NUMERO: 2060A - ANNO: 2016

A P P U N T I

STUDENTE: Aimar Mauro

MATERIA: Metodi numerici per l'ingegneria geotecnica (I parte) - Prof. Barla Musso

Geotechnical engineering and design

1 Geotechnical engineering:

it is the branch of civil engineering concerned with the engineering behaviour of earth materials.
Geotechnical engineering uses the principles of soil mechanics and rock mechanics to investigate subsurface conditions and materials, determine their properties, evaluate stability of natural slopes and man-made soil deposits, design earthworks, monitor site conditions (as we don't know properties, we test for determine real behaviour and test the validity of the model), etc.

↳ GEOTECHNICAL ENGINEERING IS THE APPLICATION OF SOIL MECHANICS AND ROCK MECHANICS

Geotechnical engineering is wider than engineering fundamentals because it involves not only mechanics, but there is an interaction with geology, hydrogeology, structural engineering, design, regulation, interaction with public policy and lawiers

⇒ many aspects are related to geotechnical engineering

2 Design:

it is the logic process that goes from an idea (idealisation of a structure interacting with the ground) to the construction

DESIGN = IDEA → CONSTRUCTION

This process very often requires to adopt numerical methods.

PROPERTIES → in the process of design an important step is define an appropriate **model** = SCHEMATIC AND SIMPLIFIED REPRESENTATION OF THE ENGINEERING PROBLEM.

→ Is imperative to know the properties of the geomaterials interacting with the structure.
For this reason, LABORATORY AND IN SITU INVESTIGATIONS AND TESTING ARE REQUIRED, in relation with the problem.
Sometimes happens that investigation and design are performed indipendently: planning of investigation is carried out without knowing the design requirements.
This approach is forbidden by the "Legge quadro".

"Ogni elemento del progetto [the ground is a component of the project] deve essere identificabile in forma, tipologia, qualità, dimensione e prezzo".

By consequence, bidding of infrastructural works (also investigation) can be taken place only if a final design has been carried out.

In Italy, today we also refer to the Norme Tecniche per le Costruzioni (NTC2008).

As they use limit state approach, they are in line with the Eurocodes, and they define the PARTIAL SAFETY FACTORS.

The chapter 6 is devoted to geotechnical design but it isn't equilibrated, as it is detailed about foundations while gives not much information about tunnels and slope stability (it only attributes to the engineer the responsibility in the choice of the safety factor).

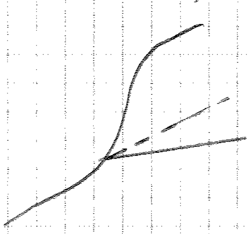
In EC7 and NTC2008, there are two fundamental aspects

→ **separation between geological report and geotechnical report:**

these must be two different document and the second is engineer's responsibility.
In geotechnical report, there is a description of site investigation and laboratory tests and geotechnical computations.

→ introduction of the **observational method**, which is very helpful especially in geotechnical engineering.

The method was introduced by Terzaghi and Peck, from the consideration that, if we have some level of uncertainty, we don't fully understand the problem and we can't make a conclusion.



For instance, the problem of stability of a rock slope with discontinuities is dramatically influenced by orientation.
In investigation, we obtain an average orientation. With this, we do the computation and make the design.
In reality, the orientation may be different and this dramatically changes the design.

⇒ there is UNCERTAINTY, as we don't know reality, and this is bigger than in structural materials

A solution consists in reducing uncertainty (but it won't never be zero) by doing many investigations, if possible.

This way is complex and usually observational method is used:

in this method, the design is reviewed during the construction and becomes an **interactive geotechnical design**.

The observational method consists in some steps.

