

PLAXIS
3D FOUNDATION
Scientific Manual
Version 2

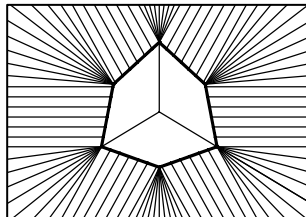


TABLE OF CONTENTS

1 Introduction.....1-1

2 Deformation theory.....2-1

2.1 Basic equations of continuum deformation.....2-1

2.2 Finite element discretisation2-2

2.3 Implicit integration of differential plasticity models.....2-3

2.4 Global iterative procedure.....2-5

3 Consolidation theory.....3-1

3.1 Basic equations of consolidation.....3-1

3.2 Finite element discretisation3-2

3.3 Elastoplastic consolidation.....3-4

4 Element formulations4-1

4.1 Interpolation functions of point elements4-1

4.1.1 Structural elements.....4-1

4.2 Interpolation functions and numerical integration of line elements4-2

4.2.1 Interpolation functions of line elements.....4-3

4.2.2 Numerical integration of line elements4-4

4.2.3 Structural elements.....4-4

4.2.4 Derivatives of interpolation functions.....4-7

4.2.5 Calculation of element stiffness matrix.....4-9

4.3 Interpolation functions and numerical integration of area elements ...4-10

4.3.1 Interpolation functions of area elements4-10

4.3.2 Numerical integration of area elements4-12

4.3.3 Structural elements.....4-13

4.4 Interpolation functions and numerical integration of volume elements4-15

4.4.1 15-node wedge elements.....4-16

4.4.2 Numerical integration over volumes4-17

4.4.3 Derivatives of interpolation functions.....4-18

4.4.4 Calculation of element stiffness matrix.....4-20

4.5 Special elements4-20

4.5.1 Embedded piles4-20

4.5.2 Ground anchor4-25

5 References.....5-1

Appendix A - Calculation process

Appendix B - Symbols

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 general chapters on deformation theory and consolidation theory, as well as 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:

$$\underline{\underline{L}}^T \underline{\underline{\sigma}} + \underline{\underline{b}} = \underline{\underline{0}} \quad (2.1)$$

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

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

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

$$\underline{\underline{\varepsilon}} = \underline{\underline{L}} \underline{\underline{u}} \quad (2.3)$$

This equation expresses the six strain components, assembled in vector $\underline{\underline{\varepsilon}}$, as the spatial derivatives of the three displacement components, assembled in vector $\underline{\underline{u}}$, 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:

$$\underline{\underline{\dot{\sigma}}} = \underline{\underline{M}} \underline{\underline{\dot{\varepsilon}}} \quad (2.4)$$

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):

$$\int \delta \underline{u}^T (\underline{L}^T \underline{\sigma} + \underline{b}) dV = 0 \quad (2.5)$$

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:

$$\int \delta \underline{\varepsilon}^T \underline{\sigma} dV = \int \delta \underline{u}^T \underline{b} dV + \int \delta \underline{u}^T \underline{t} dS \quad (2.6)$$

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:

$$\underline{\sigma}^i = \underline{\sigma}^{i-1} + \Delta \underline{\sigma} \quad \Delta \underline{\sigma} = \int \underline{\dot{\sigma}} dt \quad (2.7)$$

In this relation $\underline{\sigma}^i$ represents the actual state of stress which is unknown and $\underline{\sigma}^{i-1}$ 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):

$$\int \delta \underline{\varepsilon}^T \Delta \underline{\sigma} dV = \int \delta \underline{u}^T \underline{b}^i dV + \int \delta \underline{u}^T \underline{t}^i dS - \int \delta \underline{\varepsilon}^T \underline{\sigma}^{i-1} dV \quad (2.8)$$

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{N} :

$$\underline{u} = \underline{N} \underline{v} \quad (2.9)$$

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

$$\underline{\varepsilon} = \underline{L} \underline{N} \underline{v} = \underline{B} \underline{v} \quad (2.10)$$

In this relation \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:

$$\int (\underline{B} \delta \underline{v})^T \Delta \underline{\sigma} dV = \int (\underline{N} \delta \underline{v})^T \underline{b}^i dV + \int (\underline{N} \delta \underline{v})^T \underline{t}^i dS - \int (\underline{B} \delta \underline{v})^T \underline{\sigma}^{i-1} dV \quad (2.11)$$

The discrete displacements can be placed outside the integral:

$$\delta \underline{v}^T \int \underline{B}^T \Delta \underline{\sigma} dV = \delta \underline{v}^T \int \underline{N}^T \underline{b}^i dV + \delta \underline{v}^T \int \underline{N}^T \underline{t}^i dS - \delta \underline{v}^T \int \underline{B}^T \underline{\sigma}^{i-1} dV \quad (2.12)$$

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

$$\int \underline{B}^T \Delta \underline{\sigma} dV = \int \underline{N}^T \underline{b}^i dV + \int \underline{N}^T \underline{t}^i dS - \int \underline{B}^T \underline{\sigma}^{i-1} dV \quad (2.13)$$

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:

$$\Delta \underline{\sigma} = \underline{D}^e (\Delta \underline{\varepsilon} - \Delta \underline{\varepsilon}^p) \quad (2.14)$$

In this relation \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 $\Delta \underline{y}$ using the strain interpolation matrix \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:

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

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:

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

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

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

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:

$$f(\underline{\sigma}^i) = 0 \quad (2.18)$$

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

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

where:

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

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:

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

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

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

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:

$$\underline{\underline{K}}^i \Delta \underline{v}^i = \underline{f}_{ex}^i - \underline{f}_{in}^{i-1} \quad (2.22)$$

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:

$$\underline{\underline{K}}^j \delta \underline{v}^j = \underline{f}_{-ex}^i - \underline{f}_{-in}^{j-1} \quad (2.23)$$

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 :

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

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:

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

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:

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

where:

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

$\underline{\sigma}$ 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:

$$p_{steady} = p_{input} \quad (3.3)$$

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 $\underline{\dot{\sigma}}'$ and a strain increment as $\underline{\dot{\epsilon}}$, the constitutive equation is:

$$\underline{\dot{\sigma}}' = \underline{M} \underline{\dot{\epsilon}} \quad (3.4)$$

where:

$$\underline{\dot{\epsilon}} = (\dot{\epsilon}_{xx} \dot{\epsilon}_{yy} \dot{\epsilon}_{zz} \dot{\gamma}_{xy} \dot{\gamma}_{yz} \dot{\gamma}_{zx})^T \quad (3.5)$$

and \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:

$$\underline{u} = \underline{N} \underline{v} \quad p = \underline{N} \underline{p}_n \quad \underline{\varepsilon} = \underline{B} \underline{v} \quad (3.6)$$

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 p is the (excess) pore pressure. The matrix \underline{N} contains the interpolation functions and \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:

$$\int \underline{B}^T d\underline{\sigma} dV = \int \underline{N}^T d\underline{b} dV + \int \underline{N}^T d\underline{t} dS + \underline{r}_0 \quad (3.7)$$

with:

$$\underline{r}_0 = \int \underline{N}^T \underline{b}_0 dV + \int \underline{N}^T \underline{t}_0 dS - \int \underline{B}^T \underline{\sigma}_0 dV \quad (3.8)$$

where \underline{b} 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:

$$\underline{K} d\underline{v} + \underline{L} d\underline{p}_n = d\underline{f}_n \quad (3.9)$$

