delta\underline{\sigma} dV = \delta\,\underline{v}^T \int \underline{\underline{N}}^T \underline{p}^i\, dV + \delta\,\underline{v}^T \int \underline{\underline{N}}^T \underline{t}^{\,i}\, dS - \delta\,\underline{v}^T \int \underline{\underline{B}}^T \underline{\sigma}^{i-1}\, dV \tag{2.12}$$

Provided that Eq. (2.12) holds for any kinematically admissible displacement variation $\delta\underline{v}^T$, the equation can be written as:

$$\int \underline{\underline{B}}^T \Delta\underline{\sigma}\; dV = \int \underline{\underline{N}}^T \underline{p}^i\, dV + \int \underline{\underline{N}}^T \underline{t}^{\,i}\, dS - \int \underline{\underline{B}}^T \underline{\sigma}^{i-1}\, dV \tag{2.13}$$

The above equation is the elaborated equilibrium condition in discretised form. The first term on the right-hand side together with the second term represent the current external force vector and the last term represents the internal reaction vector from the previous step. A difference between the external force vector and the internal reaction vector should be balanced by a stress increment $\Delta\underline{\sigma}$.

The relation between stress increments and strain increments is usually non-linear. As a result, strain increments can generally not be calculated directly, and global iterative procedures are required to satisfy the equilibrium condition (2.13) for all material points. Global iterative procedures are described later in Section 2.4, but the attention is first focussed on the (local) integration of stresses.

2.3 IMPLICIT INTEGRATION OF DIFFERENTIAL PLASTICITY MODELS

The stress increments $\Delta\underline{\sigma}$ are obtained by integration of the stress rates according to Eq. (2.7). For differential plasticity models the stress increments can generally be written as:

$$\Delta\underline{\sigma} = \underline{\underline{D}}^e \left(\Delta\underline{\varepsilon} - \Delta\,\underline{\varepsilon}^p\right) \tag{2.14}$$

In this relation $\underline{\underline{D}}^e$ represents the elastic material matrix for the current stress increment. The strain increments $\Delta\underline{\varepsilon}$ are obtained from the displacement increments $\Delta\underline{v}$ using the strain interpolation matrix $\underline{\underline{B}}$, similar to Eq. (2.10).

For elastic material behaviour, the plastic strain increment $\Delta\underline{\varepsilon}^p$ is zero. For plastic material behaviour, the plastic strain increment can be written, according to Vermeer (1979), as:

$$\Delta\underline{\varepsilon}^p = \Delta\lambda\left[\ (1-\omega)\left(\frac{\partial g}{\partial\underline{\sigma}}\right)^{i\text{-}1} + \omega\left(\frac{\partial g}{\partial\underline{\sigma}}\right)^{i}\ \right] \tag{2.15}$$

In this equation $\Delta\lambda$ is the increment of the plastic multiplier and ω is a parameter indicating the type of time integration. For $\omega = 0$ the integration is called explicit and for $\omega = 1$ the integration is called implicit.

Vermeer (1979) has shown that the use of implicit integration ($\omega = 1$) has some major advantages, as it overcomes the requirement to update the stress to the yield surface in the case of a transition from elastic to elastoplastic behaviour. Moreover, it can be proven that implicit integration, under certain conditions, leads to a symmetric and positive differential matrix $\partial\underline{\varepsilon}\ /\ \partial\underline{\sigma}$, which has a positive influence on iterative procedures. Because of these major advantages, restriction is made here to implicit integration and no attention is given to other types of time integration.

Hence, for $\omega = 1$ Eq. (2.15) reduces to:

$$\Delta\underline{\varepsilon}^p\ =\ \Delta\lambda\left(\frac{\partial g}{\partial\underline{\sigma}}\right)^{i} \tag{2.16}$$

Substitution of Eq. (2.16) into Eq. (2.14) and successively into Eq. (2.7) gives:

$$\underline{\sigma}^i\ =\ \underline{\sigma}^{tr} - \Delta\lambda\,\underline{\underline{D}}^e\left(\frac{\partial g}{\partial\underline{\sigma}}\right)^{i} \qquad \text{with:} \qquad \underline{\sigma}^{tr}\ =\ \underline{\sigma}^{i\text{-}1} + \underline{\underline{D}}^e\,\Delta\underline{\varepsilon} \tag{2.17}$$

In this relation $\underline{\sigma}^{tr}$ is an auxiliary stress vector, referred to as the *elastic stresses* or *trial stresses*, which is the new stress state when considering purely linear elastic material behaviour.

The increment of the plastic multiplier $\Delta\lambda$, as used in Eq. (2.17), can be solved from the condition that the new stress state has to satisfy the yield condition:

$$f\left(\underline{\sigma}^i\right) = 0 \tag{2.18}$$

For perfectly-plastic and linear hardening models the increment of the plastic multiplier can be written as:

$$\Delta\lambda = \frac{f\left(\underline{\sigma}^{tr}\right)}{d+h} \tag{2.19}$$

where:

$$d = \left(\frac{\partial f}{\partial \underline{\sigma}}\right)^{\underline{\sigma}^{tr}} \underline{\underline{D}}^{e} \left(\frac{\partial g}{\partial \underline{\sigma}}\right)^{i} \tag{2.20}$$

The symbol h denotes the hardening parameter, which is zero for perfectly-plastic models and constant for linear hardening models. In the latter case the new stress state can be formulated as:

$$\underline{\sigma}^{i} = \underline{\sigma}^{tr} - \frac{\left\langle f\left(\underline{\sigma}^{tr}\right)\right\rangle}{d+h} \underline{\underline{D}}^{e} \left(\frac{\partial g}{\partial \underline{\sigma}}\right)^{i} \tag{2.21}$$

The $\langle\ \rangle$-brackets are referred to as McCauley brackets, which have the following convention:

$$\langle x\rangle = 0 \quad \text{for:} \quad x \le 0 \qquad \text{and:} \qquad \langle x\rangle = x \quad \text{for:} \quad x > 0$$

For non-linear hardening models the increment of the plastic multiplier is obtained using a Newton-type iterative procedure with convergence control.

