M.Sc. in Geological Engineering: Subject No. 081803B02

# Numerical Analysis in Geotechnical Engineering

Hong-Hu ZHU

School of Earth Sciences and Engineering, Nanjing University

南京大學 NANJING UNIVERSITY

## Lecture 3: Geotechnical considerations

**3.1 Synopsis and 3.2 Introduction**
**3.3 Total stress analysis**
**3.4 Pore pressure calculation**
**3.5 Finite elements to model structural components**
**3.6 Finite elements to model interfaces**
**3.7 Boundary conditions**
**3.8 Summary**

南京大學 NANJING UNIVERSITY

## 3.1 Synopsis

- Discusses enhancements of the standard FE for geotechnical analysis
- Undrained analysis, modeling structural and interface

## 3.2 Introduction

- Constitutive relationships in effective stresses
- Interaction problems
- Wider range of boundary conditions
- Excavation and construction

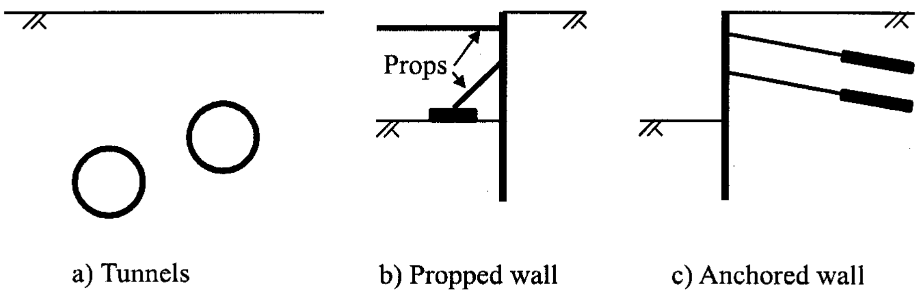
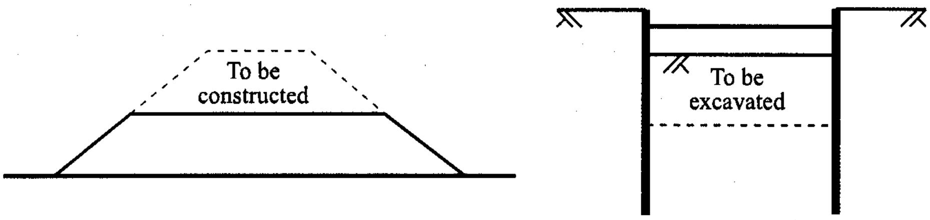



*Figure 3.1: Examples of soil-structure interaction*

if realistic analyses are to be performed. For example, many geotechnical problems involve excavation and construction, see Figure 3.2. Many also involve changes in pore water pressure. Special boundary conditions are often required to model soil structure interaction.

This chapter describes how the above conditions can be accommodated in finite element analyses. Their application in specific geotechnical problems is discussed in Volume 2 of this book.

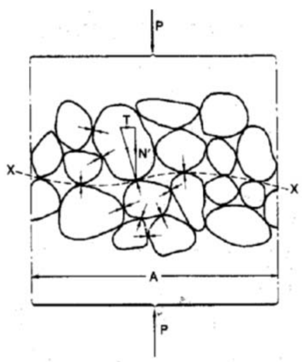



a) Embankment construction　　b) Basement excavation

*Figure 3.2: Examples of construction and excavation*



## Effective stress principle: 有效应力原理




Interpretation of effective stress.

$$\sigma = \frac{P}{A} \qquad P = \Sigma N' + uA$$

$$\sigma' = \frac{\Sigma N'}{A} \qquad \frac{P}{A} = \frac{\Sigma N'}{A} + u$$

Normal stress: $\sigma = \sigma' + u$

Shear stress: no change

**Effective stresses control: both deformation and shear resistance (or shear strength) since they reflect soil particle interaction**

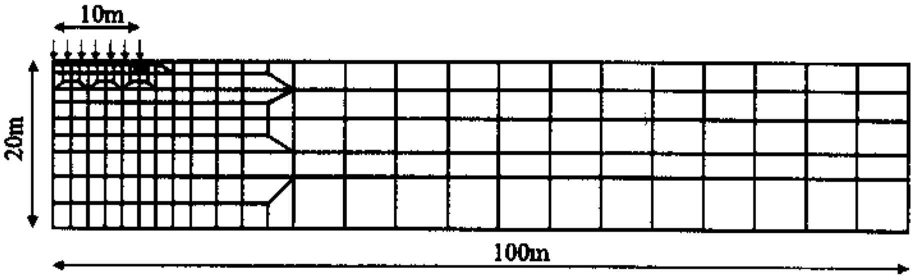
## 3.3 Total stress analysis

$$\{\Delta\sigma\} = [\boldsymbol{D}]\{\Delta\varepsilon\} \qquad (3.1)$$

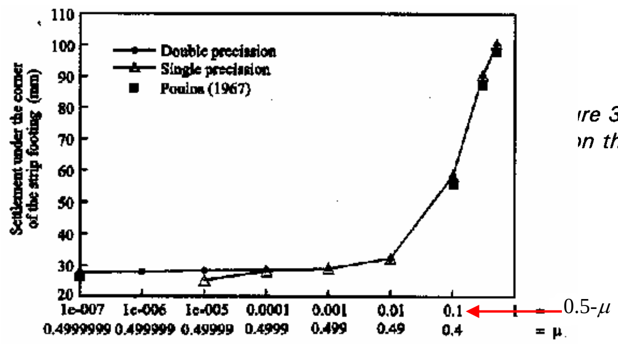
where $\{\Delta\boldsymbol{\sigma}\}=[\Delta\sigma_x, \Delta\sigma_y, \Delta\sigma_z, \Delta\tau_{xy}, \Delta\tau_{xz}, \Delta\tau_{yz}]^T$ and $\{\Delta\boldsymbol{\varepsilon}\}=[\Delta\varepsilon_x, \Delta\varepsilon_y, \Delta\varepsilon_z, \Delta\gamma_{xy}, \Delta\gamma_{xz}, \Delta\gamma_{yz}]^T$ are the incremental total stress and strain vectors and $[\boldsymbol{D}]$ is the assumed relationship between these vectors. For the present considerations, $[\boldsymbol{D}]$ is assumed to be the isotropic linear elastic matrix, given in Section 1.5.5 of Chapter 1.

- Fully drained problems in which there is no change in pore fluid pressure, $\Delta p_f = 0$. This implies that changes in effective and total stress are the same, i.e. $\{\Delta\boldsymbol{\sigma}'\}=\{\Delta\boldsymbol{\sigma}\}$, and that the $[\boldsymbol{D}]$ matrix contains the effective constitutive behaviour. For example, for isotropic linear elastic behaviour $[\boldsymbol{D}]$ will be based on a drained Young's modulus, $E'$, and drained Poisson's ratio, $\mu'$.
- Fully undrained behaviour in which the $[\boldsymbol{D}]$ matrix is expressed in terms of total stress parameters. For isotropic linear elastic behaviour $[\boldsymbol{D}]$ is based on an undrained Young's modulus, $E_u$, and an undrained Poisson's ratio, $\mu_u$.

$\mu_u$=0.499 <0.5 for undrained analysis – there is NO need to separate effective stress and pore water pressure




a) Finite element mesh




b) Settlement under the corner of strip footing vs. Poisson's ratio

ıre 3.3: Effect of Poisson's ratio ın the behaviour of a smooth flexible strip footing

NANJING UNIVERSITY

## 3.4 Pore pressure calculation

- The previous undrained analysis is done using total stress $\Delta\sigma$ and total stress parameters $E_u$ and $\mu_u$.
- The undrained analysis is preferred to be done using effective stress and parameters $E'$ and $\mu'$ :
  (a) Pore water pressure $\Delta p_f$ calculation is needed.
  (b) Constitutive relationships are normally expressed in terms of effective stresses $\Delta\sigma'$

The principle of effective stress is:

$$\{\Delta\sigma\} = \{\Delta\sigma'\} + \{\Delta\sigma_f\} \qquad (3.2)$$

$$\{\Delta\sigma_f\} = \{\Delta p_f \quad \Delta p_f \quad \Delta p_f \quad 0 \quad 0 \quad 0\}^T \qquad (3.3)$$

NANJING UNIVERSITY

$$\{\Delta\sigma'\} = [D']\{\Delta\varepsilon\} \qquad (3.4)$$

$$\{\Delta\sigma_f\} = [D_f]\{\Delta\varepsilon\} \qquad (3.5)$$

Substituting Equations (3.1), (3.4) and (3.5) into Equation (3.2) gives:

$$[D] = [D'] + [D_f] \qquad (3.6)$$

$$[D_f] = K_e\begin{bmatrix} 1_3 & 0_3 \\ 0_3 & 0_3 \end{bmatrix} \qquad (3.7)$$

in which $K_e$ is a constant, $1_3$ is a 3 x 3 matrix all elements of which are 1, and $0_3$ is a 3x3 null matrix. It is shown below how the equivalent bulk modulus of the pore fluid, $K_e$, is related to $K_f$. This follows from Naylor (1974).