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

$$\underline{K} = \int \underline{B}^T \underline{M} \underline{B} dV \quad (3.10a)$$

$$\underline{L} = \int \underline{B}^T \underline{m} \underline{N} dV \quad (3.10b)$$

$$df_n = \int \underline{N}^T d\underline{b} dV + \int \underline{N}^T dt dS \quad (3.10c)$$

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

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

where \underline{R} is the permeability matrix:

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

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:

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

the continuity equation takes the following form:

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

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

$$- \underline{H} \underline{p}_n + \underline{L}^T \frac{d \underline{v}}{dt} - \underline{S} \frac{d \underline{p}_n}{dt} = \underline{q}_n \quad (3.15)$$

where:

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

and \underline{q}_n 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):

$$\frac{K_w}{n} = \frac{3(v_u - v)}{(1 - 2v_u)(1 + v)} K_{skeleton} \quad (3.17)$$

where v_u has a default value of 0.495. The value can be modified in the input program on the basis of Skempton's B-parameter. 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:

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

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

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

where:

$$\underline{\underline{S}}^* = \alpha \Delta t \underline{\underline{H}} + \underline{\underline{S}} \quad \underline{q}_n^* = \underline{q}_{n0} + \alpha \Delta \underline{q}_n \quad (3.20)$$

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. (3.9) the equilibrium equation is written in sub-incremental form:

$$\underline{\underline{K}} \delta \underline{v} + \underline{\underline{L}} \delta \underline{p}_n = \underline{r}_n \quad (3.21)$$

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:

$$\underline{r}_n = \int \underline{\underline{N}}^T \underline{b} dV + \int \underline{\underline{N}}^T \underline{t} dS - \int \underline{\underline{B}}^T \underline{\sigma} dV \quad (3.22)$$

with:

$$\underline{b} = \underline{b}_0 + \Delta b \quad \text{and:} \quad \underline{t} = \underline{t}_0 + \Delta \underline{t} \quad (3.23)$$

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. For consolidation problems degrees-of-freedom are both displacement components and (excess) pore pressures. 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 OF POINT ELEMENTS

Point elements are elements existing of only one single node. Hence, the displacement field of the element \underline{u} itself is only defined by the displacement field of this single node \underline{v} :

$$\underline{u} = \underline{v} \quad (4.1)$$

with $\underline{u} = (u_x \ u_y \ u_z)^T$ and $\underline{v} = (v_x \ v_y \ v_z)^T$.

4.1.1 STRUCTURAL ELEMENTS

Springs

In PLAXIS springs are considered to be point elements. The contribution of the spring element to the stiffness matrix can be derived from the traction the spring imposes on a point in the the geometry due to the displacement of this point (see Eq. (2.13)). As a spring has only an axial stiffness and no bending stiffness, it is more convenient to rotate the global displacement field \underline{v} to the displacement field \underline{v}^* such that the first axis of the rotated coordinate system coincides with the direction of the spring:

$$\underline{v}^* = \underline{R}_\theta \underline{v} \quad (4.2)$$

where \underline{R}_θ denotes the rotation matrix. As only axial displacements are relevant, the spring element will only have one degree of freedom in the rotated coordinate system. The traction in the rotated coordinate system \underline{t}^* can be derived as:

$$\underline{t}^* = D^s \underline{u}^* \quad (4.3)$$

where D^s denotes the constitutive relationship of a spring as defined in the Material Models Manual. Converting the traction in the rotated coordinate system to the traction in the global coordinate system \underline{t} by using the rotation matrix again and substituting Eq. (4.1) gives:

$$\underline{t} = \underline{R}_\theta^T D^s \underline{R}_\theta \underline{v} \quad (4.4)$$

Substituting this equation in Eq. (2.13) gives the element stiffness matrix of the spring $\underline{\underline{K}}^s$:

$$\underline{\underline{K}}^s = \underline{\underline{R}}_{\theta}^T \underline{\underline{D}}^s \underline{\underline{R}}_{\theta} \tag{4.5}$$

In case of non-linear behaviour of the spring the maximum stiffness will be used in the stiffness matrix whereas the non-linear behaviour is solved for iteratively.

4.2 INTERPOLATION FUNCTIONS AND NUMERICAL INTEGRATION OF LINE ELEMENTS

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

$$\underline{u} = \underline{\underline{N}} \underline{v} \tag{4.6}$$

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 line loads, beams and node-to-node anchors. 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 :

$$u(\xi) = \sum_{i=1}^n N_i(\xi) v_i \tag{4.7}$$

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.

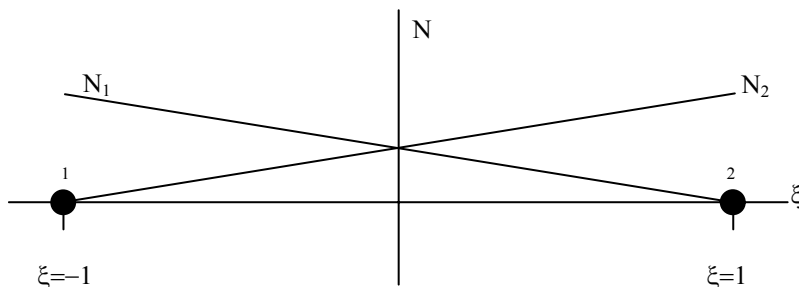


Figure 4.1 Shape functions for a 2-node line element

4.2.1 INTERPOLATION FUNCTIONS OF LINE ELEMENTS

Interpolation functions or shape functions are derived in a local coordinate system. This has several advantages like programming only one function per element type, a simple application of numerical integration and allowing higher-order elements to have curved edges.

2-node line elements

In Figure 4.1, an example of a 2-node line element is given. In contrast to a 3-node line element, this element is not compatible with an area element or volume element in the PLAXIS 3D FOUNDATION program. The 2-node line elements are the basis for node-to-node anchors, which are part of ground anchors. The shape functions N_i have the property that the function value is equal to unity at node i and zero at the other node. For 2-node line elements the nodes are located at $\xi = -1$ and $\xi = 1$. The shape functions are given by:

$$\begin{aligned} N_1 &= \frac{1}{2}(1-\xi) \\ N_2 &= \frac{1}{2}(1+\xi) \end{aligned} \quad (4.8)$$

2-node line elements provide a first-order (linear) interpolation of displacements.

3-node line elements

In Figure 4.2, 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:

$$\begin{aligned} N_1 &= -\frac{1}{2}(1-\xi)\xi \\ N_2 &= (1+\xi)(1-\xi) \\ N_3 &= \frac{1}{2}(1+\xi)\xi \end{aligned} \quad (4.9)$$

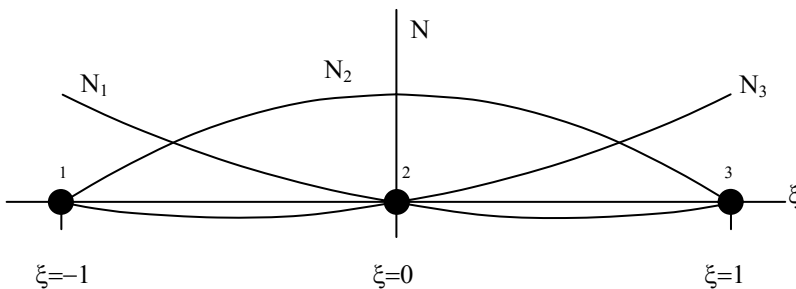


Figure 4.2 Shape functions for a 3-node line element

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

4.2.2 NUMERICAL INTEGRATION OF LINE ELEMENTS

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

$$\int_{\xi=-1}^1 F(\xi) d\xi \approx \sum_{i=1}^k F(\xi_i) w_i \quad (4.10)$$

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 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 types of integration used for the 2-node line elements and the 3-node line elements are shaded.