2.4 GLOBAL ITERATIVE PROCEDURE

Substitution of the relationship between increments of stress and increments of strain, $\Delta\underline{\sigma} = \underline{\underline{M}}\,\Delta\underline{\varepsilon}$, into the equilibrium equation (2.13) leads to:

$$\underline{\underline{K}}^{i}\,\Delta\underline{v}^{i} = \underline{f}_{ex}^{\,i} - \underline{f}_{in}^{\,i\text{-}1} \tag{2.22}$$

In this equation $\underline{\underline{K}}$ is a stiffness matrix, $\Delta\underline{v}$ is the incremental displacement vector, $\underline{f}_{ex}$ is the external force vector and $\underline{f}_{in}$ is the internal reaction vector. The superscript i refers to the step number. However, because the relation between stress increments and strain increments is generally non-linear, the stiffness matrix cannot be formulated exactly beforehand. Hence, a global iterative procedure is required to satisfy both the equilibrium condition and the constitutive relation. The global iteration process can be written as:

$$\underline{\underline{K}}^{j}\,\delta\underline{v}^{j} = \underline{f}_{ex}^{\,i} - \underline{f}_{in}^{\,j\text{-}1} \tag{2.23}$$

The superscript j refers to the iteration number. $\delta \underline{v}$ is a vector containing sub-incremental displacements, which contribute to the displacement increments of step i:

$$\Delta \underline{v}^i = \sum_{j=1}^{n} \delta \underline{v}^j \tag{2.24}$$

where n is the number of iterations within step i. The stiffness matrix $\underline{\underline{K}}$, as used in Eq. (2.23), represents the material behaviour in an approximated manner. The more accurate the stiffness matrix, the fewer iterations are required to obtain equilibrium within a certain tolerance.

In its simplest form $\underline{\underline{K}}$ represents a linear-elastic response. In this case the stiffness matrix can be formulated as:

$$\underline{\underline{K}} = \int \underline{\underline{B}}^T \underline{\underline{D}}^e \underline{\underline{B}} \, dV \qquad \text{(elastic stiffness matrix)} \tag{2.25}$$

where $\underline{\underline{D}}^e$ is the elastic material matrix according to Hooke's law and $\underline{\underline{B}}$ is the strain interpolation matrix. The use of an elastic stiffness matrix gives a robust iterative procedure as long as the material stiffness does not increase, even when using non-associated plasticity models. Special techniques such as arc-length control (Riks, 1979), over-relaxation and extrapolation (Vermeer & Van Langen, 1989) can be used to improve the iteration process. Moreover, the automatic step size procedure, as introduced by Van Langen & Vermeer (1990), can be used to improve the practical applicability. For material models with linear behaviour in the elastic domain, such as the standard Mohr-Coulomb model, the use of an elastic stiffness matrix is particularly favourable, as the stiffness matrix needs only be formed and decomposed before the first calculation step. This calculation procedure is summarised in Appendix A.

For hardening-plasticity models with stress-dependent stiffness behaviour, the stiffness matrix is based on the elastic stiffness at the beginning of each step. Hence, for such models the stiffness matrix is updated at the beginning of each new step on the basis of the stress state at the end of the previous step and kept constant during the equilibrium iteration procedure.

3 CONSOLIDATION THEORY

In this chapter we will review the theory of consolidation as used in PLAXIS. In addition to a general description of Biot's theory for coupled consolidation, attention is focused on the finite element formulation. Moreover, a separate section is devoted to the use of advanced soil models in a consolidation analysis (elastoplastic consolidation).

3.1 BASIC EQUATIONS OF CONSOLIDATION

The governing equations of consolidation as used in PLAXIS follow Biot's theory (Biot, 1956). Darcy's law for fluid flow and elastic behaviour of the soil skeleton are also assumed. The formulation is based on small strain theory. According to Terzaghi's principle, stresses are divided into effective stresses and pore pressures:

$$\underline{\sigma} = \underline{\sigma}' + \underline{m}\left(p_{steady} + p_{excess}\right) \tag{3.1}$$

where:

$$\underline{\sigma} = (\sigma_{xx}\ \sigma_{yy}\ \sigma_{zz}\ \sigma_{xy}\ \sigma_{yz}\ \sigma_{zx})^T \qquad \text{and:} \qquad \underline{m} = (\,1\ 1\ 1\ 0\ 0\ 0\,)^T \tag{3.2}$$

$\underline{\sigma}$ is the vector with total stresses, $\underline{\sigma}'$ contains the effective stresses, p_{excess} is the excess pore pressure and m is a vector containing unity terms for normal stress components and zero terms for the shear stress components. The steady state solution at the end of the consolidation process is denoted as p_{steady}. Within PLAXIS p_{steady} is defined as:

$$p_{steady} = p_{input} \tag{3.3}$$

where p_{input} is the pore pressure generated in the input program based on phreatic lines after the use of the *K0 procedure* or gravity loading. Note that within PLAXIS compressive stresses are considered to be negative; this applies to effective stresses as well as to pore pressures. In fact it would be more appropriate to refer to p_{excess} and p_{steady} as pore stresses, rather than pressures. However, the term pore pressure is retained, although it is positive for tension.

The constitutive equation is written in incremental form. Denoting an effective stress increment as $\dot{\underline{\sigma}}'$ and a strain increment as $\dot{\underline{\varepsilon}}$, the constitutive equation is:

$$\dot{\underline{\sigma}}' = \underline{\underline{M}}\,\dot{\underline{\varepsilon}} \tag{3.4}$$

where:

$$\dot{\underline{\varepsilon}} = \left(\dot{\varepsilon}_{xx}\dot{\varepsilon}_{yy}\dot{\varepsilon}_{zz}\dot{\gamma}_{xy}\dot{\gamma}_{yz}\dot{\gamma}_{zx}\right)^T \tag{3.5}$$

and $\underline{\underline{M}}$ represents the material stiffness matrix. For details on constitutive relations, see the Material Models manual.

3.2 FINITE ELEMENT DISCRETISATION

To apply a finite element approximation we use the standard notation:

$$\underline{u} = \underline{\underline{N}}\,\underline{v} \qquad \underline{p} = \underline{\underline{N}}\,\underline{p_n} \qquad \underline{\varepsilon} = \underline{\underline{B}}\,\underline{v} \tag{3.6}$$

where $\underline{v}$ is the nodal displacement vector, $\underline{p_n}$ is the excess pore pressure vector, $\underline{u}$ is the continuous displacement vector within an element and $\underline{p}$ is the (excess) pore pressure. The matrix $\underline{\underline{N}}$ contains the interpolation functions and $\underline{\underline{B}}$ is the strain interpolation matrix.

In general the interpolation functions for the displacements may be different from the interpolation functions for the pore pressure. In PLAXIS, however, the same functions are used for displacements and pore pressures.