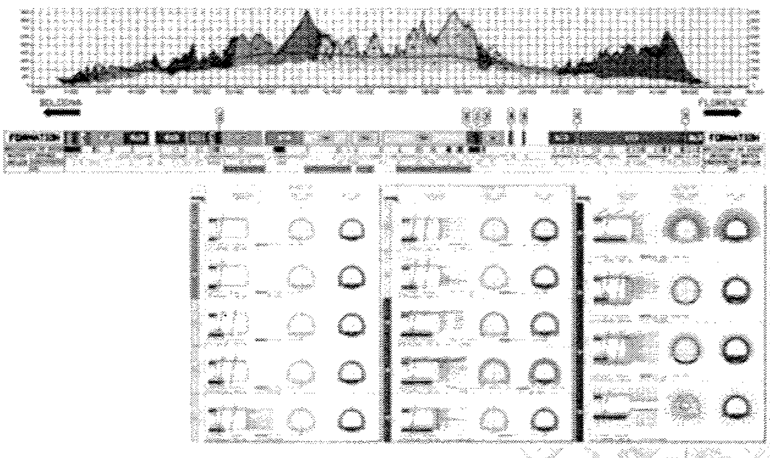
(I) Setting **limits of behaviour** which are acceptable (for instance, in slope, a displacement).
In setting limits, we also need to fix the range of possible behaviours (what can happen).

(II) Setting a **monitoring plan**, which will reveal whethere the actual behaviour lies within the acceptable limits.
The monitoring is based on a period of investigation, in which we use authomatic or manual reading instruments.

(III) **Response time**, in which we get numbers from instruments, interpret them and take decisions based on limits.
This time shall be sufficiently rapid in relation to the possible evolution of the system.

THE OBSERVATIONAL METHOD

When the observational method is applyed EARLY & PROPERLY, support systems can be designed for each condition.



NUMERICAL MODELLING WITHIN NTC

Numerical modelling is implicitly included among the computational methods but it is explicitly mentioned with reference to underground excavations (pgf. 6.7.5 Metodi di calcolo, pagina 215):

Per lo svolgimento delle analisi progettuali si deve fare riferimento ai modelli geotecnici di sottosuolo di riferimento e a leggi di comportamento note e di provata validità. Inoltre, si deve ricorrere a metodi e procedimenti di calcolo di comprovata validità, adeguati alla complessità del sistema opera-terreno e al livello di progettazione. In generale si deve ricorrere ad uno o più dei seguenti procedimenti:
a) metodi analitici;
b) metodi numerici, per simulare il comportamento del sistema opera-terreno, nelle diverse fasi di scavo e costruzione, nonché in condizioni di esercizio.

Design methods

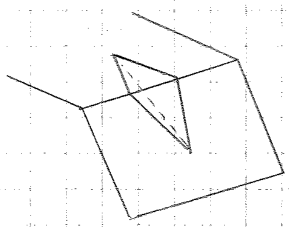
I Design methods are divided into 3 categories

→ **empirical methods:**

these are methods based on PREVIOUS EXPERIENCES (~~use~~ they use the knowledge) as they derived from construction of structures similar to the one to be designed.
They typically consist in tables and classifications (like RMR) built on base of previous experience.

→ **limit equilibrium method:**

this method is applied by performing a LIMIT EQUILIBRIUM ANALYSIS, based on strength and active forces on the rock mass and characteristics of discontinuities.



The method is applied in the situations where we can identify geometry (discontinuities) and forces and solve by ~~a~~ limit equilibrium.
Even if there is a set of discontinuities and situation is more confused, we can simplify geometry and study by limit equilibrium.