Table 4.1 Gaussian integration

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

4.2.3 STRUCTURAL ELEMENTS

Structural line elements in the PLAXIS 3D FOUNDATION program, i.e. node-to-node anchors and beams are based on the line elements as described in the previous sections. However, there are some differences.

Node-to-node anchors

Node-to-node anchors are springs that are used to model ties between two points. A node-to-node anchor consists of a 2-node element with both nodes shared with the structural elements or the volume elements the node-to-node anchor is attached to. Therefore, the nodes have three d.o.f.s in the global coordinate system. However, as a

node-to-node anchor can only sustain normal forces, only the displacement in the axial direction of the node-to-node anchor is relevant. Therefore it is more convenient to rotate the global coordinate system to a coordinate system in which the first axis coincides with the direction of the anchor. This rotated coordinate system is denoted as the (x^*, y^*, z^*) coordinate system and is similar to the $(1, 2, 3)$ coordinate system used in the Material Models Manual. In this rotated coordinate system, these elements have only one d.o.f. per node (a displacement in axial direction).

The shape functions for these axial displacements are already given by Eq. (4.8). Using index notation, the axial displacement can now be defined as:

$$\mathbf{u}_x^* = N_i v_{ix}^* \quad (4.11)$$

where v_{ix}^* denotes the nodal displacement in axial direction of node i . The nodal displacements in the rotated coordinate system can be rotated to give the nodal displacements in the global coordinate system:

$$\underline{v}_i = \underline{\underline{R}}_{\phi}^T v_{ix}^* \quad (4.12)$$

where $\underline{v}_i = (v_{ix} \ v_{iy} \ v_{iz})^T$ denotes the nodal displacement vector in the global coordinate system and $\underline{\underline{R}}_{\phi}$ denotes the rotation matrix. For further elaboration into the element stiffness matrix see Section 4.2.4 and 4.2.5.

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 elements in the sense that they have six degrees of freedom per node instead of three in the global coordinate system, 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). As the beam elements cannot sustain torsional moments, the beam elements have only 5 d.o.f.s per node in the rotated coordinate system, i.e.

- one axial displacement (u_x^*);
- two transverse displacements (u_y^* and u_z^*);
- two rotations (ϕ_y^* and ϕ_z^*).

The rotated coordinate system is denoted as the (x^*, y^*, z^*) coordinate system and is similar to the $(1, 2, 3)$ coordinate system used in the Material Models Manual. The beam elements are directly integrated over their cross section and numerically integrated along their length using 4-point Gaussian integration according to Table 4.1. Beam elements have only one local coordinate (ξ).

The element provides a quadratic interpolation of the axial displacement (see Eq. 4.9). Using index notation, the axial displacement can now be defined as:

$$\mathbf{u}_x^* = N_i v_{ix}^* \quad (4.13)$$

where v_{ix}^* denotes the nodal displacement in axial direction of node i .

The shape functions for transverse displacements N_{iu} have the property that the function value is equal to unity at node i and zero at the other nodes whereas the derivative of the function is equal to zero at all nodes. The shape functions for transverse displacements $N_{i\phi}$ have the property that the derivative of the function is equal to unity at node i and zero at the other nodes whereas the function value is equal to zero at all nodes. The element provides a fifth-order interpolation of transverse displacements. The shape function can be written as (see the local node numbering as shown in Figure 4.2):

$$\begin{aligned}
 N_{1u} &= (4\xi^2 - 5\xi^3 - 2\xi^4 + 3\xi^5) / 4 & (4.14) \\
 N_{2u} &= 1 - 2\xi^2 + \xi^4 \\
 N_{3u} &= (4\xi^2 + 5\xi^3 - 2\xi^4 - 3\xi^5) / 4 \\
 N_{1\phi} &= (\xi^2 - \xi^3 - \xi^4 + \xi^5) / 4 \\
 N_{2\phi} &= \xi - 2\xi^3 + \xi^5 \\
 N_{3\phi} &= (-\xi^2 - \xi^3 + \xi^4 + \xi^5) / 4
 \end{aligned}$$

Using index notation, the transverse displacements can now be defined as:

$$\underline{u}_{i\phi}^* = \underline{\underline{N}}_{i\phi}^* \underline{v}_{i\phi}^* \tag{4.15}$$

where

$$\underline{u}_{i\phi}^* = \begin{bmatrix} u_y^* & u_z^* \end{bmatrix}^T \tag{4.16}$$

The matrix $\underline{\underline{N}}_{i\phi}^*$ for transverse displacements is defined as:

$$\underline{\underline{N}}_{i\phi}^* = \begin{bmatrix} N_{iu}(\xi) & 0 & 0 & N_{iu}(\xi) \\ 0 & N_{i\phi}(\xi) & N_{i\phi}(\xi) & 0 \end{bmatrix} \tag{4.17}$$

The local nodal transverse displacements and rotations of node i are given by $\underline{v}_{i\phi}^*$:

$$\underline{v}_{i\phi}^* = \begin{bmatrix} v_{iy}^* & v_{iz}^* & \phi_{iy}^* & \phi_{iz}^* \end{bmatrix}^T \tag{4.18}$$

where v_{iy}^* and v_{iz}^* denote the nodal transverse displacements and ϕ_{iy}^* and ϕ_{iz}^* denote the nodal rotations.

The nodal displacements in the rotated coordinate system can be rotated back to give the nodal displacements in the global coordinate system (see Eq. 4.12). For further elaboration into the element stiffness matrix see Section 4.2.4 and 4.2.5.

4.2.4 DERIVATIVES OF INTERPOLATION FUNCTIONS

Node-to-node anchors

As node-to-node anchors can only sustain axial forces, only the axial strains are of interest: $\varepsilon^* = du^*/dx^*$. Using the chain rule for differentiation gives:

$$\varepsilon^* = \frac{du^*}{dx^*} = \frac{du^*}{d\xi} \frac{d\xi}{dx^*} \quad (4.19)$$

where (using index notation)

$$\frac{du^*}{d\xi} = \frac{dN_i}{d\xi} v_i^* \quad (4.20)$$

and

$$\frac{dx^*}{d\xi} = \frac{dN_i}{d\xi} x_i^* \quad (4.21)$$

The parameter x_i^* denotes the coordinate of the nodes in the rotated coordinate system. In case of 2-node line elements Eq. (4.21) can be simplified to:

$$\frac{dx^*}{d\xi} = \frac{L}{2} \quad (4.22)$$

where L denotes the length of the element in the global coordinate system. Inserting Eqs. (4.22), (4.20) and (4.13) into Eq. (4.19) will give:

$$\varepsilon^* = B_i^* v_{ix}^* \quad (4.23)$$

where the rotated strain interpolation function B_i^* is given by:

$$B_i^* = \frac{2}{L} \frac{dN_i}{d\xi} \quad (4.24)$$

Rotating the local nodal displacements back to the global coordinate system gives:

$$B_i = \frac{2}{L} \frac{dN_i}{d\xi} R_{i\theta} \quad (4.25)$$

Note that this strain interpolation function is still a function of the local coordinate ξ as the shape functions N_i are a function of ξ .