In effective stress analysis – there is a need to separate effective stress and pore water pressure (calculated separately –要分开计算)

南京大學 NANJING UNIVERSITY

$n$=void ratio, $V_v$=volume of voids, $V_s$=volume of solids, $V$=total volume. The relationship is:

$$n=\frac{V_v}{V},\ V_v=nV,\ V_s=V-V_v=V(1-\frac{V_v}{V})=V(1-n)$$

$$\Delta\varepsilon_{v,void}=\frac{\Delta V_v}{V_v}=\frac{\Delta p_f}{K_f},\ \Delta V_v=\frac{V_v\Delta p_f}{K_f},\ \ \Delta\varepsilon_{v,solid}=\frac{\Delta V_s}{V_s}=\frac{\Delta p_f}{K_s},\ \Delta V_s=\frac{V_s\Delta p_f}{K_s}$$

$$\Delta\varepsilon_v=\frac{\Delta V_v+\Delta V_s}{V}=\frac{V_v}{V}\frac{\Delta p_f}{K_f}+\frac{V_s}{V}\frac{\Delta p_f}{K_s}=n\frac{\Delta p_f}{K_f}+(1-n)\frac{\Delta p_f}{K_s}$$

$$\Delta\varepsilon_v=\frac{n}{K_f}\Delta p_f+\frac{(1-n)}{K_s}\Delta p_f \qquad (3.8)$$

Substituting Equation (3.7) into (3.5) gives three identical equations:

$$\Delta p_f=K_e(\Delta\varepsilon_x+\Delta\varepsilon_y+\Delta\varepsilon_z)=K_e\Delta\varepsilon_v$$



$$n=\frac{V_v}{V},\ V_v=nV,\ V_s=V-V_v=V(1-\frac{V_v}{V})=V(1-n)$$

$$\Delta\varepsilon_v=\frac{n}{K_f}\Delta p_f+\frac{(1-n)}{K_s}\Delta p_f \qquad (3.8)$$

Substituting Equation (3.7) into (3.5) gives three identical equations:

$$\Delta p_f=K_e(\Delta\varepsilon_x+\Delta\varepsilon_y+\Delta\varepsilon_z)=K_e\Delta\varepsilon_v$$

$$\Delta\varepsilon_v=\frac{\Delta p_f}{K_e} \qquad (3.9)$$

Equating Equations (3.8) and (3.9) and re-arranging gives:

$$K_e=\frac{1}{\frac{n}{K_f}+\frac{(1-n)}{K_s}} \qquad (3.10)$$

$$K_e=\frac{K_f}{n}$$

$$K_e=K_f$$

- In undrained analysis, $K_e$ must be set. For saturated case, results are not sensitive to as long as it is large, but not too large. $K_e=\beta K_{skel}$

($\beta$=100 to 1000, $K_{skel}$=bulk modulus of soil skeleton)

NANJING UNIVERSITY

- Relation between $\mu_u$ and $\mu'$ :

$$\mu_u = \frac{A}{(1+2A)} \text{ , where } A = \frac{(1+\mu')}{(1-2\mu')}\left[\frac{\mu'}{(1+\mu')}+\frac{\beta}{3}\right] \qquad (3.13)$$

Table 3.1 indicates how $\mu_u$ varies with $\beta$ for $\mu'=0.1$ and $\mu'=0.3$.

*Table 3.1: Equivalent values of $\mu_u$*

| $\beta$ | $\mu_u$ | |
|---|---|---|
| | $\mu'$=0.1 | $\mu'$=0.3 |
| 10 | 0.4520 | 0.4793 |
| 100 | 0.4946 | 0.4977 |
| 1000 | 0.4994 | 0.4998 |

NANJING UNIVERSITY

## 3.5 Finite elements to model structural components

- Many geotechnical problems involves soil-structural interaction: retaining walls, props, anchors, etc.
- How to model a structure?
  (a) Using 2-D/3-D continuum elements – structure size is small which needs large number of elements or unacceptable ratio
  (b) Using structural elements – beam, membrane, cable elements for averaged quantity such as bending, axial force, etc.

NANJING UNIVERSITY

- Classic (pure bending) beam element – *E, I, A(A is area)*
- Mindlin/Timoshenko (bending and shearing) beam element – $E$, $G$ ($k$), $\mu$ , $I$, $A$
- Membrane (no bending and tension only) element – $E$, $\mu$, $A$

NANJING UNIVERSITY

## 3.6 Finite elements to model interface

### 3.6.1 Introduction

- Soil-structural interaction – large relative movements, even separation
- Three common interaction modeling methods:
  (a) Using continuum elements – problem with separation and large relative movement (large deformation?)
  (b) Linkage elements using discrete springs
  (c) Using interface elements with zero thickness

NANJING UNIVERSITY

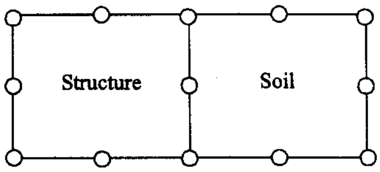



*Figure 3.9: Soil-structure interface using continuum elements*

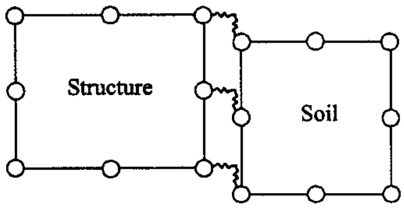



*Figure 3.11: Use of springs to model interface*

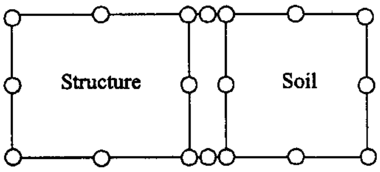



*Figure 3.10: Use of continuum elements to model interface*

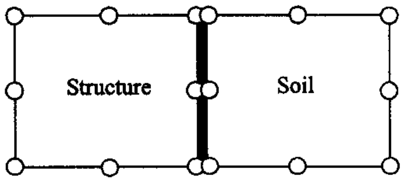
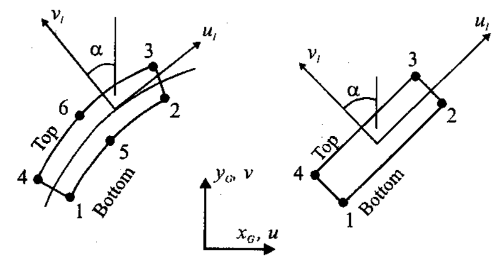



*Figure 3.12: Use of special interface elements*

## 3.6.2 Basic theory

- Isoparametric interface element

$$\begin{Bmatrix} \Delta\tau \\ \Delta\sigma \end{Bmatrix} = [\boldsymbol{D}] \begin{Bmatrix} \Delta\gamma \\ \Delta\varepsilon \end{Bmatrix} \qquad (3.44)$$




a) 6-noded element  b) 4-noded element

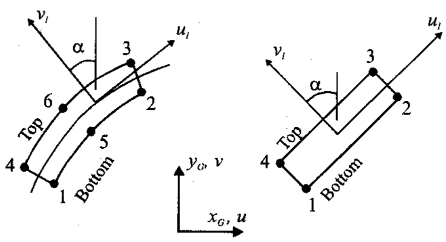
*Figure 3.13: Isoparametric interface elements*

For isotropic linear elastic behaviour the $[\boldsymbol{D}]$ matrix takes the form:

$$[\boldsymbol{D}] = \begin{bmatrix} K_s & 0 \\ 0 & K_n \end{bmatrix} \qquad (3.45)$$

where $K_s$ and $K_n$ are the elastic shear stiffness and normal stiffness respectively.