Starting from the incremental equilibrium equation and applying the above finite element approximation we obtain:

$$\int \underline{\underline{B}}^T d\underline{\sigma}\, dV = \int \underline{\underline{N}}^T d\,\underline{f}\, dV + \int \underline{\underline{N}}^T d\,\underline{t}\, dS + \underline{r}_0 \tag{3.7}$$

with:

$$\underline{r}_0 = \int \underline{\underline{N}}^T \underline{f}_0 dV + \int \underline{\underline{N}}^T \underline{t}_0 dS - \int \underline{\underline{B}}^T \underline{\sigma}_0 dV \tag{3.8}$$

where $\underline{f}$ is a body force due to self-weight and $\underline{t}$ represents the surface tractions. In general the residual force vector, $\underline{r_0}$, will be equal to zero, but solutions of previous load steps may have been inaccurate. By adding the residual force vector the computational procedure becomes self-correcting. The term dV indicates integration over the volume of the body considered and dS indicates a surface integral.

Dividing the total stresses into pore pressure and effective stresses and introducing the constitutive relationship gives the nodal equilibrium equation:

$$\underline{\underline{K}}\;\, d\,\underline{v} + \underline{\underline{L}}\;\, d\,\underline{p}_n = d\,\underline{f}_n \tag{3.9}$$

where $\underline{\underline{K}}$ is the stiffness matrix, $\underline{\underline{L}}$ is the coupling matrix and $d\underline{f}_n$ is the incremental load vector:

$$\underline{\underline{K}} = \int \underline{\underline{B}}^T \underline{\underline{M}}\, \underline{\underline{B}}\, dV \tag{3.10a}$$

$$\underline{\underline{L}} = \int \underline{\underline{B}}^T \underline{m}\, \underline{\underline{N}}\, dV \tag{3.10b}$$

$$d\underline{f}_n = \int \underline{\underline{N}}^T d\underline{f}\, dV + \int \underline{\underline{N}}^T d\underline{t}\, dS \tag{3.10c}$$

To formulate the flow problem, the continuity equation is adopted in the following form:

$$\nabla^T \; \underline{\underline{R}} \; \nabla \; (\gamma_w \;\; y \; - \; p_{steady} \; - \; p \;) \; / \; \gamma_w - \underline{m}^T \;\; \frac{\partial \underline{\varepsilon}}{\partial t} + \frac{n}{K_w} \; \frac{\partial p}{\partial t} = 0 \tag{3.11}$$

where $\underline{\underline{R}}$ is the permeability matrix:

$$\underline{\underline{R}} = \begin{bmatrix} k_x & 0 & 0 \\ 0 & k_y & 0 \\ 0 & 0 & k_z \end{bmatrix} \tag{3.12}$$

n is the porosity, K_w is the bulk modulus of the pore fluid and γ_w is the unit weight of the pore fluid. This continuity equation includes the sign convention that p_{steady} and p are considered positive for tension.

As the steady state solution is defined by the equation:

$$\nabla^T \; \underline{\underline{R}} \; \nabla \; (\gamma_w \;\; y \; - \; p_{steady} \;) \; / \; \gamma_w = 0 \tag{3.13}$$

the continuity equation takes the following form:

$$\nabla^T \; \underline{\underline{R}} \; \nabla \; p \; / \; \gamma_w + \underline{m}^T \;\; \frac{\partial \underline{\varepsilon}}{\partial t} \; - \; \frac{n}{K_w} \; \frac{\partial p}{\partial t} = 0 \tag{3.14}$$

Applying finite element discretisation using a Galerkin procedure and incorporating prescribed boundary conditions we obtain:

$$- \; \underline{\underline{H}} \;\; \underline{p}_n + \underline{\underline{L}}^T \;\; \frac{d\,\underline{v}}{d\,t} - \underline{\underline{S}} \;\; \frac{d\,\underline{p}_n}{d\,t} \; = \; \underline{q} \tag{3.15}$$

where:

$$\underline{\underline{H}} = \int \left(\nabla \underline{\underline{N}} \right)^T \underline{\underline{R}} \nabla \underline{\underline{N}} \, / \, \gamma_w dV \; , \qquad \underline{\underline{S}} = \int \frac{n}{K_w} \underline{\underline{N}}^T \underline{\underline{N}} dV \tag{3.16}$$

and $\underline{q}$ is a vector due to prescribed outflow at the boundary. However within PLAXIS it is not possible to have boundaries with non-zero prescribed outflow. The boundary is either closed (zero flux) or open (zero excess pore pressure). In reality the bulk modulus of water is very high and so the compressibility of water can be neglected in comparison to the compressibility of the soil skeleton.

In PLAXIS the bulk modulus of the pore fluid is taken automatically according to (also see Reference Manual):

$$\frac{K_w}{n} = \frac{3(\nu_u - \nu)}{(1-2\nu_u)(1+\nu)} K_{skeleton} \tag{3.17}$$

Where ν_u has a default value of 0.495. For drained material and material in clusters that have just been switched on, the bulk modulus of the pore fluid is neglected.

The equilibrium and continuity equations may be compressed into a block matrix equation:

$$\begin{bmatrix} \underline{\underline{K}} & \underline{\underline{L}} \\ \underline{\underline{L}}^T & -\underline{\underline{S}} \end{bmatrix} \begin{bmatrix} \frac{d\,\underline{v}}{d\,t} \\ \frac{d\,\underline{p}_n}{d\,t} \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & \underline{\underline{H}} \end{bmatrix} \begin{bmatrix} \underline{v} \\ \underline{p}_n \end{bmatrix} + \begin{bmatrix} \frac{d\,\underline{f}_n}{d\,t} \\ \underline{q}_n \end{bmatrix} \tag{3.18}$$

A simple step-by-step integration procedure is used to solve this equation. Using the symbol Δ to denote finite increments, the integration gives:

$$\begin{bmatrix} \underline{\underline{K}} & \underline{\underline{L}} \\ \underline{\underline{L}}^T & -\underline{\underline{S}}^* \end{bmatrix} \begin{bmatrix} \Delta\underline{v} \\ \Delta\underline{p}_n \end{bmatrix} = \begin{bmatrix} 0 & 0 \\ 0 & \Delta t\ \underline{\underline{H}} \end{bmatrix} \begin{bmatrix} \underline{v}_0 \\ \underline{p}_{n0} \end{bmatrix} + \begin{bmatrix} \Delta\,\underline{f}_n \\ \Delta t\ \underline{q}_n^* \end{bmatrix} \tag{3.19}$$

where:

$$\underline{\underline{S}}^* = \alpha\ \Delta t\ \underline{\underline{H}} + \underline{\underline{S}} \qquad \underline{q}_n^* = \underline{q}_{n0} + \alpha\ \Delta\,\underline{q}_n \tag{3.20}$$

and $\underline{v}_0$ and $\underline{p}_{n0}$ denote values at the beginning of a time step. The parameter α is the time integration coefficient. In general the integration coefficient α can take values from 0 to 1. In PLAXIS the fully implicit scheme of integration is used with $\alpha = 1$.