Beam elements

In case of axial displacements in the rotated coordinate system, the strain interpolation matrix for beams can be derived from Eq. (4.19) till (4.21). As node 2 of the beam element is located in the middle of the element by default, Eq (4.21) can be simplified to Eq. (4.22). So, the strain interpolation matrix in the global coordinate system for the longitudinal displacements of beams is given by:

$$B_i = \frac{2}{L} \frac{dN_i}{d\xi} R_{=0} \quad (4.26)$$

In case of bending moments, a curvature interpolation matrix is needed to define the stiffness matrix. The curvature interpolation function describes the kinematic relationship between curvatures and displacements:

$$\underline{\kappa}^* = \begin{bmatrix} \kappa_2 \\ \kappa_3 \end{bmatrix} = \begin{bmatrix} -\frac{d^2 u_z^*}{dx^{*2}} \\ \frac{d^2 u_y^*}{dx^{*2}} \end{bmatrix} = \underline{B}_{=i\phi}^* \underline{v}_{i\phi}^* \quad (4.27)$$

where

$$\underline{B}_{=i\phi}^* = - \begin{bmatrix} 0 & \frac{d^2 N_{iu}}{dx^{*2}} & \frac{d^2 N_{i\phi}}{dx^{*2}} & 0 \\ \frac{d^2 N_{iu}}{dx^{*2}} & 0 & 0 & \frac{d^2 N_{i\phi}}{dx^{*2}} \end{bmatrix} \quad (4.28)$$

and $\underline{v}_{i\phi}^*$ is defined by Eq. (4.18). Using the chain rule for differentiation twice gives:

$$\frac{d^2 N}{dx^{*2}} = \frac{d}{d\xi} \frac{d\xi}{dx^*} \frac{dN}{d\xi} \frac{d\xi}{dx^*} = \frac{d^2 N}{d\xi^2} \left(\frac{d\xi}{dx^*} \right)^2 \quad (4.29)$$

As Eq. (4.22) still holds, Eq. (4.26) can be simplified to:

$$\underline{B}_{=i\phi}^* = -\frac{4}{L^2} \begin{bmatrix} 0 & \frac{d^2 N_{iu}}{d\xi^2} & \frac{d^2 N_{i\phi}}{d\xi^2} & 0 \\ \frac{d^2 N_{iu}}{d\xi^2} & 0 & 0 & \frac{d^2 N_{i\phi}}{d\xi^2} \end{bmatrix} \quad (4.30)$$

Using Eq. (4.12) to rotate local nodal transverse displacements and rotations to global nodal displacements and rotations and inserting this equation in Eq. (4.30) gives:

$$\underline{\underline{B}}_{i\phi} = -\frac{4}{L^2} \begin{bmatrix} 0 & \frac{d^2 N_{iu}}{d\xi^2} & \frac{d^2 N_{i\phi}}{d\xi^2} & 0 \\ \frac{d^2 N_{iu}}{d\xi^2} & 0 & 0 & \frac{d^2 N_{i\phi}}{d\xi^2} \end{bmatrix} \underline{\underline{R}}_{\theta} \quad (4.31)$$

Note that this strain interpolation function is still a function of the local coordinate ξ as the shape functions N_i are a function of ξ .

4.2.5 CALCULATION OF ELEMENT STIFFNESS MATRIX

Node-to-node anchors

The element stiffness matrix of a node-to-node anchor is calculated by the integral (see also Eq. 2.25):

$$\underline{\underline{K}}^e = \int \underline{\underline{B}}^T D^a \underline{\underline{B}} dV \quad (4.32)$$

where D^a denotes the elastic constitutive relationship of the node-to-node anchor as discussed in the Material Models Manual. As the strain interpolation matrix is still a function of the local coordinate ξ it will make more sense to solve the integral of Eq. (4.32) in the local coordinate system. Applying the change of variables theorem to change the integral to the local coordinate system gives:

$$\underline{\underline{K}}^e = \int \underline{\underline{B}}^T D^a \underline{\underline{B}} \frac{dx^*}{d\xi} dV^* \quad (4.33)$$

In case of a 2-node line element, $dx^*/d\xi = L/2$. This integral is estimated by numerical integration as described in Section 4.2.2. 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:

$$\underline{\underline{K}}_{ij}^e = \sum_k \underline{\underline{B}}_i^T D^a \underline{\underline{B}}_j \frac{dx^*}{d\xi} w_k \quad (4.34)$$

In case of non-linear behaviour of the node-to-node anchor the maximum stiffness will be used in the stiffness matrix whereas the non-linear behaviour is solved for iteratively.

Beam elements

In case of axial forces, the element stiffness matrix is given by Eq. (4.32) till (4.34). In case of bending moments the stiffness matrix of a beam is calculated by the integral:

$$\underline{\underline{K}}^e = \int_{\underline{\underline{\phi}}} \underline{\underline{B}}^T \underline{\underline{D}}^b \underline{\underline{B}} dV \quad (4.35)$$

where $\underline{\underline{D}}^b$ denotes the constitutive relationship of a beam in bending (see Material Models Manual):

$$\underline{\underline{D}}^b = \begin{bmatrix} EI_2 & EI_{23} \\ EI_{23} & EI_3 \end{bmatrix} \quad (4.36)$$

To solve the integral of Eq. (4.33) in the local coordinate system, the change of variables theorem should be applied:

$$\underline{\underline{K}}^e = \int_{\underline{\underline{\phi}}} \underline{\underline{B}}^T \underline{\underline{D}}^b \underline{\underline{B}} \frac{dx^*}{d\xi} dV^* \quad (4.37)$$

In PLAXIS 3-node beam elements, $dx^*/d\xi = L/2$. This integral is estimated by numerical integration as described in Section 4.2.2. 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:

$$\underline{\underline{K}}_{ij}^e = \sum_k \underline{\underline{B}}_{i\phi}^{*T} \underline{\underline{D}}^b \underline{\underline{B}}_{j\phi}^* \frac{dx^*}{d\xi} w_k \quad (4.38)$$

In case of non-linear behaviour of the beam the maximum stiffness will be used in the stiffness matrix whereas the non-linear behaviour is solved for iteratively.

4.3 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 are described in the following subsections.

4.3.1 INTERPOLATION FUNCTIONS OF AREA ELEMENTS

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 N_i have the property that the function value is equal to unity at node i and zero at the other nodes. The shape functions can be written as (see the local node numbering as shown in Figure 4.3):

$$\begin{aligned}
 N_1 &= \zeta(2\zeta - 1) \\
 N_2 &= \xi(2\xi - 1) \\
 N_3 &= \eta(2\eta - 1) \\
 N_4 &= 4\zeta\xi \\
 N_5 &= 4\xi\eta \\
 N_6 &= 4\eta\zeta
 \end{aligned}
 \tag{4.39}$$

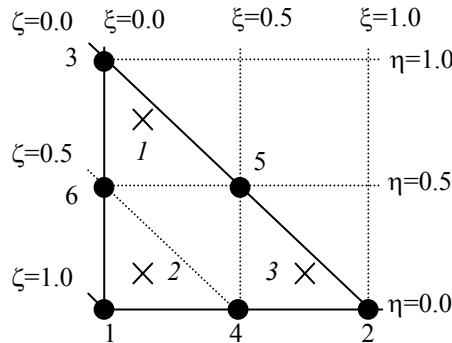


Figure 4.3 Local numbering and positioning of nodes (●) and integration points (x) of a 6-node triangular element