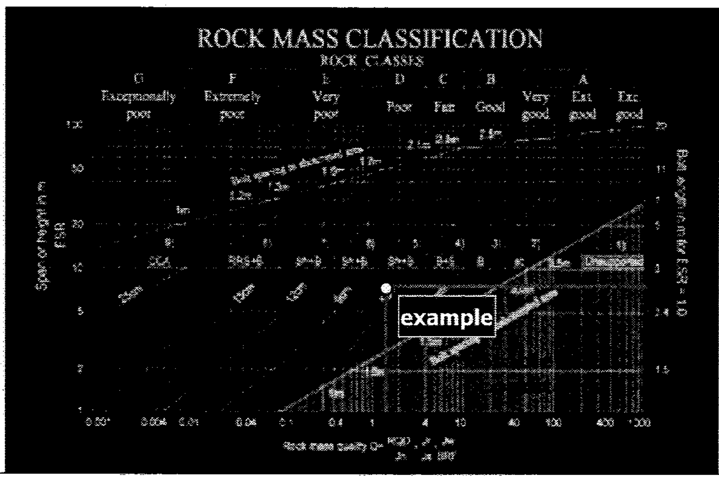
→ **stress analysis**

This method consists in applying the STATE OF STRESS - STRAIN in the structure with consideration given to the strength and deformability properties of rock mass and discontinuities.
So, we focus on stress, strains and displacement in whole structure (for instance, in a foundation we study stresses and strains in concrete and ground) and we work in a continuous or discontinuous environment - in soil mechanics we often refer to a continuous environment; in rock mechanics we have to decide in relation to the problem.
Stress analysis uses both analytical methods (like Kirsch solution), even if their application is limited, and NUMERICAL METHODS.

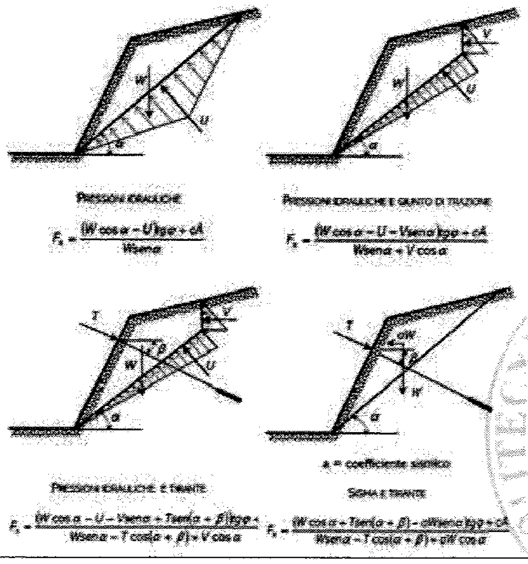
EMPIRICAL METHODS

Stability evaluation based on Q index

- ESR = 1.6 D_{eq} = 2.5 m



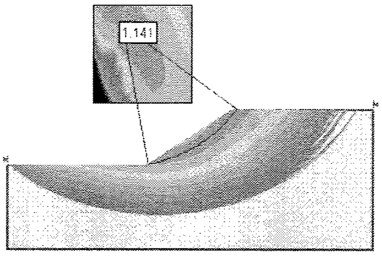
LIMIT EQUILIBRIUM METHOD



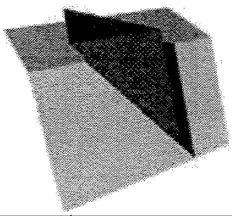
Practical examples of evaluation of the safety factor by LEM.

LIMIT EQUILIBRIUM METHOD

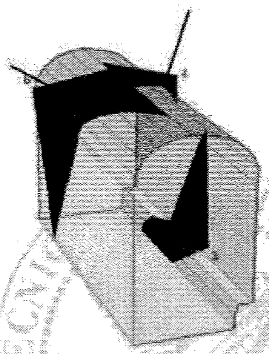
LEM can be applied to a number of geotechnical problems