3.3 ELASTOPLASTIC CONSOLIDATION

In general, when a non-linear material model is used, iterations are needed to arrive at the correct solution. Due to plasticity or stress-dependent stiffness behaviour the equilibrium equations are not necessarily satisfied using the technique described above. Therefore the equilibrium equation is inspected here. Instead of Eq. (4.9) the equilibrium equation is written in sub-incremental form:

$$\underline{\underline{K}}\ \delta\,\underline{v} + \underline{\underline{L}}\ \delta\,\underline{p}_n = \underline{r}_n \tag{3.21}$$

where $\underline{r}_n$ is the global residual force vector. The total displacement increment $\Delta\underline{v}$ is the summation of sub-increments $\delta\underline{v}$ from all iterations in the current step:

$$\underline{r}_n = \int \underline{\underline{N}}^T\,\underline{f}\ dV + \int \underline{\underline{N}}^T\,\underline{t}\ dS - \int \underline{\underline{B}}^T\,\underline{\sigma}\ dV \tag{3.22}$$

with:

$$\underline{f} = \underline{f}_0 + \Delta \underline{f} \qquad \text{and:} \qquad \underline{t} = \underline{t}_0 + \Delta \underline{t} \tag{3.23}$$

In the first iteration we consider $\underline{\sigma} = \underline{\sigma}_0$, i.e. the stress at the beginning of the step. Successive iterations are used on the current stresses that are computed from the appropriate constitutive model.

4 ELEMENT FORMULATIONS

In this chapter the interpolation functions of the finite elements used in the PLAXIS 3D FOUNDATION program are described. Each element consists of a number of nodes. Each node has a number of degrees of freedom that correspond to discrete values of the unknowns in the boundary value problem to be solved. In the case of deformation theory the degrees of freedom correspond to the displacement components. In addition to the interpolation functions it is described which type of numerical integration over elements is used in the program.

4.1 INTERPOLATION FUNCTIONS AND NUMERICAL INTEGRATION OF LINE ELEMENTS

Within an element the displacement field $\underline{u} = (u_x\ u_y\ u_z)^T$ is obtained from the discrete nodal values in a vector $\underline{v} = (v_1\ v_2\ \ldots\ v_n)^T$ using interpolation functions assembled in matrix $\underline{\underline{N}}$:

$$\underline{u} = \underline{\underline{N}}\,\underline{v} \tag{4.1}$$

Hence, interpolation functions $\underline{\underline{N}}$ are used to interpolate values inside an element based on known values in the nodes. Interpolation functions are also denoted as shape functions.

Let us first consider a line element. Line elements are the basis for distributed loads on vertical planes in the 3D model. The extension of this theory to areas and volumes is given in the subsequent sections.

When the local position, ξ, of a point (usually a stress point or an integration point) is known, one can write for a displacement component u:

$$u(\xi) = \sum_{i=1}^{n} N_i\ (\xi) v_i \tag{4.2}$$

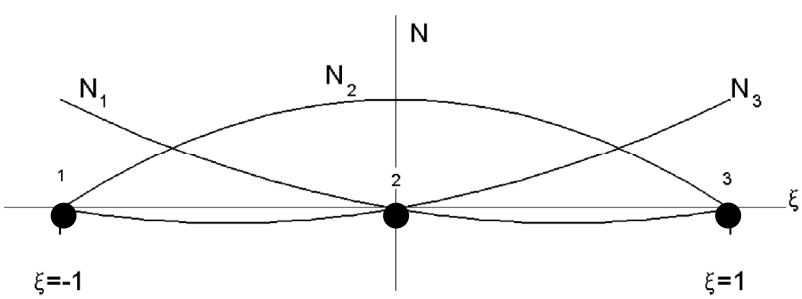

Figure 4.1 Shape functions for a 3-node line element

where:

v_i	the nodal values,
$N_i(\xi)$	the value of the shape function of node i at position ξ,
$u(\xi)$	the resulting value at position ξ and
n	the number of nodes per element.

4.1.1 3-NODE LINE ELEMENTS

In Fig. 4.1, an example of a 3-node line element is given, which is compatible with the side of a 6-node triangle, an 8-node quadrilateral or a 15-node volume element in the PLAXIS 3D FOUNDATION program, since these elements also have three nodes on a side. The shape functions N_i have the property that the function value is equal to unity at node i and zero at the other nodes. For 3-node line elements, where nodes 1, 2 and 3 are located at $\xi = -1$, 0 and 1 respectively, the shape functions are given by:

$$N_1 = -\tfrac{1}{2}(1-\xi)\,\xi \qquad (4.3)$$

$$N_2 = (1+\xi)(1-\xi)$$

$$N_3 = \tfrac{1}{2}(1+\xi)\,\xi$$

3-node line elements provide a second-order interpolation of displacements. These elements are the basis for distributed line loads and for beam elements.

4.1.2 NUMERICAL INTEGRATION OF LINE ELEMENTS

In order to obtain the integral over a certain line, the integral is numerically estimated as:

$$\int_{\xi=-1}^{1} F(\xi)d\xi \approx \sum_{i=1}^{k} F(\xi_i)\,w_i \qquad (4.4)$$

where $F(\xi_i)$ is the value of the function F at position ξ_i and w_i the weight factor for point i. A total of k sampling points is used. A method that is commonly used for numerical integration is Gaussian integration, where the positions ξ_i and weights w_i are chosen in a special way to obtain high accuracy. For Gaussian-integration a polynomial function of degree $2k$-1 can be integrated exactly by using k points. The position and weight factors of the two types of integration are given in Table 4.1. Note that the sum of the weight factors is equal to 2, which is equal to the length of the line in local coordinates. The type of integration used for the 3-node line elements is shaded.