The interface element strain is defined as the relative displacement of the top and bottom of the interface element:

$$\gamma = \Delta u_l = u_l^{\text{bot}} - u_l^{\text{top}} \qquad (3.46)$$

$$\varepsilon = \Delta v_l = v_l^{\text{bot}} - v_l^{\text{top}} \qquad (3.47)$$

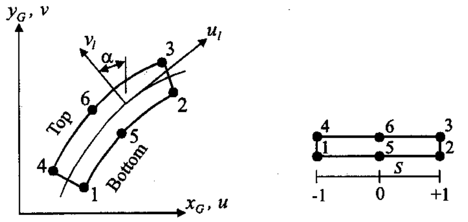
where:

$$\begin{aligned} u_l &= v\sin\alpha + u\cos\alpha \\ v_l &= v\cos\alpha - u\sin\alpha \end{aligned} \qquad (3.48)$$

and $u$ and $v$ are the global displacements in the $x_G$ and $y_G$ directions respectively. Hence:

$$\begin{aligned} \gamma &= (v^{\text{bot}} - v^{\text{top}})\sin\alpha + (u^{\text{bot}} - u^{\text{top}})\cos\alpha \\ \varepsilon &= (v^{\text{bot}} - v^{\text{top}})\cos\alpha - (u^{\text{bot}} - u^{\text{top}})\sin\alpha \end{aligned} \qquad (3.49)$$

## 3.6.3 Finite element formulation




a) Global coordinates   b) Natural ordinate

*Figure 3.14: 6 noded interface element*

Figure 3.14 shows a six noded interface element. The strains are defined as:

$$\begin{Bmatrix} \gamma \\ \varepsilon \end{Bmatrix} = \begin{Bmatrix} u_l^{\text{bot}} - u_l^{\text{top}} \\ v_l^{\text{bot}} - v_l^{\text{top}} \end{Bmatrix} \tag{3.50}$$

The transformation of local to global displacements is written in matrix form as (Figure 3.14):

$$\begin{Bmatrix} u_l \\ v_l \end{Bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix} \tag{3.51}$$



Substitution into Equation (3.50) gives:

$$\begin{Bmatrix} \gamma \\ \varepsilon \end{Bmatrix} = \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \begin{Bmatrix} u^{\text{bot}} - u^{\text{top}} \\ v^{\text{bot}} - v^{\text{top}} \end{Bmatrix} \tag{3.52}$$

The global displacements ($u$, $v$) at any point in the element are expressed in terms of the nodal displacements, using the isoparametric shape functions, $N_i$:

$$\begin{aligned} u^{\text{top}} &= N_3 u_3 + N_4 u_4 + N_6 u_6 \\ u^{\text{bot}} &= N_1 u_1 + N_2 u_2 + N_5 u_5 \end{aligned} \tag{3.53}$$

and

$$\begin{aligned} v^{\text{top}} &= N_3 v_3 + N_4 v_4 + N_6 v_6 \\ v^{\text{bot}} &= N_1 v_1 + N_2 v_2 + N_5 v_5 \end{aligned} \tag{3.54}$$

where the subscript refers to the node number.

The isoparametric shape functions, $N_i$, are defined as:

$$\begin{aligned} N_1 &= N_4 = \tfrac{1}{2} s(s-1) \\ N_2 &= N_5 = \tfrac{1}{2} s(s+1) \\ N_3 &= N_6 = (1-s^2) \end{aligned} \tag{3.55}$$

where $s$ is the natural ordinate that varies from -1 to +1 over the element length

(Figure 3.14). Substitution of Equations (3.53) and (3.54) into (3.52) gives:

$$\begin{Bmatrix} \gamma \\ \varepsilon \end{Bmatrix} = [\boldsymbol{B}]\{\delta\} \tag{3.56}$$

where $\boldsymbol{\delta}$ is the vector of nodal displacements (degrees of freedom) defined as:

$$\{\delta\} = \{ u_1 \ v_1 \ u_2 \ v_2 \ u_3 \ v_3 \ u_4 \ v_4 \ u_5 \ v_5 \ u_6 \ v_6 \}^{\mathrm{T}} \tag{3.57}$$

and

$$[\boldsymbol{B}] = \begin{bmatrix} \cos\alpha & \sin\alpha \\ -\sin\alpha & \cos\alpha \end{bmatrix} \begin{bmatrix} N_1 & 0 & N_2 & 0 & -N_3 & 0 & -N_4 & 0 & N_5 & 0 & -N_6 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & -N_3 & 0 & -N_4 & 0 & N_5 & 0 & -N_6 \end{bmatrix} \tag{3.58}$$

The element stiffness matrix, $[\boldsymbol{K_E}]$, is given by (see Section 2.6):

$$[\boldsymbol{K_E}] = \int_0^l [\boldsymbol{B}]^{\mathrm{T}}[\boldsymbol{D}][\boldsymbol{B}]\,\mathrm{d}l \tag{3.59}$$

where $l$ is the length of the element and the constitutive matrix $[\boldsymbol{D}]$ is given by Equation (3.44) or (3.45). The integral is evaluated in the natural ordinate system, see Figure 3.14, giving:

NANJING UNIVERSITY

$$[\boldsymbol{K_E}] = \int_{-1}^{1} [\boldsymbol{B}]^{\mathrm{T}}[\boldsymbol{D}][\boldsymbol{B}]|\boldsymbol{J}|\,\mathrm{d}s \tag{3.60}$$

where $|\boldsymbol{J}|$ is given by Equation (3.31).

The coordinates of the top and bottom of the interface are defined in terms of the nodal coordinates, using the shape functions (Equation (3.55)). For small displacement analysis, in which the calculations are based on the original geometry, coordinates $x$ and $y$ are:

$$\begin{aligned} x^{\text{top}} &= x^{\text{bot}} = N_1 x_1 + N_2 x_2 + N_5 x_5 \\ y^{\text{top}} &= y^{\text{bot}} = N_1 y_1 + N_2 y_2 + N_5 y_5 \end{aligned} \tag{3.61}$$

and

$$\begin{aligned} \frac{\mathrm{d}x}{\mathrm{d}s} &= N_1' x_1 + N_2' x_2 + N_5' x_5 \\ \frac{\mathrm{d}y}{\mathrm{d}s} &= N_1' y_1 + N_2' y_2 + N_5' y_5 \end{aligned} \tag{3.62}$$

where the prime denotes the derivative with respect to $s$. The trigonometric functions, $\sin\alpha$ and $\cos\alpha$, are given by Equation (3.30).

南京大學
NANJING UNIVERSITY

## 3.6.4 Comments on zero thickness interface elements

- Suffer from numerical instabilities if they have widely different stiffness from adjacent continuum/beam elements

- How to determine $K_n$ and $K_s$ ?

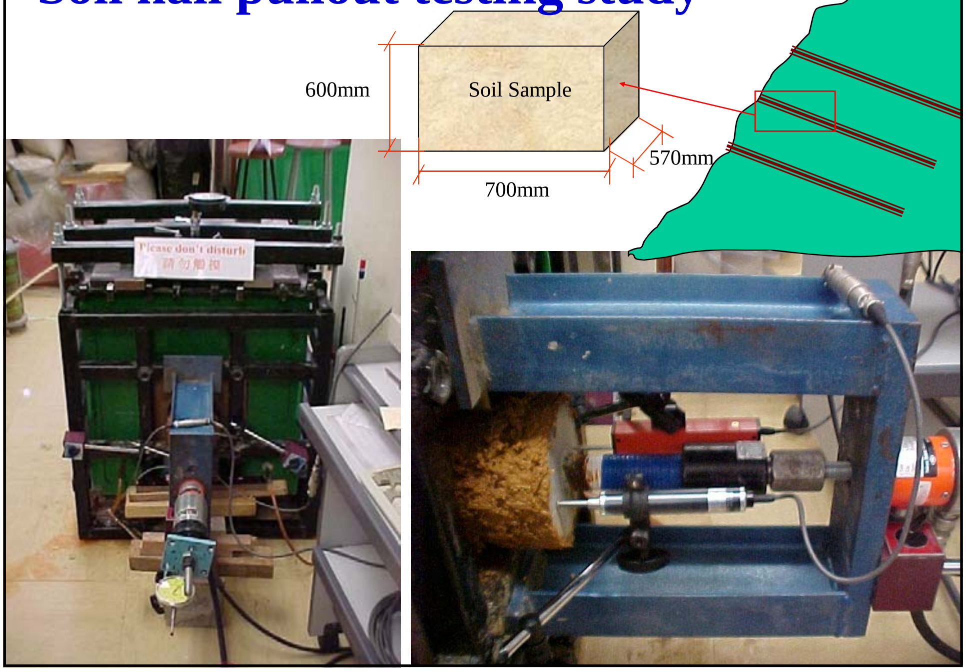
- Interface
  - true zero thickness case
  - non-zero thickness case