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, distributed loads on a vertical plane 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 N_i have the property that the function value is equal to unity at node i and zero at the other nodes. The shape functions of 8-node elements can be written as (see the local node numbering as shown in Figure 4.4):

$$\begin{aligned}
 N_1 &= (1 - \xi)(1 - \eta)(-1 - \xi - \eta) / 4 \\
 N_2 &= (1 + \xi)(1 - \eta)(-1 + \xi - \eta) / 4 \\
 N_3 &= (1 + \xi)(1 + \eta)(-1 + \xi + \eta) / 4 \\
 N_4 &= (1 - \xi)(1 + \eta)(-1 - \xi + \eta) / 4
 \end{aligned}
 \tag{4.40}$$

$$\begin{aligned}
 N_5 &= (1-\xi)(1+\xi)(1-\eta)/2 \\
 N_6 &= (1-\xi)(1+\xi)(1+\eta)/2 \\
 N_7 &= (1-\eta)(1+\eta)(1+\xi)/2 \\
 N_8 &= (1-\eta)(1+\eta)(1-\xi)/2
 \end{aligned}$$

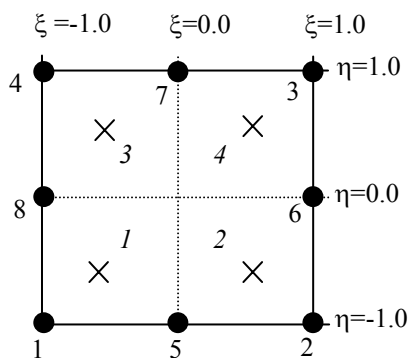


Figure 4.4 Local numbering and positioning of nodes (●) and integration points (x) of an 8-node quadrilateral element

4.3.2 NUMERICAL INTEGRATION OF AREA ELEMENTS

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

$$\iint F(\xi, \eta) d\xi d\eta \approx \sum_{i=1}^k F(\xi_i, \eta_i) w_i \tag{4.41}$$

The PLAXIS 3D FOUNDATION program uses Gaussian integration within the area elements.

6-node triangular elements

For 6-node triangular elements the integration is based on 3 sample points (see Figure 4.3). 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

8-node triangular elements

For 8-node quadrilateral elements the numerical integration is based on 4 (2x2) Gauss points (see Figure 4.4), 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

4.3.3 STRUCTURAL ELEMENTS

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

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.

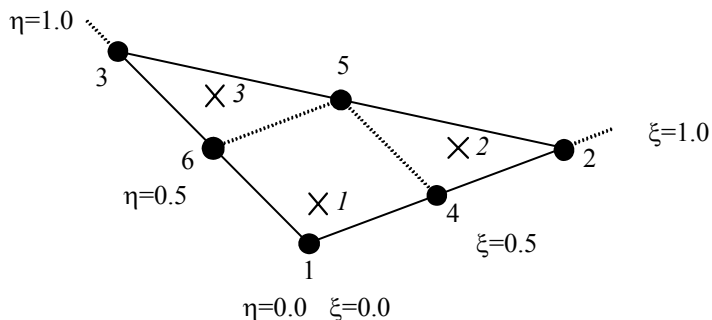


Figure 4.5 Local numbering and positioning of nodes (●) and integration points (x) of a 6-node plate triangle.

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 4.6 and corresponds with Table 4.3.

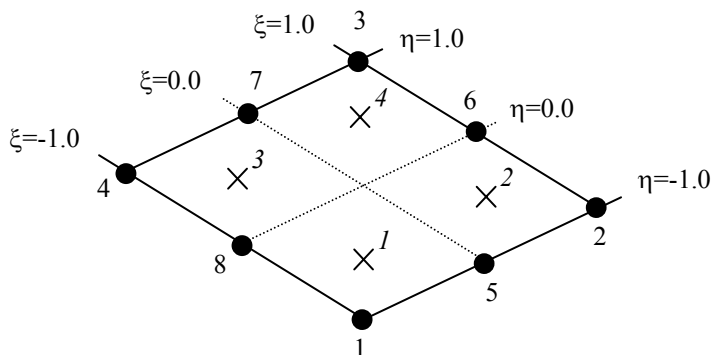


Figure 4.6 Local numbering and positioning of nodes (●) and integration points (x) of an 8-node plate element

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 4.7 (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).

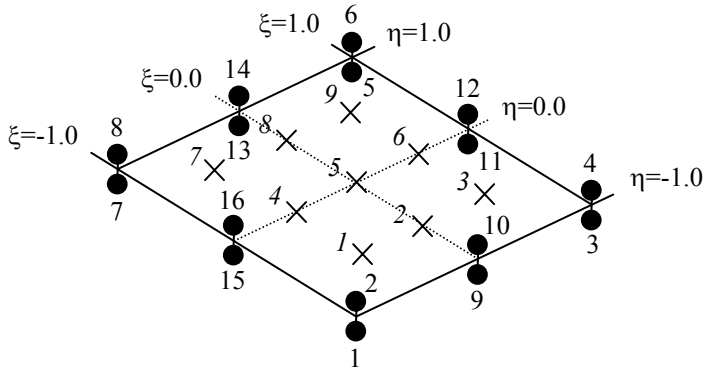


Figure 4.7 Local numbering and positioning of nodes (●) and integration points (x) of a 16-node interface element

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

4.4 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.

4.4.1 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 N_i have the property that the function value is equal to unity at node i and zero at the other nodes. The shape functions of these 15-node volume elements can be written as (see the local node numbering as shown in the Figure 4.8):

$$\begin{aligned}
 N_1 &= -(1-\xi-\eta)(1-\zeta)(+2\xi+2\eta+\zeta)/2 & (4.42) \\
 N_2 &= -\xi(1-\zeta)(2-2\xi+\zeta)/2 \\
 N_3 &= -\eta(1-\zeta)(2-2\eta+\zeta)/2 \\
 N_4 &= -(1-\xi-\eta)(1+\zeta)(+2\xi+2\eta-\zeta)/2 \\
 N_5 &= -\xi(1+\zeta)(2-2\xi-\zeta)/2 \\
 N_6 &= -\eta(1+\zeta)(2-2\eta-\zeta)/2 \\
 N_7 &= (1-\xi-\eta)\xi(1-\zeta)*2 \\
 N_8 &= \xi\eta(1-\zeta)*2 \\
 N_9 &= \eta(1-\xi-\eta)(1-\zeta)*2 \\
 N_{10} &= (1-\xi-\eta)(1-\zeta)(1+\zeta) \\
 N_{11} &= \xi(1-\zeta)(1+\zeta) \\
 N_{12} &= \eta(1-\zeta)(1+\zeta) \\
 N_{13} &= (1-\xi-\eta)\xi(1+\zeta)*2 \\
 N_{14} &= \xi\eta(1+\zeta)*2 \\
 N_{15} &= \eta(1-\xi-\eta)(1+\zeta)*2
 \end{aligned}$$