Table 4.1 Gaussian integration

	ξ_i	w_i	max. polyn. degree
1 point	0.000000...	2	1
2 points	±0.577350...(±1/√3)	1	3
3 points	±0.774596... (±√0.6) 0.000000...	0.55555... (5/9) 0.88888... (8/9)	5
4 points	±0.861136... ±0.339981...	0.347854... 0.652145...	7
5 points	±0.906179... ±0.538469... 0.000000...	0.236926... 0.478628... 0.568888...	9

4.2 INTERPOLATION FUNCTIONS AND NUMERICAL INTEGRATION OF AREA ELEMENTS

Areas and surfaces in the PLAXIS 3D FOUNDATION program are either formed by 6-node triangular elements or by 8-node quadrilateral elements. The interpolation functions and the type of integration of these elements is described in the following subsections.

4.2.1 6-NODE TRIANGULAR ELEMENTS

The 6-node triangles are created in the 2D mesh generation process and used in the (pseudo-) horizontal planes of the 3D model to form the faces of the 15-node wedge elements for soil. The 6-node triangles are also the basis for floor elements and distributed loads on work planes in the 3D model.

For triangular elements there are two local coordinates (ξ and η). In addition we use an auxiliary coordinate $\zeta = 1-\xi-\eta$. 6-node triangular elements provide a second-order interpolation of displacements. The shape functions can be written as (see the local node numbering as shown in Figure 4.2) :

$$\begin{aligned} N_1 &= \zeta(2\zeta-1) \\ N_2 &= \xi(2\xi-1) \\ N_3 &= \eta(2\eta-1) \\ N_4 &= 4\,\zeta\,\xi \\ N_5 &= 4\,\xi\,\eta \\ N_6 &= 4\,\eta\,\zeta \end{aligned} \tag{4.5}$$

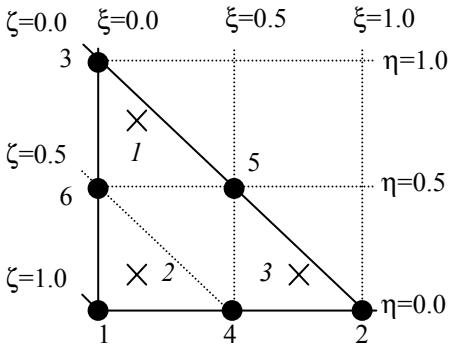

Figure 4.2 Local numbering and positioning of nodes (•) and integration points (x) of a 6-node triangular element

As for line elements, one can formulate the numerical integration over areas as:

$$\int\int F(\xi,\eta)\ d\xi\ d\eta \approx \sum_{i=1}^{k} F(\xi_i,\eta_i) w_i \tag{4.6}$$

The PLAXIS 3D FOUNDATION program uses Gaussian integration within the area elements. For 6-node triangular elements the integration is based on 3 sample points (see Fig. 4.2). The position and weight factors of the integration points are given in Table 4.2. Note that the sum of the weight factors is equal to 1.

Table 4.2 3-point Gaussian integration for 6-node triangular elements

Point	ξ_i	η_i	w_i
1	1/6	2/3	1/3
2	1/6	1/6	1/3
3	2/3	1/6	1/3

4.2.2 8-NODE QUADRILATERAL ELEMENTS

The 8-node quadrilateral elements are created in the 3D mesh extension process and they are used at the faces of the 15-node wedge elements in the *y*-direction. These elements are the basis for wall elements and distributed loads between work planes in the 3D model and for interface elements. 8-node quadrilateral elements provide a second-order interpolation of displacements. Quadrilateral elements have two local coordinates (ξ and η). The shape functions of 8-node elements can be written as (see the local node numbering as shown in Figure 4.3):

$$N_1 = (1-\xi)\,(1-\eta)\,(-1-\xi-\eta)\,/\,4 \tag{4.7}$$

$$N_2 = (1+\xi)\,(1-\eta)\,(-1+\xi-\eta)\,/\,4$$

$$N_3 = (1+\xi)(1+\eta)(-1+\xi+\eta)/4$$

$$N_4 = (1-\xi)(1+\eta)(-1-\xi+\eta)/4$$

$$N_5 = (1-\xi)(1+\xi)(1-\eta)/2$$

$$N_6 = (1-\xi)(1+\xi)(1+\eta)/2$$

$$N_7 = (1-\eta)(1+\eta)(1+\xi)/2$$

$$N_8 = (1-\eta)(1+\eta)(1-\xi)/2$$

For 8-node quadrilateral elements the numerical integration is based on 4 (2x2) Gauss points (see Figure 4.3), equivalent to the integration of line elements, but in two directions. The position and weight factors of the integration points are given in Table 4.3. The sum of the weight factors is equal to 4, which is equal to the area of the quadrilateral in local coordinates.

Table 4.3 4-point Gaussian integration for 8-node quadrilateral elements

Point	ξ_i	η_i	w_i
1	$-1/3\sqrt{3}$	$-1/3\sqrt{3}$	1
2	$+1/3\sqrt{3}$	$-1/3\sqrt{3}$	1
3	$-1/3\sqrt{3}$	$+1/3\sqrt{3}$	1
4	$+1/3\sqrt{3}$	$+1/3\sqrt{3}$	1

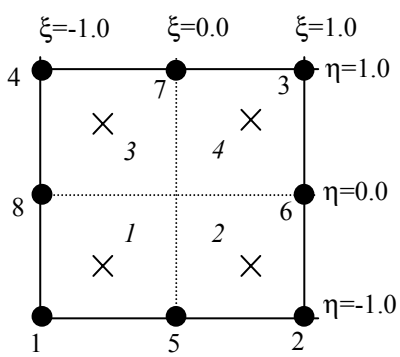

Figure 4.3 Local numbering and positioning of nodes (•) and integration points (x) of an 8-node quadrilateral element

4.2.3 STRUCTURAL ELEMENTS

Structural area elements in the PLAXIS 3D FOUNDATION program, i.e. beams, floors, walls and interfaces are based on the line elements and area elements as described in the previous sections. However, there are some differences.