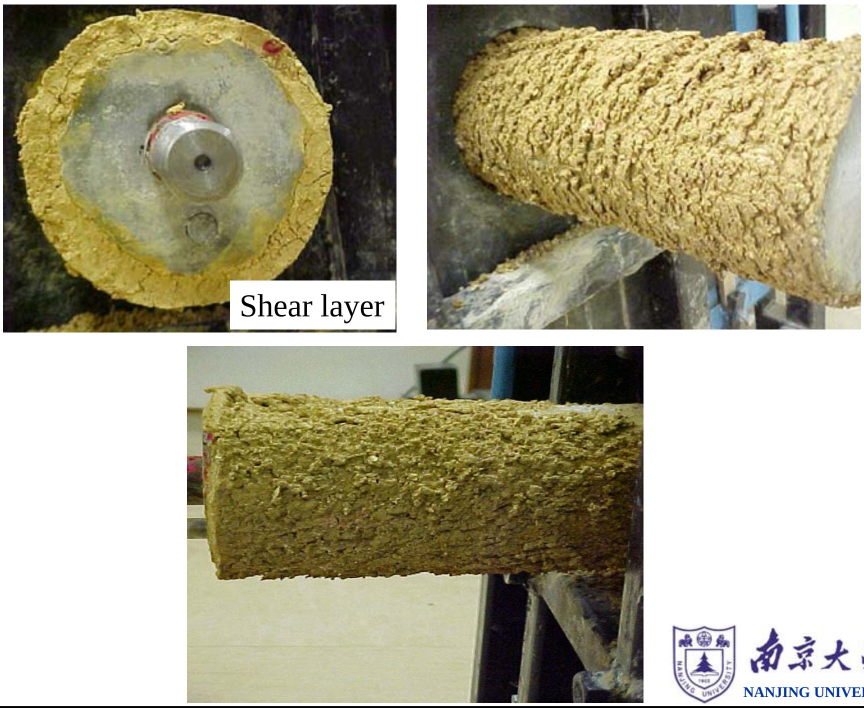
Large direct shear tests on soil-geotextile interface

Large direct shear tests on soil-geomembrane interface


Large direct shear tests on soil-geogrid interface

# Soil nail pullout testing study




南京大學
NANJING UNIVERSITY

## 3.7 Boundary conditions

### 3.7.1 Introduction

$$\left[K_G\right]\left\{\Delta d\right\}_{nG} = \left\{\Delta R_G\right\}$$

Boundary condition (BCs) types:

(a) BCs affecting $\{\Delta R_G\}$ - Point loads, Boundary stresses, Body forces, Construction, and Excavation

(b) BCs affecting $\{\Delta d\}_{nG}$ - Prescribed displacements

(c) BCs affecting the whole structure of the system equations - Local axes, Tied freedoms, Springs

NANJING UNIVERSITY

### 3.7.2 Local axes

- Local axes requires a transformation of the stiffness matrix and the right hand side load vector
- Local axes simulate sliding in a certain direction – only one degree of freedom ($v_l$=0 in Fig 3.15))

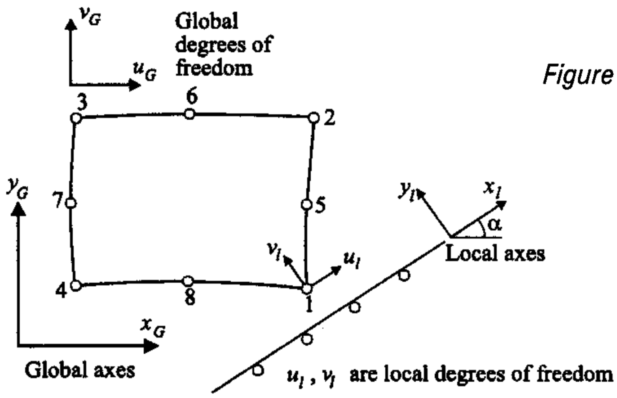



*Figure 3.15: Sliding boundary condition*

NANJING UNIVERSITY

If local axes are defined, it is necessary to transform the element stiffness matrices and the load boundary conditions prior to assembling the global system of equations. For 2D plane strain and axi-symmetric analyses the element stiffness matrix $[\boldsymbol{K_E}]$ is transformed from global axes to local axes by:

$$\left[\boldsymbol{K_E}\right]_{local} = \left[\boldsymbol{Q}\right]^{\mathrm{T}}\left[\boldsymbol{K_E}\right]_{global}\left[\boldsymbol{Q}\right] \tag{3.63}$$

where $[\boldsymbol{Q}]$ is a rotation matrix of direction cosines defined by the expression:

$$\left\{\Delta \boldsymbol{d}\right\}_{global} = \left[\boldsymbol{Q}\right]\left\{\Delta \boldsymbol{d}\right\}_{local} \tag{3.64}$$

which relates the local displacements to the global displacements. For example, for a 4 noded isoparametric element, the rotation matrix $[\boldsymbol{Q}]$ takes the form:

$$\left[\boldsymbol{Q}\right] = \begin{bmatrix} \cos\alpha_1 & -\sin\alpha_1 & 0 & 0 & 0 & 0 & 0 & 0 \\ \sin\alpha_1 & \cos\alpha_1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & \cos\alpha_2 & -\sin\alpha_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & \sin\alpha_2 & \cos\alpha_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & \cos\alpha_3 & -\sin\alpha_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & \sin\alpha_3 & \cos\alpha_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & \cos\alpha_4 & -\sin\alpha_4 \\ 0 & 0 & 0 & 0 & 0 & 0 & \sin\alpha_4 & \cos\alpha_4 \end{bmatrix} \tag{3.65}$$

where angles $\alpha_1$, $\alpha_2$, $\alpha_3$, $\alpha_4$ define the orientation of the local axes with respect to the global axes at each of the four nodes. In practice, the number of multiplications performed in evaluating Equation (3.63) is greatly reduced by only processing the non-zero sub-matrices of the matrix (3.65).

The transformation of the right hand side load vector can be performed in a similar manner. Using the definition of $\{\Delta \boldsymbol{R_E}\}$ given in Section 2.6, the transformed load vector becomes:

$$\left\{\Delta \boldsymbol{R_E}\right\}_{local} = \left[\boldsymbol{Q}\right]^{\mathrm{T}}\left\{\Delta \boldsymbol{R_E}\right\}_{global} \tag{3.66}$$

where $[\boldsymbol{Q}]$ is again of the form of Equation (3.65) (note: $[\boldsymbol{Q}]^{-1} = [\boldsymbol{Q}]^{\mathrm{T}}$). The transformation Equation (3.66) is indicated at the element level, however, in practice it is more convenient to take account of the local axes in the assembled right hand side vector, $\{\Delta \boldsymbol{R_G}\}$.

南京大學
NANJING UNIVERSITY

### 3.7.3 Prescribed displacements

- Certain prescribed displacements are needed to at least to restrict any rigid body translation or rotation; otherwise, the global stiffness matrix becomes singular and the system equations can NOT be solved.
- How to apply prescribed displacements:

(a) Specifying natural structure boundaries – footing/soil/rock bottom etc.