Slopes



Rock wedges



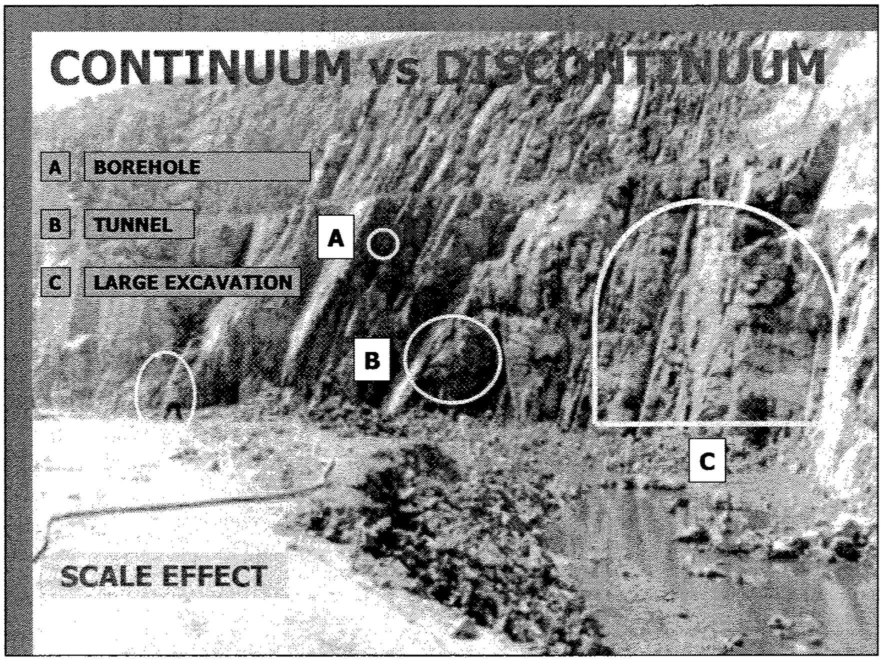
Underground excavations

STRESS ANALYSIS

- STRESS $\{\sigma\}$
- STRAIN $\{\varepsilon\}$
- DISPLACEMENT $\{u\}$

Stress analysis is the most complete method as both stress and strain distributions are obtained following computation

The soil/rock mass is represented as a continuum or discontinuum



CONTINUUM vs DISCONTINUUM

EQUIVALENT CONTINUUM MODEL, the rock mass is treated as a continuum with equal in all directions input data for the strength and deformability properties, which define a given constitutive relation for the medium: elastic, elasto-plastic, etc.

DISCONTINUUM MODEL, the rock mass is represented as a discontinuum and most of the attention is devoted to the characterisation of the rock elements and the rock joints/ discontinuities. The modelling approach consists in considering the blocky nature of the system being analysed. Each block may interact with the neighbouring blocks trough the joints.

Stresses and strains

1. Matrix algebra:

A **matrix** consist on a collection of quantities, called COMPONENTS.
Focusing on 2D matrixes, we can identify a certain number of columns and rows and the dimension of the matrix is defined as it follows.

rows X columns

Ex.

$$a = \begin{pmatrix} a_1 \\ a_2 \\ a_3 \end{pmatrix} \quad 3\times1 \qquad A = \begin{bmatrix} A_{11} & A_{12} \\ A_{21} & A_{22} \\ A_{31} & A_{32} \end{bmatrix} \quad 3\times2$$

PROPERTIES → the **transpose** is a matrix in which we change the positions and rows become columns. It's written whit a small letter "t".

$$a^t = \begin{pmatrix} a_1 & a_2 & a_3 \end{pmatrix} \qquad A^t = \begin{bmatrix} A_{11} & A_{21} & A_{31} \\ A_{12} & A_{22} & A_{32} \end{bmatrix}$$

→ the **identity matrix** is a square matrix of size n with all 1 one the main diagonal and 0 elsewhere.

$$I_2 = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

The **zero matrix** is a square matrix of size n with 0 everywhere.

$$O_2 = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix}$$

→ **matricial multiplication** is an operation done rows for columns and we have the following expression.

$$\underset{m\times p}{C} = \underset{m\times n}{A} \cdot \underset{n\times p}{B}$$

We notice that engineering shear γ_{xy} (angular shear of initially orthogonal fibers) is different from angle shear ε_{12}.

$$\gamma_{xy} = \frac{1}{2}\varepsilon_{12}$$

If we have the stress tensore in a point, sometimes we have to change the reference system. This operation uses a matrix which components are the direction cosines l, m, n.

$$\begin{bmatrix} \sigma_x' & \tau_{xy}' & \tau_{xz}' \\ \tau_{yx}' & \sigma_y' & \tau_{yz}' \\ \tau_{zx}' & \tau_{zy}' & \sigma_z' \end{bmatrix} = \begin{bmatrix} l_1 & l_2 & l_3 \\ m_1 & m_2 & m_3 \\ n_1 & n_2 & n_3 \end{bmatrix} \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \begin{bmatrix} l_1 & l_2 & l_3 \\ m_1 & m_2 & m_3 \\ n_1 & n_2 & n_3 \end{bmatrix}$$

$$l_1 = x'x \qquad l_2 = x'y \qquad l_3 = x'z$$

$$m_1 = y'x \qquad m_2 = y'y \qquad m_3 = y'z$$

$$n_1 = z'x \qquad n_2 = z'y \qquad n_3 = z'z$$

When we change the reference system, some quantities don't change and these are called invariants