Beam Elements

The 3-node beam elements are used to describe semi-one-dimensional structural objects with flexural rigidity. Beam elements are slightly different from 3-node line element in the sense that they have six degrees of freedom per node instead of three, i.e. three translational d.o.f.s (u_x, u_y, u_z) and three rotational d.o.f.s (ϕ_x, ϕ_y, ϕ_z). These elements are directly integrated over their cross-section and numerically integrated along their length using 4-point Gaussian integration according to Table 4.1. The element provides a quadratic interpolation of the longitudinal displacements (See Eq. 4.3) and a fifth-order interpolation of transverse displacements. For beam elements there is one local coordinate (ξ). The shape function for transverse displacement can be written as (see the local node numbering as shown in Figure 4.1):

$$
\begin{aligned}
N_{1u} &= (4\xi^2 - 5\,\xi^3 - 2\,\xi^4 + 3\,\xi^5)\,/\,4 \\
N_{2u} &= 1 - 2\,\xi^2 + \xi^4 \\
N_{3u} &= (4\,\xi^2 + 5\,\xi^3 - 2\,\xi^4 - 3\,\xi^5)\,/\,4 \\
N_{1\varphi} &= (\xi^2 - \xi^3 - \xi^4 + \xi^5)\,/\,4 \\
N_{2\varphi} &= \xi - 2\xi^3 + \xi^5 \\
N_{3\varphi} &= (-\xi^2 - \xi^3 + \xi^4 + \xi^5)\,/\,4
\end{aligned}
\qquad (4.8)
$$

Wall elements

Wall elements are slightly different from 8-node quadrilaterals in the sense that they have six degrees of freedom per node instead of three, i.e. three translational d.o.f.s (u_x, u_y, u_z) and three rotational d.o.f.s (ϕ_x, ϕ_y, ϕ_z). These elements are directly integrated over their cross-section and numerically integrated over their area using 4 (2x2) point Gaussian integration. The position of the integration points is indicated in Figure 4.4 and corresponds with Table 4.3.

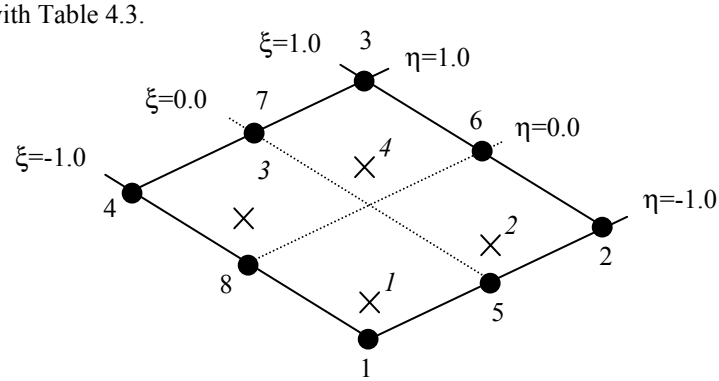

Figure 4.4 Local numbering and positioning of nodes (•) and integration points (x) of an 8-node plate element

Floor elements

Floor elements are different from the 6-node triangles in the sense that they have six degrees of freedom per node instead of three, i.e. three translational d.o.f.s (u_x, u_y, u_z) and three rotational d.o.f.s (ϕ_x, ϕ_y, ϕ_z). These elements are directly integrated over their cross-section and numerically integrated using 3 point Gaussian integration. The position of the integration points is indicated in Figure 4.5 and corresponds with Table 4.2.

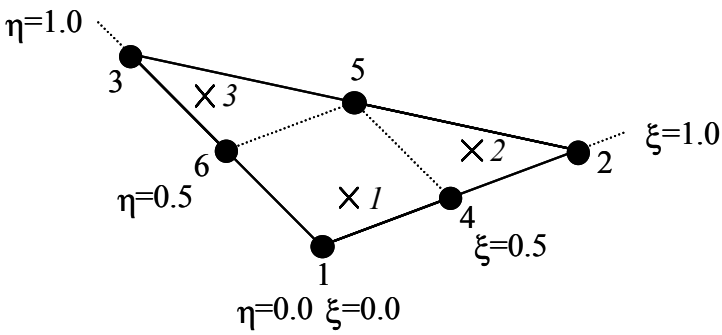

Figure 4.5 Local numbering and positioning of nodes (•) and integration points (x) of a 6-node plate triangle.

Interface elements

Interface elements are different from the 8-node quadrilaterals in the sense that they have pairs of nodes instead of single nodes. Moreover, interface elements have a 3x3 point Gaussian integration instead of 2x2. The position and numbering of the nodes and integration points is indicated in Figure 4.6 (see also Table 4.4). The distance between the two nodes of a node pair is zero. Each node has three translational degrees of freedom (u_x, u_y, u_z). As a result, interface elements allow for differential displacements between the node pairs (slipping and gapping). For more information see Van Langen (1991).

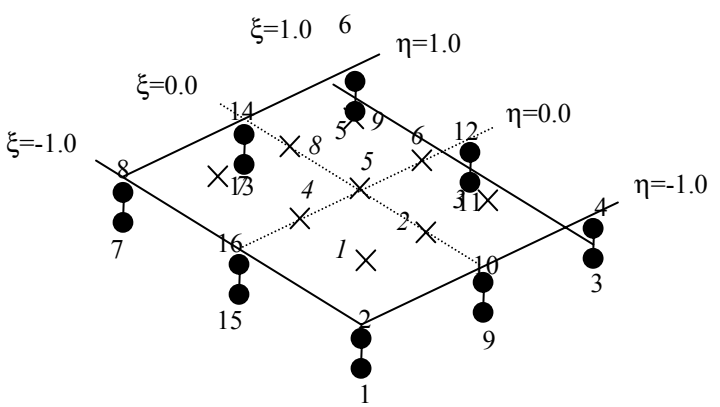

Figure 4.6 Local numbering and positioning of nodes (•) and integration points (x) of a 16-node interface element

Table 4.4 9-point Gaussian integration for 16-node interface elements

Point	ξ_i	η_i	w_i
1	–0.774596	–0.774596	0.308642
2	0.000000	–0.774596	0.493827
3	+0.774596	–0.774596	0.308642
4	–0.774596	0.000000	0.493827
5	0.000000	0.000000	0.790124
6	+0.774596	0.000000	0.493827
7	–0.774596	+0.774596	0.308642
8	0.000000	+0.774596	0.493827
9	+0.774596	+0.774596	0.308642

4.3 INTERPOLATION FUNCTIONS AND NUMERICAL INTEGRATION OF VOLUME ELEMENTS

The soil volume in the PLAXIS 3D FOUNDATION program is modelled by means of 15-node wedge elements. The interpolation functions, their derivatives and the numerical integration of this type of element are described in the following subsections.