(b) Specifying sufficient number of displacements –
Example 1: left/right infinite domain by letting $u$=0 (see Figs. 2.20 and 2.21)
Example 2: see Fig.3.16

(c) For beam elements:
one point with $u$=$v$=$\theta$ *(rotation)*=0

NANJING UNIVERSITY

### Example 1:

- Strip footing (12m wide) – 2-D plane strain
- Half FE mesh – 20m x 20m > 3(B/2), reducing BC effects

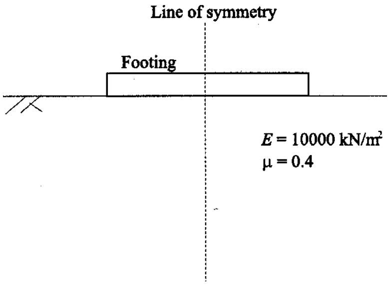



*Figure 2.20: Footing problem*

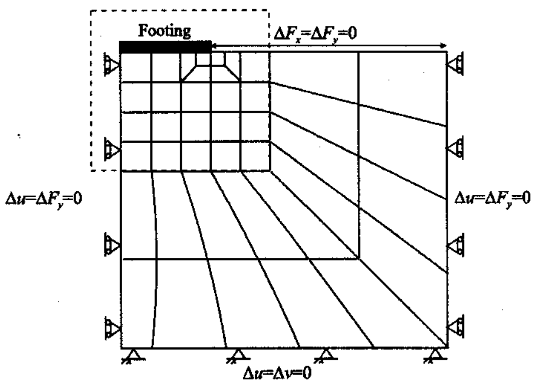



*Figure 2.21: Finite element mesh for strip footing problem*

## Example 2:

- A stress boundary problem with removal of rigid body mode (no translational move and no rotation)

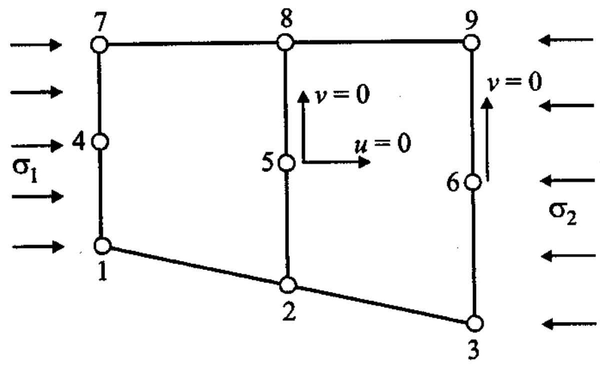



*Figure 3.16: Removal of rigid body modes*

南京大學
ANJING UNIVERSITY

The global equilibrium Equations (2.30), can be partitioned in the form:

$$\begin{bmatrix} \boldsymbol{K}_u & \boldsymbol{K}_{up} \\ \boldsymbol{K}_{up}^{\mathrm{T}} & \boldsymbol{K}_p \end{bmatrix} \begin{Bmatrix} \Delta \boldsymbol{d}_u \\ \Delta \boldsymbol{d}_p \end{Bmatrix} = \begin{Bmatrix} \Delta \boldsymbol{R}_u \\ \Delta \boldsymbol{R}_p \end{Bmatrix} \tag{3.67}$$

where $\Delta \boldsymbol{d}_u$ are the unknown degrees of freedom and $\Delta \boldsymbol{d}_p$ corresponds to the prescribed displacements. The first matrix equation from Equation (3.67) gives:

$$[\boldsymbol{K}_u]\{\Delta \boldsymbol{d}_u\} = \{\overline{\Delta \boldsymbol{R}_u}\} \tag{3.68}$$

where

$$\{\overline{\Delta \boldsymbol{R}_u}\} = \{\Delta \boldsymbol{R}_u\} - [\boldsymbol{K}_{up}]\{\Delta \boldsymbol{d}_p\} \tag{3.69}$$

Thus the unknown displacements $\{\Delta \boldsymbol{d}_u\}$ can be calculated from a modified system of global equilibrium Equations (3.68).

Having determined $\{\Delta \boldsymbol{d}_u\}$ from Equation (3.68), the second matrix Equation (3.67) gives:

$$\{\Delta \boldsymbol{R}_p\} = [\boldsymbol{K}_{up}]^{\mathrm{T}}\{\Delta \boldsymbol{d}_u\} + [\boldsymbol{K}_p]\{\Delta \boldsymbol{d}_p\} \tag{3.70}$$

Hence, the *Reaction forces* corresponding to each prescribed displacement can also be calculated.

## 3.7.4 Tired degrees of freedom

- This boundary condition allows **equal** displacement components to be imposed at one or more nodes whilst the magnitude of the components remain **unknown**
- Example: a smooth rigid strip footing

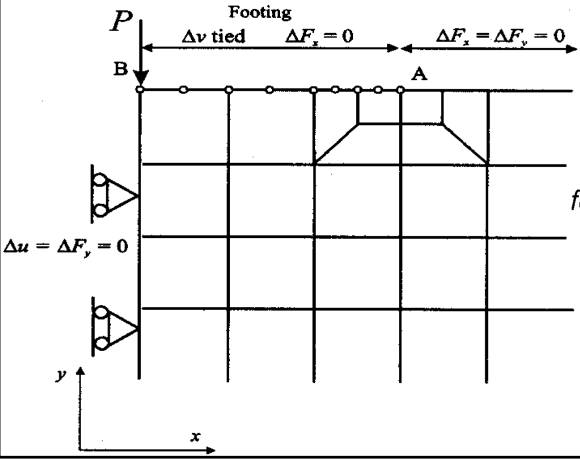



*Figure 3.17: Boundary conditions for a smooth strip footing subject to a vertical load P*



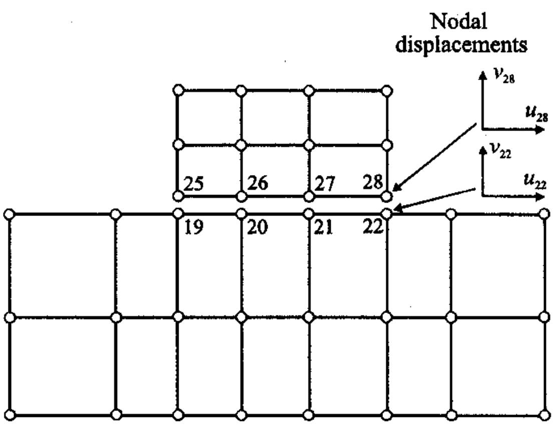



*Figure 3.18: Frictionless contact problem*

$$v_{19} = v_{25} \;;\; v_{20} = v_{26} \;;\; v_{21} = v_{27} \;;\; v_{22} = v_{28} \qquad (3.71)$$

### 3.7.5 Springs

- Spring is an alternative to membrane element which takes axial force only.
- Used to model strut, cable, anchor, soil nails or other structural elements subjected axial forces only
- The parameter is $k_s$
- First case: for an inclined spring (see Fig.3.21):



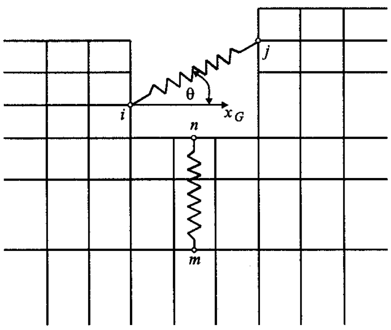



*Figure 3.21: Spring between 2 nodes*

$$k_s \begin{bmatrix} \cos^2\theta & \sin\theta\cos\theta & -\cos^2\theta & -\sin\theta\cos\theta \\ \sin\theta\cos\theta & \sin^2\theta & -\sin\theta\cos\theta & -\sin^2\theta \\ -\cos^2\theta & -\sin\theta\cos\theta & \cos^2\theta & \sin\theta\cos\theta \\ -\sin\theta\cos\theta & -\sin^2\theta & \sin\theta\cos\theta & \sin^2\theta \end{bmatrix} \begin{Bmatrix} \Delta u_i \\ \Delta v_i \\ \Delta u_j \\ \Delta v_j \end{Bmatrix} = \begin{Bmatrix} \Delta R_{xi} \\ \Delta R_{yi} \\ \Delta R_{xj} \\ \Delta R_{yj} \end{Bmatrix} \quad (3.72)$$

where $\theta$ is the inclination of the spring to the global $x_G$ axis, see Figure 3.21. The spring stiffness, $k_s$, multiplied by the trigonometric matrix above must be added to the global stiffness matrix during the assemble process. It affects the terms relating to the displacement degrees of freedom of nodes $i$ and $j$.