The soil elements have three degrees of freedom per node: u_x , u_y and u_z . The shape function matrix $\underline{\underline{N}}_i$ can now be defined as:

$$\underline{\underline{N}}_i = \begin{bmatrix} N_i & 0 & 0 \\ 0 & N_i & 0 \\ 0 & 0 & N_i \end{bmatrix} \quad (4.43)$$

and the nodal displacement vector \underline{v} is defined as:

$$\underline{v} = [v_{ix} \quad v_{iy} \quad v_{iz}]^T \quad (4.44)$$

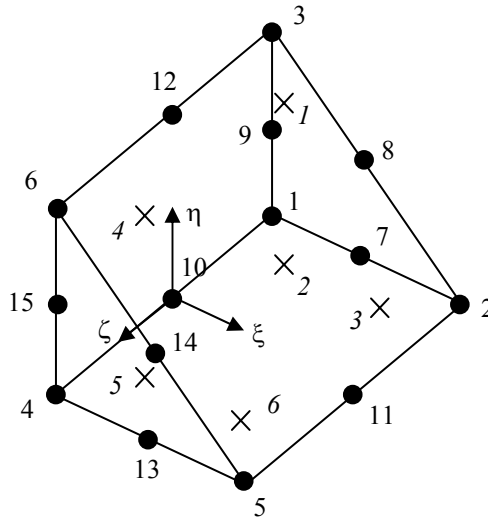


Figure 4.8 Local numbering and positioning of nodes (●) and integration points (x) of a 15-node wedge element

For one fold degenerated wedge elements, a 15 node wedge element is used in which the three nodes along one side coincide (Figure 4.9). For two fold degenerated wedge elements, a 15 node wedge element is used in which the nodes, five in total, along two sides coincide.

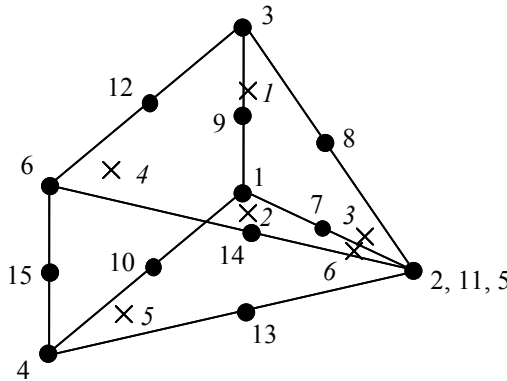


Figure 4.9 Local numbering and positioning of nodes (●) and integration points (x) of a one fold degenerated (reduced) 15-node wedge element

4.4.2 NUMERICAL INTEGRATION OVER VOLUMES

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

$$\iiint F(\xi, \eta, \zeta) d\xi d\eta d\zeta \approx \sum_{i=1}^k F(\xi_i, \eta_i, \zeta_i) w_i \quad (4.45)$$

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 4.8 and Figure 4.9 for the local numbering of integration points. Note that the sum of the weight factors is equal to 1.

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√3	1/6
2	1/6	1/6	-1/3√3	1/6
3	2/3	1/6	-1/3√3	1/6
4	1/6	2/3	+1/3√3	1/6
5	1/6	1/6	+1/3√3	1/6
6	2/3	1/6	+1/3√3	1/6

4.4.3 DERIVATIVES OF INTERPOLATION 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).

$$\underline{\underline{\varepsilon}} = \underline{\underline{B}}_i v_i \quad (4.46)$$

where

$$\underline{\underline{B}}_i = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} & 0 & \frac{\partial N_i}{\partial x} \end{bmatrix} \quad (4.47)$$

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 :

$$\begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{bmatrix} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix} \begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} = J \begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} \quad (4.48)$$

Or inversely:

$$\begin{bmatrix} \frac{\partial N_i}{\partial x} \\ \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} \end{bmatrix} = J^{-1} \begin{bmatrix} \frac{\partial N_i}{\partial \xi} \\ \frac{\partial N_i}{\partial \eta} \\ \frac{\partial N_i}{\partial \zeta} \end{bmatrix} \quad (4.49)$$

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:

$$\begin{bmatrix} \epsilon_{xx} \\ \epsilon_{yy} \\ \epsilon_{zz} \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} = \sum_i B_i \begin{bmatrix} v_{ix} \\ v_{iy} \\ v_{iz} \end{bmatrix} \quad (4.50)$$

where v_i are the displacement components in node i .

4.4.4 CALCULATION OF ELEMENT STIFFNESS MATRIX

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

$$\underline{\underline{K}}^e = \int \underline{\underline{B}}^T \underline{\underline{D}}^e \underline{\underline{B}} dV \quad (4.51)$$

As it is more convenient to calculate the element stiffness matrix in the local coordinate system, the change of variables theorem should be applied to change the integral to the local coordinate system:

$$\underline{\underline{K}}^e = \int \underline{\underline{B}}^T \underline{\underline{D}}^e \underline{\underline{B}} j dV^* \quad (4.52)$$

where j denotes the determinant of the Jacobian. The integral is estimated by numerical integration as described in Section 4.4.2. 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:

$$\underline{\underline{K}}_{ij}^e = \sum_k \underline{\underline{B}}_i^T \underline{\underline{D}}^e \underline{\underline{B}}_j j w_k \quad (4.53)$$

In case of plastic deformations of the soil only the elastic part of the soil stiffness will be used in the stiffness matrix whereas the plasticity is solved for iteratively.

4.5 SPECIAL ELEMENTS

As special elements in PLAXIS embedded piles and ground anchors will be considered. Embedded piles and ground anchors are based on the embedded beam approach (Sadek & Shahrour, 2004). Embedded piles consist of beam elements to model the pile itself and embedded interface elements to model the interaction between the soil and the pile at the pile skin as well as at the pile foot. A ground anchor consists of a node-to-node anchor together with an embedded pile to simulate the grout body.

4.5.1 EMBEDDED PILES

The embedded pile has been developed to describe the interaction of a pile with its surrounding soil. The interaction at the pile skin and at the pile foot is described by means of embedded interface elements. The pile is considered as a beam which can cross a 15-node wedge element at any place with any arbitrary orientation (Figure 4.10). Due to the existence of the beam element three extra nodes are introduced inside the 15-node wedge element.

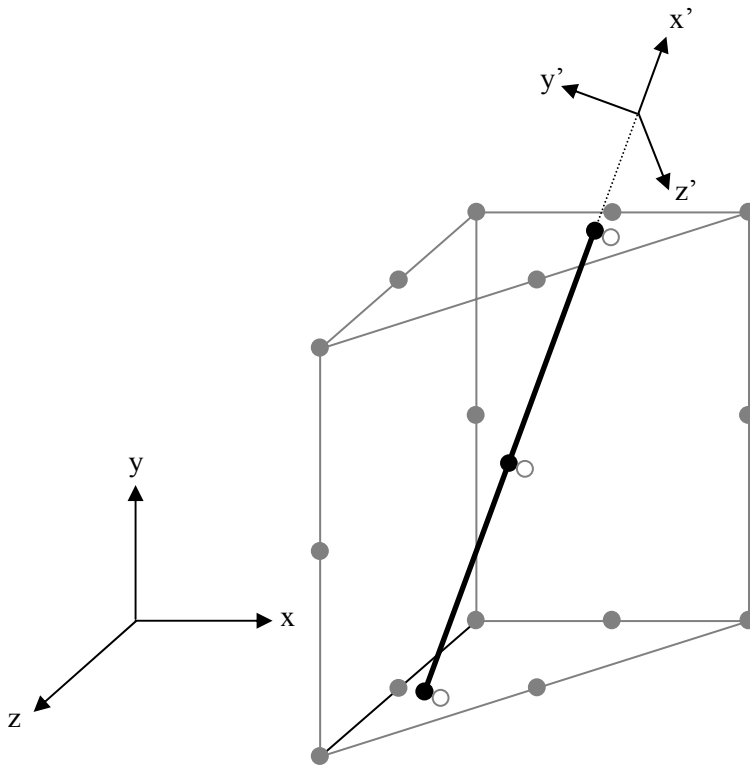


Figure 4.10 Illustration of the embedded beam element denoted by the solid line. The blank grey circles denote the virtual nodes of the soil element.