4.3.1 15-NODE WEDGE ELEMENTS

The 15-node wedge elements are created in the 3D mesh extension procedure. This type of element provides a second-order interpolation of displacements. For wedge elements there are three local coordinates (ξ, η and ζ). The shape functions of these 15-node volume elements can be written as (see the local node numbering as shown in the Fig. 4.7):

$$N_1 = -(1-\xi-\eta)(1-\zeta)(+2\xi+2\eta+\zeta)/2 \quad (4.9)$$

$$N_2 = -\xi(1-\zeta)(2-2\xi-\zeta)/2$$

$$N_3 = -\eta(1-\zeta)(2-2\eta+\zeta)/2$$

$$N_4 = -(1-\xi-\eta)(1+\zeta)(+2\xi+2\eta-\zeta)/2$$

$$N_5 = -\xi(1+\zeta)(2-2\xi+\zeta)/2$$

$$N_6 = -\eta(1+\zeta)(2-2\eta-\zeta)/2$$

$$N_7 = (1-\xi-\eta)\,\xi(1-\zeta)*2$$

$$N_8 = \xi\,\eta(1-\zeta)*2$$

$$N_9 = \eta(1-\xi-\eta)(1-\zeta)*2$$

$$N_{10} = (1-\xi-\eta)(1-\zeta)(1+\zeta)$$

$$N_{11} = \xi(1-\zeta)(1+\zeta)$$

$$N_{12} = \eta\,(1-\zeta)\,(1+\zeta)$$

$$N_{13} = (1-\xi-\eta)\,\xi\,(1+\zeta) * 2$$

$$N_{14} = \xi\eta\,(1+\zeta) * 2$$

$$N_{15} = \eta\,(1-\xi-\eta)\,(1+\zeta) * 2$$

For one fold degenerated wedge elements, a 15 node wedge element is used in which the three nodes along one side coincide (Figure 4.8). For two fold degenerated wedge elements, a 15 node wedge element is used in which the nodes, five in total, along two sides coincide.

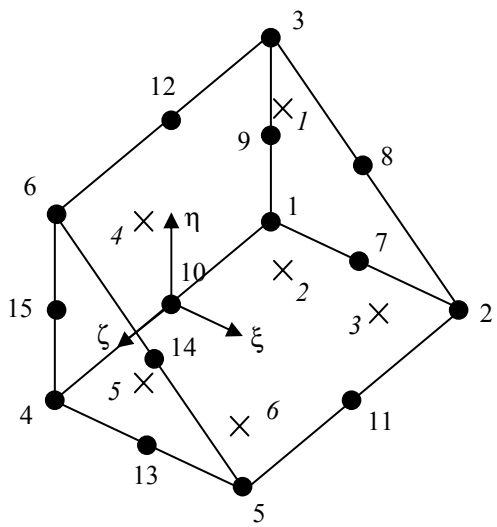

Figure 4.7 Local numbering and positioning of nodes (•) and integration points (x) of a 15-node wedge element

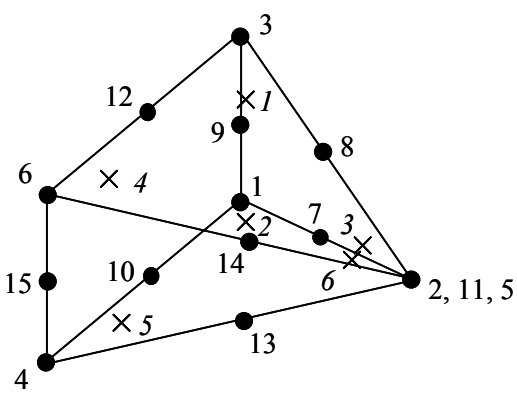

Figure 4.8 Local numbering and positioning of nodes (•) and integration points (x) of a one fold degenerated (reduced) 15-node wedge element

4.3.2 NUMERICAL INTEGRATION OVER VOLUMES

As for lines and areas, one can formulate the numerical integration over volumes as:

$$\int\int\int F(\xi,\eta,\zeta)\,d\xi\,d\eta\,d\zeta \approx \sum_{i=1}^{k} F(\xi_i,\eta_i,\zeta_i)w_i \tag{4.10}$$

The PLAXIS 3D FOUNDATION program uses Gaussian integration within the wedge elements. For 15-node wedge elements the integration is based on 6 sample points. The integration is a mixture between the 3-point integration of a 6-node triangular element and the 4-point integration of an 8-node quadrilateral. The position and weight factors of the integration points are given in Table 4.5. See Figure 4.7 and Figure 4.8 for the local numbering of integration points. Note that the sum of the weight factors are equal to 2.

Table 4.5 6-point Gaussian integration for 15-node wedge element

Point	ξ_i	η_i	ζ_i	w_i
1	1/6	2/3	$-1/3\sqrt{3}$	1/3
2	1/6	1/6	$-1/3\sqrt{3}$	1/3
3	2/3	1/6	$-1/3\sqrt{3}$	1/3
4	1/6	2/3	$+1/3\sqrt{3}$	1/3
5	1/6	1/6	$+1/3\sqrt{3}$	1/3
6	2/3	1/6	$+1/3\sqrt{3}$	1/3

4.3.3 DERIVATIVES OF SHAPE FUNCTIONS

In order to calculate Cartesian strain components from displacements, such as formulated in Eq. (2.10), derivatives need to be taken with respect to the global system of axes (x,y,z).

$$\underline{\varepsilon} = \underline{\underline{B}}_i\,\underline{v}_i \tag{4.11}$$

where

$$\underline{\underline{B}}_i = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} & 0 & \frac{\partial N_i}{\partial x} \end{bmatrix} \tag{4.12}$$

Within the elements, derivatives are calculated with respect to the local coordinate system (ξ, η, ζ). The relationship between local and global derivatives involves the Jacobian J:

$$\begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \varsigma} \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \varsigma} & \frac{\partial y}{\partial \varsigma} & \frac{\partial z}{\partial \varsigma} \end{bmatrix} \begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} = J \begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} \tag{4.13}$$

Or inversely:

$$\begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} = J^{-1} \begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \varsigma} \end{bmatrix} \tag{4.14}$$

The local derivatives $\partial N_i/\partial \xi$, etc., can easily be derived from the element shape functions, since the shape functions are formulated in local coordinates. The components of the Jacobian are obtained from the differences in nodal coordinates. The inverse Jacobian J^{-1} is obtained by numerically inverting J.

The Cartesian strain components can now be calculated by summation of all nodal contributions:

$$\begin{bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} = \sum_i \underline{\underline{B}}_i \begin{bmatrix} v_{x,i} \\ v_{y,i} \\ v_{z,i} \end{bmatrix} \tag{4.15}$$

where v_i are the displacement components in node i.