- Second case: a horizontal spring for strut (see Fig.3.22):

$$k_s\begin{bmatrix}\cos^2\theta & \sin\theta\cos\theta \\ \sin\theta\cos\theta & \sin^2\theta\end{bmatrix}\begin{Bmatrix}\Delta u_i \\ \Delta v_i\end{Bmatrix}=\begin{Bmatrix}\Delta R_{xi} \\ \Delta R_{yi}\end{Bmatrix} \quad (3.73)$$

where, in general, $\theta$ is the inclination of the spring to the global $x_G$ direction. In the above example $\theta$ =0°. Again, the spring stiffness, $k_s$, multiplied by the trigonometric matrix must be added to the global stiffness matrix during the assembly process.

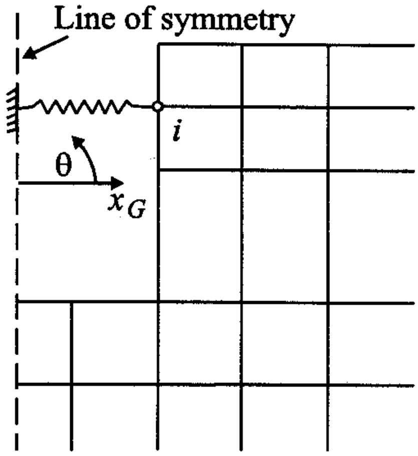



*Figure 3.22: Spring at a single node*



- Third case: continuous spring support (see Fig.3.23):

For a single element side, contribution to the global stiffness matrix takes the form:

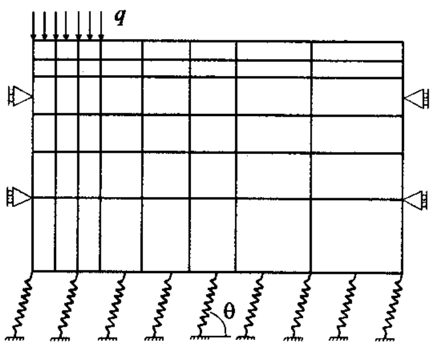



*Figure 3.23: Continuous spring along mesh boundary*

$$\int_{Srf}[N]^{T}[K_s][N]\,\mathrm{d}Srf$$

where:

$$[K_s]=k_s\begin{bmatrix}\cos^2\theta & \sin\theta\cos\theta \\ \sin\theta\cos\theta & \sin^2\theta\end{bmatrix}$$

### 3.7.6 Boundary stresses

- Examples such as pressure loading (see Fig.3.24)
- Stress boundary conditions must be converted to **equivalent nodal forces**

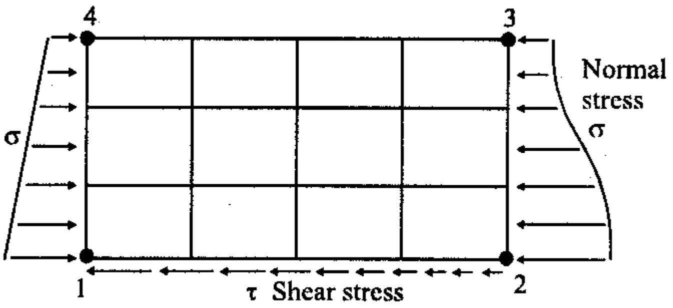



*Figure 3.24: Example of stress boundary conditions*

Equivalent nodal forces:

$$\{\Delta \boldsymbol{R}_E\} = \int_{Srf} [N]^{\mathrm{T}} \{\Delta \boldsymbol{T}\}\, \mathrm{d}Srf \tag{3.75}$$

where $[N]$ is given by Equation (2.8), $\{\Delta \boldsymbol{T}\}$ is the incremental global surface traction vector (i.e. boundary stresses) and '$Srf$' (i.e. Surface) is the element side over which tractions are prescribed. The integral (3.75) can be evaluated numerically for each element side over which the tractions act. The first stage of this process is to transform the surface integral (3.75) into an one dimensional form in the natural coordinate system:

$$\{\Delta \boldsymbol{R}_E\} = \int_{-1}^{1} t [N']^{\mathrm{T}} \{\Delta \boldsymbol{T}\} |\boldsymbol{J}'|\, \mathrm{d}s \tag{3.76}$$

where $t$ is unity for plane strain problems and equals $2\pi r$ for axi-symmetric problems, $[N']$ contains the interpolation functions on the element side, and $|\boldsymbol{J}'|$ is the Jacobian determinant obtained from mapping the element side from the

南京大學 NANJING UNIVERSITY

global element to the parent element. For example, for a 4 noded isoparametric element, $[N']$ takes the form:

$$[\boldsymbol{N}'] = \begin{bmatrix} N_1' & 0 & N_2' & 0 \\ 0 & N_1' & 0 & N_2' \end{bmatrix} \tag{3.77}$$

where

$$\begin{aligned} N_1' &= \tfrac{1}{2}(1-S) \\ N_2' &= \tfrac{1}{2}(1+S) \end{aligned} \tag{3.78}$$

are interpolation functions for the element side. The Jacobian determinant for each point on the element side is given by:

$$|\boldsymbol{J}'| = \left(\left(\frac{dx}{dS}\right)^2 + \left(\frac{dy}{dS}\right)^2\right)^{\frac{1}{2}} \tag{3.79}$$

where the coordinate derivatives are obtained by differentiating the simplified isoparametric relations:

$$x = \sum_{i=1}^{2} N_i' x_i \quad ; \quad y = \sum_{i=1}^{2} N_i' y_i \tag{3.80}$$

and $x_i$, $y_i$ are the global coordinates of the two nodes on the element side.

NANJING UNIVERSITY

of the integrand evaluated at a number of Gaussian integration points.

To determine the integration point value of the surface traction vector $\{\Delta T\}$, the applied stress must be transformed according to the orientation of the surface element at the integration point and the defined sign convention for stresses. One such sign convention is that normal stresses ($\sigma$) are positive if oriented outwards from the boundary of the body, while shear stresses ($\tau$) are positive if oriented in a tangentially anticlockwise sense with respect to the boundary of the body. Using this convention gives:

$$\{\Delta \boldsymbol{T}_I\} = \sigma_I \begin{Bmatrix} \cos\theta_I \\ \sin\theta_I \end{Bmatrix} \tag{3.81}$$

if normal stresses are prescribed, or

$$\{\Delta \boldsymbol{T}_I\} = \tau_I \begin{Bmatrix} -\sin\theta_I \\ \cos\theta_I \end{Bmatrix} \tag{3.82}$$

if shear stresses are prescribed, where $\theta_I$ is the angle between the boundary normal and the global $x_G$ axis, and the subscript $I$ denotes the integration point value. The angle $\theta_I$ is determined by inverting the expression:

NANJING UNIVERSITY

$$\tan(\theta_l + 90) = \frac{dy}{dx} = \left[ \frac{\sum_{i=1}^{2} \frac{\partial N_i^l}{\partial S} y_i}{\sum_{i=1}^{2} \frac{\partial N_i^l}{\partial S} x_i} \right]_{at\ S_l} \tag{3.83}$$

which is obtained by differentiating Equation (3.80).

In all cases the equivalent nodal forces, which are calculated from Equation (3.76), are initially referred to the global system of axes. If local axes are defined, the nodal forces are transformed accordingly, as described in Section 3.7.2.

NANJING UNIVERSITY

### 3.7.7 Point loads

- True point loads
- Traction boundary (stress) conditions is converted to discrete nodal point loads
- For plane strain and axi-symmetrical analysis, they are line loads !