→ I STRESS INVARIANT : it is the sum of normal components

$$I_\sigma = I_{1\sigma} = \sigma_x + \sigma_y + \sigma_z$$

→ II STRESS INVARIANT

$$II_\sigma = I_{2\sigma} = \sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_z\sigma_x - \tau_{xy}^2 - \tau_{yz}^2 - \tau_{zx}^2$$

→ III STRESS INVARIANT

$$III_\sigma = I_{3\sigma} = \sigma_x\sigma_y\sigma_z - \sigma_x\tau_{yz}^2 - \sigma_y\tau_{zx}^2 - \sigma_z\tau_{xy}^2 + 2\tau_{xy}\tau_{yz}\tau_{zx}$$

Exists a specific reference system, where shear stresses are 0 and normal stresses are called PRINCIPAL STRESSES. This is the principal reference system.
In this system, the expressions of the invariants become simpler.

$$I_{1\sigma} = \sigma_1 + \sigma_2 + \sigma_3$$

$$I_{2\sigma} = \sigma_1\sigma_2 + \sigma_2\sigma_3 + \sigma_1\sigma_3 \qquad \sigma_1, \sigma_2, \sigma_3 = \text{principal stresses}$$

$$I_{3\sigma} = \sigma_1\sigma_2\sigma_3$$

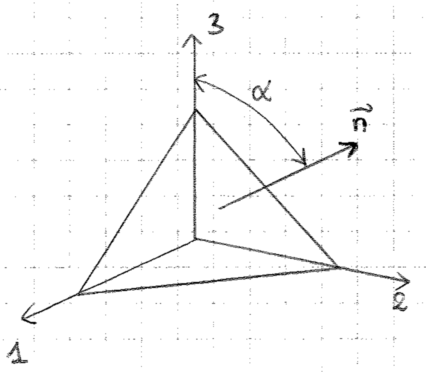
The invariants of deviatoric stress are the following.

$$J_{1\sigma} = \left(\sigma_x - \frac{I_\sigma}{3}\right) + \left(\sigma_y - \frac{I_\sigma}{3}\right) + \left(\sigma_z - \frac{I_\sigma}{3}\right) = \cancel{I_\sigma} - \cancel{I_\sigma} = 0$$

$$J_{2\sigma} = \frac{I_\sigma^2}{3} - II_\sigma = \frac{1}{6}\left[(\sigma_1-\sigma_2)^2 + (\sigma_2-\sigma_3)^2 + (\sigma_3-\sigma_1)^2\right]$$

$$J_{3\sigma} = I_\sigma - II_\sigma p + 2p$$

PROPERTY: octahedral plane



The octahedral plane is a plane which normal vector is forming equal angles with principal stress axes. In particular, the ~~cosin~~ director cosines with respect to the principal axes are equal to $\sqrt{3}/3$. On this plane, stress assume the following values.

$$\sigma_{oct} = \frac{1}{3}(\sigma_1 + \sigma_2 + \sigma_3) = p$$

$$\tau_{oct} = \frac{1}{3}\sqrt{(\sigma_1-\sigma_2)^2 + (\sigma_2-\sigma_3)^2 + (\sigma_1-\sigma_3)^2}$$

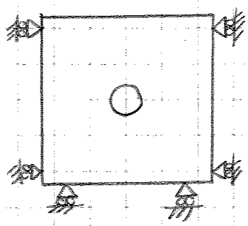
3 Solution of boundary value problems in continuous mechanics

In engineering, we search for displacement, strains and stresses; in mathematics, we look for a solution of a boundary value problem.

In mathematics, a boundary value problem is a differential equation together with a set of additional restraints, in numerical modelling called "boundary conditions"

B.V.P. = ODE / PDE + ADDITIONAL RESTRAINTS

Thus, the solution of a boundary value problem is the solution of a differential equation which also satisfies the additional restraints.



We need additional restraints, even because we are not able to model the infinite. In this case, as far away displacement is zero, we introduce a kinematic restraint, given by a roller.