4.3.4 CALCULATION OF ELEMENT STIFFNESS MATRIX

The element stiffness matrix, K^e, is calculated by the integral (see also Eq. 2.25):

$$\underline{\underline{K}}^e = \int \underline{\underline{B}}^T \underline{\underline{D}}^e \underline{\underline{B}} \, dV \tag{4.16}$$

The integral is estimated by numerical integration as described in Section 4.3.2. In fact, the element stiffness matrix is composed of submatrices K_{ij}^e where i and j are the local nodes. The process of calculating the element stiffness matrix can be formulated as:

$$\underline{\underline{K}}_{ij}^e = \sum_k \underline{\underline{B}}_i^T \underline{\underline{D}}^e \underline{\underline{B}}_j w_k \tag{4.17}$$

5 REFERENCES

[1] Biot M.A. (1956), General solutions of the equations of elasticity and consolidation for porous material. Journal of Applied Mechanics, Vol. 23 No. 2

[2] Brinkgreve R.B.J. (1994), Geomaterial Models and Numerical Analysis of Softening. Dissertation. Delft University of Technology.

[3] Riks E. (1979), An incremental approach to the solution of snapping and buckling problems. Int. J. Solids & Struct. Vol. 15, pp. 529-551.

[4] Van Langen H. (1991), Numerical analysis of soil-structure interaction. Dissertation. Delft University of Technology.

[5] Vermeer P.A. (1979), A modified initial strain method for plasticity problems. In: Proc. 3rd Int. Conf. Num. Meth. Geomech. Balkema, Rotterdam, pp. 377-387.

[6] Vermeer P.A., De Borst R. (1984), Non-associated plasticity for soils, concrete and rock. Heron, Vol. 29, No. 3.

[7] Vermeer P.A., Van Langen H. (1989), Soil collapse computations with finite elements. Ingenieur-Archiv 59, pp. 221-236.

[8] Van Langen H., Vermeer P.A. (1990), Automatic step size correction for non-associated plasticity problems. Int. J. Num. Meth. Eng. Vol. 29, pp. 579-598.

[9] Zienkiewicz O.C. (1967), The finite element method in structural and continuum mechanics. McGraw-Hill, London, UK.

APPENDIX A - CALCULATION PROCESS

Finite element calculation process based on the elastic stiffness matrix

Read input data

Form stiffness matrix $\underline{\underline{K}} = \int \underline{\underline{B}}^T \underline{\underline{D}}^e \underline{\underline{B}} \, dV$

- New step $i \rightarrow i + 1$
- Form new load vector $\underline{f}_{ex}^{i} = \underline{f}_{ex}^{i-1} + \Delta \underline{f}_{ex}$
- Form reaction vector $\underline{f}_{in} = \int \underline{\underline{B}}^T \underline{\sigma}_c^{i-1} \, dV$
- Calculate unbalance $\Delta \underline{f} = \underline{f}_{ex}^{i} - \underline{f}_{in}$
- Reset displacement increment $\Delta \underline{v} = 0$
  - New iteration $j \rightarrow j + 1$
  - Solve displacements $\delta \underline{v} = \underline{\underline{K}}^{-1} \Delta \underline{f}$ *
  - Update displacement increments $\Delta \underline{v}^{j} = \Delta \underline{v}^{j-1} + \delta \underline{v}$
  - Calculate strain increments $\Delta \underline{\varepsilon} = \underline{\underline{B}} \Delta \underline{v}$; $\delta \underline{\varepsilon} = \underline{\underline{B}} \delta \underline{v}$
  - Calculate stresses: Elastic $\underline{\sigma}^{tr} = \underline{\sigma}_c^{i-1} + \underline{\underline{D}}^e \Delta \underline{\varepsilon}$
    Equilibrium $\underline{\sigma}^{eq} = \underline{\sigma}_c^{i,j-1} + \underline{\underline{D}}^e \delta \underline{\varepsilon}$
    Constitutive $\underline{\sigma}_c^{i,j} = \underline{\sigma}^{tr} - \dfrac{\left\langle f\left(\underline{\sigma}^{tr}\right)\right\rangle}{d} \underline{\underline{D}}^e \dfrac{\partial g}{\partial \underline{\sigma}}$
  - Form reaction vector $\underline{f}_{in} = \int \underline{\underline{B}}^T \underline{\sigma}_c^{i,j} \, dV$
  - Calculate unbalance $\Delta \underline{f} = \underline{f}_{ex}^{i} - \underline{f}_{in}$
  - Calculate error $e = \dfrac{|\Delta \underline{f}|}{|\underline{f}_{ex}^{i}|}$
  - Accuracy check if $e > e_{tolerated} \rightarrow$ new iteration
- Update displacements $\underline{v}^{i} = \underline{v}^{i-1} + \Delta \underline{v}$
- Write output data (results)
- If not finished → new step

Finish

* The solution of the system of equations is done using a sparse iterative solution procedure with smart preconditioning. For normal elastoplastic deformation calculations the solution is

based on the Conjugate Gradient method (CG), whereas for consolidation calculations (resulting in an indefinite matrix) the solution is based on SYM-QMR[1]. The preconditioning is based on the elastic material stiffness matrix with diagonal scaling and using a variable drop tolerance.

1 Freund R.W., Jarre F. (1996). A QMR-based interior-point algorithm for solving linear programs. Mathematical Programming Series ~ B 76, pp. 183-210.

APPENDIX B - SYMBOLS

$\underline{\underline{B}}$	:	Strain interpolation matrix
$\underline{\underline{D}}^e$	:	Elastic material stiffness matrix representing Hooke's law
f	:	Yield function
$\underline{f}$	:	Load vector
g	:	Plastic potential function
$\underline{\underline{K}}$	:	Stiffness matrix
$\underline{\underline{L}}$	:	Differential operator
$\underline{\underline{M}}$	:	Material stiffness matrix
$\underline{\underline{N}}$	:	Matrix with shape functions
$\underline{p}$	:	Body forces vector
t	:	Time
$\underline{t}$	:	Boundary tractions
$\underline{u}$	:	Vector with displacement components
$\underline{v}$	:	Vector with nodal displacements
V	:	Volume
w	:	Weight factor
γ	:	Volumetric weight
$\underline{\varepsilon}$	:	Vector with strain components
λ	:	Plastic multiplier
$\xi\ \eta\ \zeta$	:	Local coordinates
$\underline{\sigma}$	:	Vector with stress components
ω	:	Integration constant (explicit: $\omega = 0$; implicit: $\omega = 1$)