Finite element discretisation

The finite element discretisation of the pile is similar to beam elements, as discussed in Section 4.2.3. The finite element discretisation of the interaction with the soil will be discussed in this chapter. Using the standard notation the displacement of the soil \underline{u}_s and the displacement of the pile \underline{u}_p can be discretised as:

$$\underline{u}_s = \underline{\underline{N}}_s \underline{v}_s \quad \underline{u}_p = \underline{\underline{N}}_p \underline{v}_p \quad (4.54)$$

where $\underline{\underline{N}}_s$ and $\underline{\underline{N}}_p$ are the matrices containing the interpolation functions of the soil elements and the beam elements respectively (see Section 4.4.1 and 4.2.3) and \underline{v}_s and \underline{v}_p are the nodal displacement vectors of the soil elements and the beam elements respectively.

Interaction at the skin

First, the interaction between the soil and the pile at the skin of the pile will be described by embedded interface elements. These interface elements are based on 3-node line elements with pairs of nodes instead of single nodes. One node of each pair belongs to the beam element, whereas the other (virtual) node is a point in the 15-node wedge element (Figure 4.10). The interaction can be represented by a skin traction \underline{t}^{skin} . The development of the skin traction can be regarded as an incremental process:

$$\underline{t}^{skin} = \underline{t}_0^{skin} + \Delta \underline{t}^{skin} \quad (4.55)$$

In this equation \underline{t}_0^{skin} denotes the initial skin traction and $\Delta \underline{t}^{skin}$ denotes the skin traction increment. The constitutive relation between the skin traction increment and the relative displacement increment is formulated as:

$$\Delta \underline{t}^{skin} = \underline{\underline{T}}^{skin} \Delta \underline{u}_{rel} \quad (4.56)$$

In this relation $\underline{\underline{T}}^{skin}$ denotes the material stiffness of the embedded interface element in the global coordinate system. The increment in the relative displacement vector $\Delta \underline{u}_{rel}$ is defined as the difference in the increment of the soil displacement and the increment of the pile displacement:

$$\Delta \underline{u}_{rel} = \Delta \underline{u}_p - \Delta \underline{u}_s = \underline{\underline{N}}_p \Delta \underline{v}_p - \underline{\underline{N}}_s \Delta \underline{v}_s = \underline{\underline{N}}_{rel} \Delta \underline{v}_{rel} \quad (4.57)$$

where

$$\underline{\underline{N}}_{rel} = \begin{bmatrix} \underline{\underline{N}}_p & -\underline{\underline{N}}_s \end{bmatrix} \quad (4.58)$$

and

$$\Delta \underline{v}_{rel} = \begin{bmatrix} \Delta \underline{v}_p \\ \Delta \underline{v}_s \end{bmatrix} \quad (4.59)$$

Looking to the virtual work equation (2.6) the traction increment at the pile skin can be discretised as:

$$\int \delta \underline{u}_{rel}^T \Delta \underline{t}^{skin} dS = \delta \underline{v}_{rel}^T \int \underline{\underline{N}}_{rel}^T \underline{\underline{T}}^{skin} \underline{\underline{N}}_{rel} dS \Delta \underline{v}_{rel} = \underline{\underline{K}}^{skin} \Delta \underline{v}_{rel} \quad (4.60)$$

In this formulation the element stiffness matrix $\underline{\underline{K}}^{skin}$ represents the interaction between the pile and the soil at the skin and consists of four parts:

$$\underline{\underline{K}}^{skin} = \begin{bmatrix} \underline{\underline{K}}_{pp}^{skin} & \underline{\underline{K}}_{ps}^{skin} \\ \underline{\underline{K}}_{sp}^{skin} & \underline{\underline{K}}_{ss}^{skin} \end{bmatrix} \quad (4.61)$$

The matrix $\underline{\underline{K}}_{pp}^{skin}$ represents the contribution of the pile nodes to the interaction, the matrix $\underline{\underline{K}}_{ss}^{skin}$ represents the contribution of the soil nodes to the interaction and the matrices $\underline{\underline{K}}_{ps}^{skin}$ and $\underline{\underline{K}}_{sp}^{skin}$ are the mixed terms:

$$\underline{\underline{K}}_{pp}^{skin} = \int \underline{\underline{N}}_p^T T^{skin} \underline{\underline{N}}_p dS \quad (4.62a)$$

$$\underline{\underline{K}}_{ps}^{skin} = -\int \underline{\underline{N}}_p^T T^{skin} \underline{\underline{N}}_s dS \quad (4.62b)$$

$$\underline{\underline{K}}_{sp}^{skin} = -\int \underline{\underline{N}}_s^T T^{skin} \underline{\underline{N}}_p dS \quad (4.62c)$$

$$\underline{\underline{K}}_{ss}^{skin} = \int \underline{\underline{N}}_s^T T^{skin} \underline{\underline{N}}_s dS \quad (4.62d)$$

These integrals are numerically estimated using Eq. (4.10):

$$\int_{\xi=-1}^1 F(\xi) d\xi \approx \sum_{i=1}^k F(\xi_i) w_i \quad (4.63)$$

However, instead of Gauss integration Newton-Cotes integration is used. In this method the points ξ_i are chosen at the position of the nodes, see Table 4.6. The type of integration used for the embedded interface elements is shaded. In case of plastic deformations of the embedded interface elements only the elastic part of the interface stiffness will be used in the stiffness matrix whereas the plasticity is solved for iteratively.

Table 4.6 Newton-Cotes integration

	ξ_i	w_i
2 nodes	± 1	1
3 nodes	$\pm 1, 0$	1/3, 4/3
4 nodes	$\pm 1, \pm 1/3$	1/4, 3/4
5 nodes	$\pm 1, \pm 1/2, 0$	7/45, 32/45, 12/45

Interaction at the foot

The interaction of the embedded pile at the foot is described by an embedded interface element. This interaction can be represented by a foot force vector \underline{f}^{foot} . Like the development of the skin traction the development of the foot force is an incremental process:

$$\underline{f}^{foot} = \underline{f}_0^{foot} + \Delta \underline{f}^{foot} \quad (4.64)$$

In this equation \underline{f}_0^{foot} denotes the initial force and $\Delta \underline{f}^{foot}$ denotes the force increment at the foot. The constitutive relation between the force increment at the foot and the relative displacement increment can be described by:

$$\Delta \underline{f}^{foot} = \underline{\underline{D}}^{foot} \Delta \underline{u}_{rel} \quad (4.65)$$

The symbol $\underline{\underline{D}}^{foot}$ denotes the material stiffness matrix of the spring element at the foot of the embedded pile in the global coordinate system. As for the skin interaction the force increment at the foot of the pile can be discretised by means of the virtual work (Eq. 2.6), as:

$$\delta \underline{u}_{rel}^T \Delta \underline{f}^{foot} = \delta \underline{v}_{rel}^T \underline{\underline{N}}_{rel}^T \underline{\underline{D}}^{foot} \underline{\underline{N}}_{rel} \Delta \underline{v}_{rel} = \underline{\underline{K}}^{foot} \Delta \underline{v}_{rel} \quad (4.66)$$