→ stress - strain law (CONSTITUTIVE BEHAVIOUR)

In the problem, we have

6 stress components + 6 strain components + 3 displacement

= 15 unknowns

and only 9 equations.
So we need to consider the constitutive behaviour, i.e. the relation between stresses and strains.
A simple model is the isotropic linear elastic behaviour (ILE), in which we have a linear relation between stresses and strains and we have only 2 deformability parametres.
The constitutive equation is

$$\sigma_{ij} = C_{ijkl}\,\varepsilon_{kl}$$

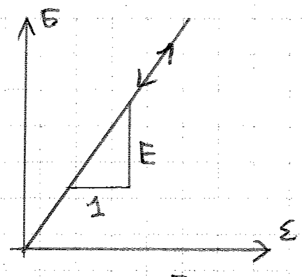
and gives us 6 equations, so we can go to the solution of the problem.

If we work with Young's modulus E and Poisson's ratio ν, we have

$$\begin{pmatrix}\varepsilon_x\\ \varepsilon_y\\ \varepsilon_z\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{xz}\end{pmatrix} = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & -\frac{\nu}{E} & 0 & 0 & 0\\ -\frac{\nu}{E} & \frac{1}{E} & -\frac{\nu}{E} & 0 & 0 & 0\\ -\frac{\nu}{E} & -\frac{\nu}{E} & \frac{1}{E} & 0 & 0 & 0\\ 0 & 0 & 0 & \frac{2(1+\nu)}{E} & 0 & 0\\ 0 & 0 & 0 & 0 & \frac{2(1+\nu)}{E} & 0\\ 0 & 0 & 0 & 0 & 0 & \frac{2(1+\nu)}{E}\end{bmatrix}\begin{pmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \tau_{xy}\\ \tau_{yz}\\ \tau_{xz}\end{pmatrix}$$

If we work with shear modulus G and bulk modulus k, we have

$$[C] = \begin{bmatrix} k+\frac{4}{3}G & k-\frac{2}{3}G & k-\frac{2}{3}G & 0 & 0 & 0\\ k-\frac{2}{3}G & k+\frac{4}{3}G & k-\frac{2}{3}G & 0 & 0 & 0\\ k-\frac{2}{3}G & k-\frac{2}{3}G & k+\frac{4}{3}G & 0 & 0 & 0\\ 0 & 0 & 0 & G & 0 & 0\\ 0 & 0 & 0 & 0 & G & 0\\ 0 & 0 & 0 & 0 & 0 & G\end{bmatrix}$$

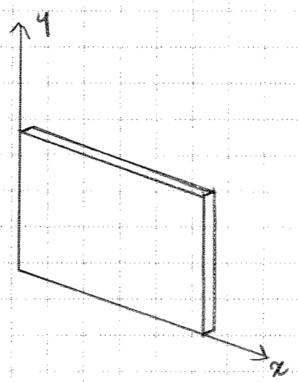


$$G = \frac{E}{2(1+\nu)}$$

$$k = \frac{E}{3(1-2\nu)}$$

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 23 di 218

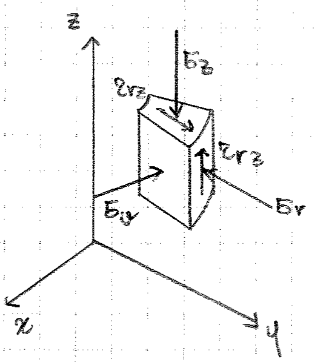
→ plane stress (rare in geotechnical problems), where we have no stress in the ~~perpen~~ orthogonal direction



$$[\sigma] = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

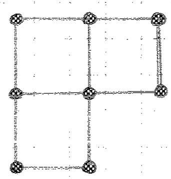
$$[C] = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

→ axi symmetry, when the behaviour is symmetric referring to an axis (single pile, tri axial sample, circular footing ...).
This allows a semplification, even if it's not much as before.



$$[\varepsilon] = \begin{pmatrix} \varepsilon_r \\ \varepsilon_\vartheta \\ \varepsilon_z \\ \gamma_{rz} \end{pmatrix} \qquad [\sigma] = \begin{pmatrix} \sigma_r \\ \sigma_\vartheta \\ \sigma_z \\ \tau_{rz} \end{pmatrix}$$