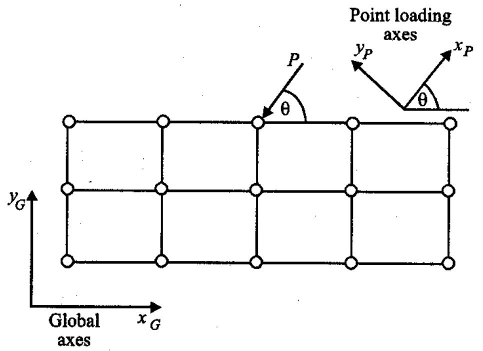



*Figure 3.25: Orientation of point loading axes*

NANJING UNIVERSITY

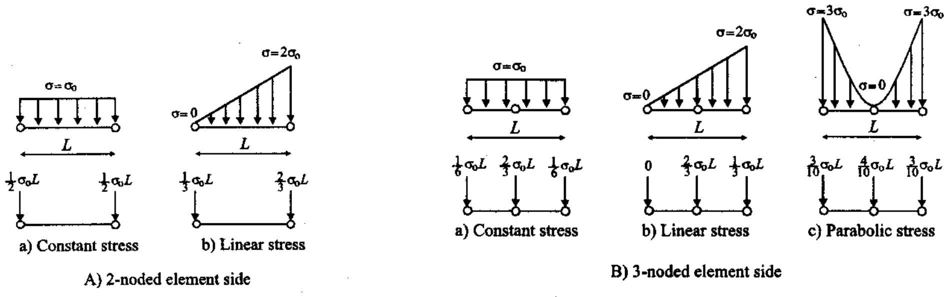



*Figure 3.26: Equivalent nodal forces for element side with A) 2 nodes and B) 3 nodes*



### 3.7.8 Body forces

- Gravity loading or body forces are important forces as they influence the deformation and failure of geotechnical structures a lot !
- Body forces shall be converted to equivalent nodal forces

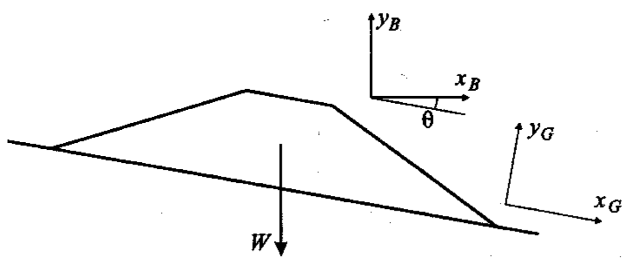



*Figure 3.27: Body force loading axes for sediment layer on a slope*

## The body force's equivalent nodal forces are:

The nodal forces equivalent to the body force are calculated element-wise, using the body force contribution to the right hand side vector, see Section 2.6:

$$\left\{\Delta \boldsymbol{R}_E\right\} = \int_{Vol} \left[\boldsymbol{N}\right]^{\mathrm{T}} \left\{\Delta \boldsymbol{F}_G\right\} \, \mathrm{d}Vol \tag{3.84}$$

where $[\boldsymbol{N}]$ is given by Equation (2.8), $\{\Delta \boldsymbol{F}_G\}$ is the global body force vector and $Vol$ is the volume of the element. The body force vector $\{\Delta \boldsymbol{F}_G\}$ is determined with respect to the global axes by using:

$$\begin{Bmatrix} \Delta F_{xG} \\ \Delta F_{yG} \end{Bmatrix} = \Delta\gamma \begin{Bmatrix} \cos\beta \\ \sin\beta \end{Bmatrix} \quad (3.8\ \ )$$



where $\Delta\gamma$ is the increment of bulk unit weight; $\beta = \theta$ or $\theta$+90° depending on whether $\Delta\gamma$ refers to the $x_B$ or $y_B$ component of body force, respectively; and $\theta$ is the angle between the $x_B$ and $x_G$ axes.

The integral (3.84) is evaluated numerically over each element, by using the procedure described in Section 2.6.1 for the element stiffness matrix. First, the body force integral is transformed to the natural coordinate system:

$$\left\{\Delta \boldsymbol{R}_E\right\} = \int_{-1}^{1}\int_{-1}^{1} t \left[\boldsymbol{N}\right]^{\mathrm{T}} \left|\boldsymbol{J}\right| \, \mathrm{d}S \, \mathrm{d}T \left\{\Delta \boldsymbol{F}_G\right\} \tag{3.86}$$

where $t$ equals unity for plane strain problems and $t$=2$\pi r$ for axi-symmetric problems; and $|\boldsymbol{J}|$ is the determinant of the Jacobian, given by Equation (2.16), arising from the mapping between the global and parent elements. The global force vector $\{\Delta \boldsymbol{F}_G\}$ can be removed from the integral, since it is constant over the element. The integral in Equation (3.86) is determined using a two dimensional Gaussian integration procedure, as described in Section 2.6.1.

## 3.7.9 Construction

- Many constructions involve placing of new earth materials
- Examples: embankment construction, backfilling behind a retaining wall, backfilling a trench, etc.
- How to use FE to simulate ?

- Elements representing the material to be constructed must be present in the original finite element mesh, but must be *deactivated*, either by prior excavation, or at the outset of the analysis. On construction the elements are reactivated.
- Construction of material must be performed incrementally since, even for a linear elastic material, superposition does not hold. When constructing an embankment the layered construction procedure must be followed, with each increment of the analysis simulating the construction of a layer of fill.
- During construction, the elements representing the new material must have a constitutive model which is consistent with its behaviour during construction. Once constructed, the constitutive model should change to represent the behaviour of the material once placed.

NANJING UNIVERSITY

- When an element is constructed, the addition of its weight to the finite element mesh must be simulated by applying self weight body forces to the constructed element.

When constructing material, the following procedure is recommended:

- Divide the analysis into a set of increments and make sure that all elements to be constructed are deactivated prior to the increment at which construction starts.
- For a particular increment the elements to be constructed are reactivated and given a constitutive model appropriate to the material behaviour during placing. This often means that the material has a low stiffness.
- Nodal forces due to the self weight body forces of the constructed material are calculated in a similar fashion to that explained for body forces in Section 3.7.8, and added to the right hand side load vector.

南京大学
NANJING UNIVERSITY

- The global stiffness matrix and all other boundary conditions are assembled for the increment. The equations are solved to obtain the incremental changes in displacements, strains and stresses.
- Before application of the next increment, the constitutive model for the elements just constructed is changed to represent the behaviour of the fill material once placed. Displacements of any nodes which are only connected to the constructed elements (i.e. not connected to elements that were active at the previous increment) are zeroed. Depending on the constitutive models used to represent the constructed material, it may be necessary to establish state parameters (e.g. hardening parameters for elasto-plastic models) and/or adjust the stresses in the constructed elements. If the stresses are adjusted, then care must be taken that equivalent changes are made to the accumulated right hand side vector to ensure equilibrium is maintained.
- Apply the next increment of analysis.



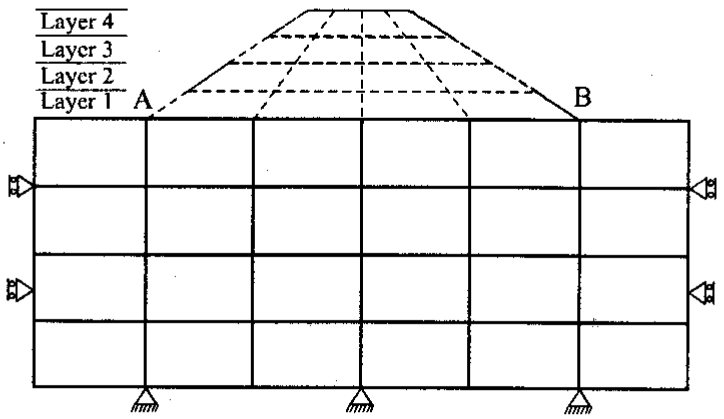



*Figure 3.28: Embankment construction*

南京大學
NANJING UNIVERSITY

## 3.7.10 Excavation

- Excavation involve removing earth materials
- Examples: excavation for foundation construction, tunnel excavation, metro ***cut***-cover-tunnel, etc.
- How to use FE to simulate excavation ?
  - Determine tractions *T* which are equal to the internal stresses in the soil mass before excavation between A and B
  - Apply the equal and opposite –*T* at the boundary A-B
  - The behavior of soil mass B is then simulated

NANJING UNIVERSITY

Simulation of a stage of excavation therefore involves determination of the tractions, *T*, at the new soil boundary, determination of the stiffness of the soil mass B, and application of tractions, -*T*, to that new soil boundary. Finite element implementation of this process involves determination of the nodal forces which are equivalent to the tractions shown in Figure 3.29c. These forces can be calculated from the excavated elements adjacent to the excavation boundary using:

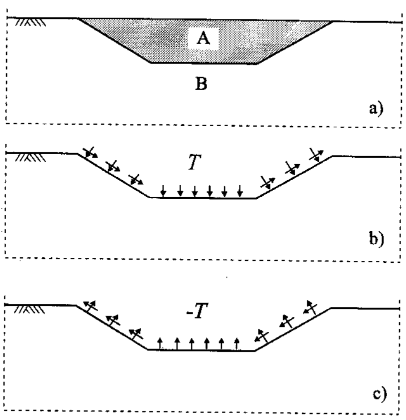



*Figure 3.29: Simulation of excavation*

$$\{\boldsymbol{R}_E\} = \int_{Vol} [\boldsymbol{B}]^{\mathrm{T}} \{\sigma\}\, \mathrm{d}Vol - \int_{Vol} [\boldsymbol{N}]^{\mathrm{T}} \gamma\, \mathrm{d}Vol \tag{3.87}$$

where $\{\sigma\}$ is the stress vector in the element, $\gamma$ is the bulk unit weight and *Vol* is the volume of the excavated element. Only the forces appropriate to the nodes on the excavated surface are placed in $\{\boldsymbol{R}_E\}$. This calculation is repeated for all excavated elements adjacent to the excavation boundary. This procedure is based on Brown and Booker (1985).

where the volume integral is now over the volume of an element and the surface integral is over that portion of the element boundary over which surface tractions are specified. The principal unknowns are the incremental nodal displacements over the whole mesh, $\{\Delta \boldsymbol{d}\}_n$. Minimising the potential energy with respect to these incremental nodal displacements gives:

$$\delta\Delta E = \sum_{i=1}^{N} (\{\delta\Delta \boldsymbol{d}\}_n^{\mathrm{T}})_i [\int_{Vol} [\boldsymbol{B}]^{\mathrm{T}}[\boldsymbol{D}][\boldsymbol{B}] \, \mathrm{d}Vol \, \{\Delta \boldsymbol{d}\}_n - \int_{Vol} [\boldsymbol{N}]^{\mathrm{T}} \{\Delta \boldsymbol{F}\} \, \mathrm{d}Vol - \int_{Srf} [\boldsymbol{N}]^{\mathrm{T}} \{\Delta \boldsymbol{T}\} \, \mathrm{d}Srf]_i = 0 \tag{2.24}$$

which is equivalent to a set of equations of the form:

$$\sum_{i=1}^{N} [\boldsymbol{K_E}]_i (\{\Delta \boldsymbol{d}\}_n)_i = \sum_{i=1}^{N} \{\Delta \boldsymbol{R_E}\} \tag{2.25}$$

where: $[\boldsymbol{K_E}] = \int_{Vol} [\boldsymbol{B}]^{\mathrm{T}}[\boldsymbol{D}][\boldsymbol{B}] \, \mathrm{d}Vol =$ *Element stiffness matrix*;

$\{\Delta \boldsymbol{R_E}\} = \int_{Vol} [\boldsymbol{N}]^{\mathrm{T}} \{\Delta \boldsymbol{F}\} \, \mathrm{d}Vol + \int_{Srf} [\boldsymbol{N}]^{\mathrm{T}} \{\Delta \boldsymbol{T}\} \, \mathrm{d}Srf =$ *Right hand side load vector*.

NANJING UNIVERSITY

When simulating excavation in a geotechnical problem it is usual that structural elements or supports are added as excavation proceeds. It is therefore necessary to split the analysis into a sequence of increments. This is also necessary if nonlinear constitutive models are used. The procedure followed in the analysis is therefore as follows:

- Specify the elements to be excavated for a particular increment.
- Using Equation (3.87) determine the equivalent nodal forces to be applied to the excavation boundary to simulate removal of the elements. Tag the elements to be excavated as *deactivated* and remove them from the active mesh.
- Assemble the remaining boundary conditions and the global stiffness matrix using the active mesh. Solve the finite element equations to give the incremental changes in displacements, stresses and strains.
- Add the incremental changes of displacements, stresses and strains to the accumulated values existing before the increment to give the updated accumulated values.
- Perform the next increment of the analysis.

NANJING UNIVERSITY

## 3.7.11 Pore pressures

- When performing analysis in which total stresses are expressed using effective stresses and pore water pressure (see Section 3.4), the pore water pressure may be specified
- This is possible to simulate deformation due to pore water pressure changes
- Examples: settlement due to de-watering, de-watering in excavation, de-watering in tunneling, etc.
- How to use FE to simulate de-watering caused deformation ? Porewater pressure is converted to equivalent nodal forces
- This approach can simulate the final deformation due to de-watering, but is not fully coupled consolidation analysis ! Biot's theory – full coupling of soil skeleton deformation and pore water pressure dissipation.

南京大學
NANJING UNIVERSITY

In the finite element analysis it is necessary to specify changes in pore fluid pressure in terms of equivalent nodal forces. It can be shown that for an element with a specified change in pore fluid pressure, $\Delta p_f$, the equivalent nodal forces are given by:

$$\{\boldsymbol{R}_E\} = -\int_{Vol} [\boldsymbol{B}]^{\mathrm{T}} \{\Delta\sigma_f\}\, \mathrm{d}Vol \qquad (3.88)$$

where $\{\Delta\sigma_f\}$ is given by Equation (3.3). Two scenarios often occur:

(a) Pore water pressure changes due to de-watering

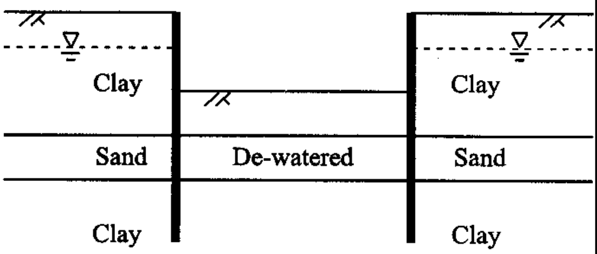



*Figure 3.30: Excavation de-watering*

(b) Pore water pressure increase due to loading $q$ and dissipation due to drainage

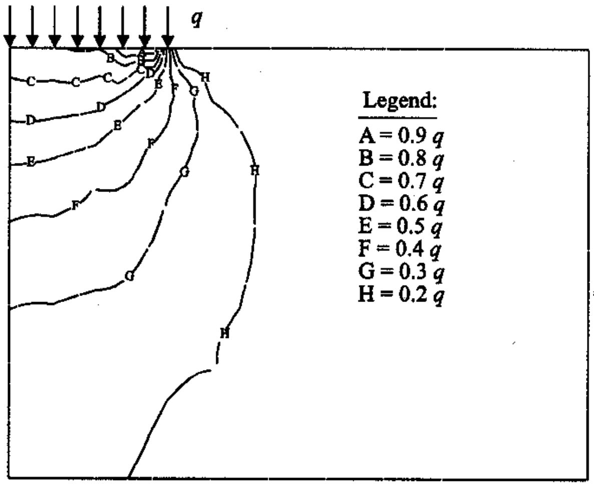



*Figure 3.31: Excess pore water pressures under smooth flexible strip footing*

## 3.8 Summary

1. Drained analysis in which pore water pressures are ignored can be performed by using effective stress material properties.
2. Undrained analysis can be performed in terms of total stress using total stress material properties. Care must be taken when Poisson's ratio approaches 0.5.
3. Analysis can also be performed in which the total stress tensor is split into effective stresses and pore fluid pressures. This enables undrained analysis to be carried out using effective stress material properties in which the change in effective stresses and pore water pressures are calculated. It also allows drained analyses to be carried out, in which changes in the pore water pressures can be prescribed.
4. It is often difficult to model structural elements using continuum finite elements. Alternative structural elements are available. Beam/shell elements can be used when the structural member can sustain bending moments and axial and shear forces. Alternatively, if the structural element can only sustain axial forces, membrane elements may be used. A detailed description of both elements has been given.
5. The interface between structural elements and soil can be modelled in a variety of ways. A special interface element which has zero thickness has been described in this chapter. The use of this element in practical boundary value problems is discussed in Volume 2 of this book.

6. There are three classes of boundary conditions that arise in finite element analysis. The first class affects only the right hand side of the governing equations. These conditions are loading conditions such as boundary stresses, body forces, etc... The second class affects the left hand side of the governing equations. These are kinematic conditions such as prescribed displacements. The third class is more complex, since they affect the whole structure of the system equations. These conditions include local axes, tied freedoms and springs.
7. The following boundary conditions have been described in detail:
    - Local axes;
    - Prescribed displacements;
    - Tied degrees of freedom;
    - Springs;
    - Boundary stresses;
    - Point loads;
    - Body forces;
    - Construction;
    - Excavation;
    - Pore pressures.