The stiffness matrix at the foot is represented by $\underline{\underline{K}}^{foot}$ and consists of four parts:

$$\underline{\underline{K}}^{foot} = \begin{bmatrix} \underline{\underline{K}}_{pp}^{foot} & \underline{\underline{K}}_{ps}^{foot} \\ \underline{\underline{K}}_{sp}^{foot} & \underline{\underline{K}}_{ss}^{foot} \end{bmatrix} \quad (4.67)$$

In this equation $\underline{\underline{K}}_{pp}^{foot}$ represents the contribution of the pile nodes, $\underline{\underline{K}}_{ss}^{foot}$ represents the contribution from the soil nodes and $\underline{\underline{K}}_{ps}^{foot}$ and $\underline{\underline{K}}_{sp}^{foot}$ are the mixed terms:

$$\underline{\underline{K}}_{pp}^{foot} = \underline{\underline{N}}_p^T \underline{\underline{D}}^{foot} \underline{\underline{N}}_p \quad (4.68a)$$

$$\underline{\underline{K}}_{ps}^{foot} = -\underline{\underline{N}}_p^T \underline{\underline{D}}^{foot} \underline{\underline{N}}_s \quad (4.68b)$$

$$\underline{\underline{K}}_{sp}^{foot} = -\underline{\underline{N}}_s^T \underline{\underline{D}}^{foot} \underline{\underline{N}}_p \quad (4.68c)$$

$$\underline{\underline{K}}_{ss}^{foot} = \underline{\underline{N}}_s^T \underline{\underline{D}}^{foot} \underline{\underline{N}}_s \quad (4.68d)$$

In case of plastic deformations of the embedded interface element only the elastic part of the interface stiffness will be used in the stiffness matrix whereas the plasticity is solved for iteratively.

4.5.2 GROUND ANCHOR

The ground anchor model consists of a node-to-node anchor in combination with an embedded pile to model the grout body (Figure 4.11). This embedded pile has no foot resistance. The discretisation of the node-to-node anchor is described in Section 4.2. The discretisation of the embedded pile is described in Section 4.5.1.

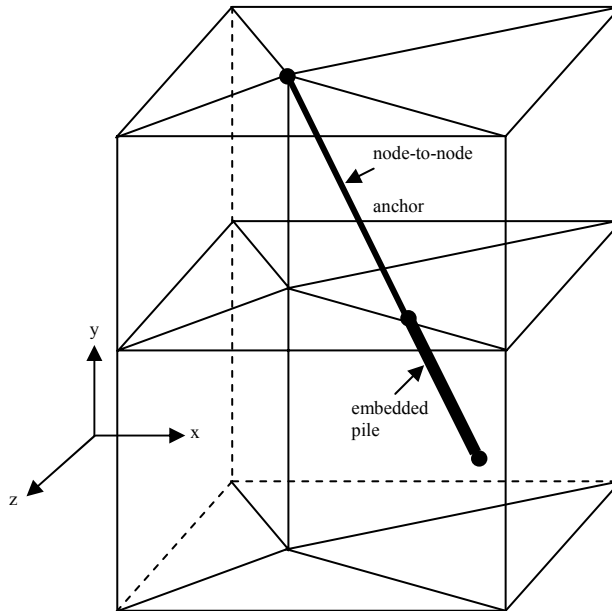


Figure 4.11 Illustration of a ground anchor.

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] Sadek M., Shahrour I. (2004), A three dimensional embedded beam element for reinforced geomaterials. *International Journal for Numerical and Analytical Methods in Geomechanics*, Vol. 28, pp. 931-946.
- [5] Van Langen H. (1991), *Numerical analysis of soil-structure interaction*. Dissertation. Delft University of Technology.
- [6] 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.
- [7] Vermeer P.A., De Borst R. (1984), Non-associated plasticity for soils, concrete and rock. *Heron*, Vol. 29, No. 3.
- [8] Vermeer P.A., Van Langen H. (1989), Soil collapse computations with finite elements. *Ingenieur-Archiv* 59, pp. 221-236.
- [9] 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.
- [10] 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

$$\underline{f}_{-in} = \int \underline{\underline{B}}^T \underline{\underline{\sigma}}_c^{i-1} dV$$

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

$$\delta \underline{v} = \underline{\underline{K}}^{-1} \Delta \underline{f} \quad *$$

Update displacement increments

$$\Delta \underline{v}^j = \Delta \underline{v}^{j-1} + \delta \underline{v}$$

Calculate strain increments

$$\Delta \underline{\underline{\varepsilon}} = \underline{\underline{B}} \Delta \underline{v} ; \quad \delta \underline{\underline{\varepsilon}} = \underline{\underline{B}} \delta \underline{v}$$

Calculate stresses: Elastic

$$\underline{\underline{\sigma}}^{tr} = \underline{\underline{\sigma}}_c^{i-1} + \underline{\underline{D}}^e \Delta \underline{\underline{\varepsilon}}$$

Equilibrium

$$\underline{\underline{\sigma}}^{eq} = \underline{\underline{\sigma}}_c^{i,j-1} + \underline{\underline{D}}^e \delta \underline{\underline{\varepsilon}}$$

Constitutive

$$\underline{\underline{\sigma}}_c^{i,j} = \underline{\underline{\sigma}}^{tr} - \frac{\langle f(\underline{\underline{\sigma}}^{tr}) \rangle}{d} \underline{\underline{D}}^e \frac{\partial \underline{g}}{\partial \underline{\underline{\sigma}}}$$

Form reaction vector

$$\underline{f}_{-in} = \int \underline{\underline{B}}^T \underline{\underline{\sigma}}_c^{i,j} dV$$

Calculate unbalance

$$\Delta \underline{f} = \underline{f}_{-ex}^i - \underline{f}_{-in}$$

Calculate error

$$e = \frac{|\Delta \underline{f}|}{|\underline{f}_{-ex}^i|}$$

Accuracy check

if $e > e_{tolerated} \rightarrow$ new iteration

Update displacements

$$\underline{v}^j = \underline{v}^{i-1} + \Delta \underline{v}$$

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-QMR¹. 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{b}	:	Vector containing the body force
\underline{B}	:	Strain interpolation matrix
\underline{D}^e	:	Elastic material stiffness matrix representing Hooke's law
f	:	Yield function
g	:	Plastic potential function
h	:	Hardening parameter
\underline{K}	:	Stiffness matrix
\underline{L}	:	Differential operator
\underline{M}	:	Material stiffness matrix
\underline{N}	:	Matrix with shape functions
p	:	(Excess) pore pressure
\underline{p}_n	:	Vector with nodal excess pore pressures
\underline{q}_n	:	Vector due to prescribed outflow at the boundary
\underline{R}	:	Permeability matrix
\underline{R}_ϕ	:	Rotation matrix
t	:	Time
\underline{t}	:	Boundary tractions
\underline{u}	:	Vector with displacement components
\underline{v}	:	Vector with nodal displacements
V	:	Volume
w	:	Weight factor
x^*, y^*, z^*	:	Coordinates in the rotated coordinate system (similar to $(1, 2, 3)$ coordinate system)
γ	:	Volumetric weight
$\underline{\varepsilon}$:	Vector with strain components
λ	:	Plastic multiplier
ξ, η, ζ	:	Local coordinates
$\underline{\sigma}$:	Vector with stress components
ω	:	Integration constant (explicit: $\omega = 0$; implicit: $\omega = 1$)