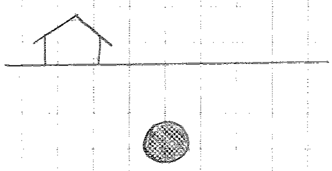
$$[C] = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 & \frac{\nu}{1-\nu} \\ \frac{\nu}{1-\nu} & 1 & 0 & \frac{\nu}{1-\nu} \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} & 0 \\ \frac{\nu}{1-\nu} & \frac{\nu}{1-\nu} & 0 & 1 \end{bmatrix}$$



We can not compute every property at each point, so we divide the domain within discrete elements, that interact in **nodes**.
Equations are solved for the nodes and use mathematical approximations to define the other values within the areas.

Principles of finite element method

Let's consider a practical case.

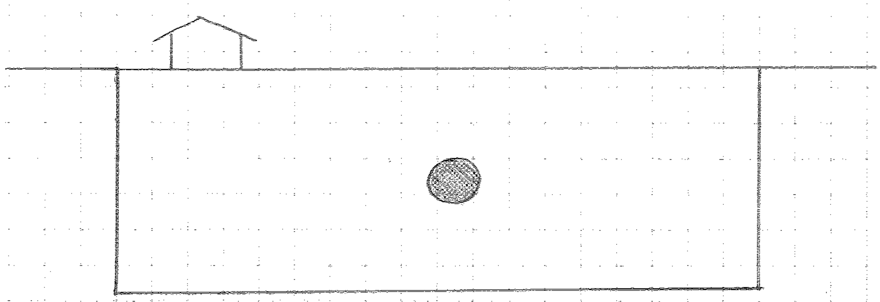


An homogeneous soil with horizontal ground surface has an house built on it.
We want to excavate a tunnel in it and use a numerical method to design it.

When we use finite element method (F.E.M.), we follow some steps.

(I) **Definition of the domain**: we have to find the boundaries.

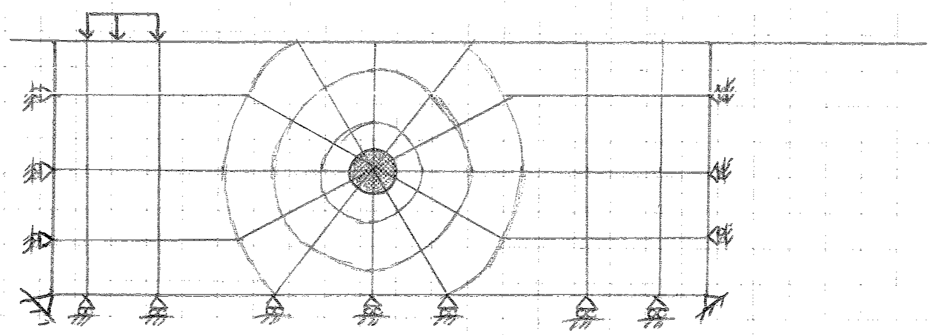
A natural boundary is the ground surface.
On the other sides, boundaries are theoretically at an infinite distance, but we can't model something which is infinite.
So, we have to set some boundaries and the simplest situation consists on considering a rectangular shape to define the domain.



(II) Element discretization (MESH)

Domain is discretized in a number of smaller regions, called finite **elements**.
This is a geometrical problem of sampling a region and a correct discretazion can optimise the process, reducing computation time and simplifing results.

We have many types of elements (quadrilateral, triangular ...) and, at the end, we obtain an assembly of finite elements, called **mesh**.



VII Solution of the global equations:

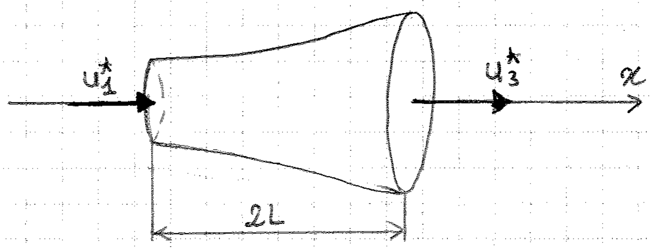
we obtain displacement at all the nodes and we derive strains and stresses as secundary quantities.

VIII Interpretation of the results

Finite element method in 1-dimensional problems

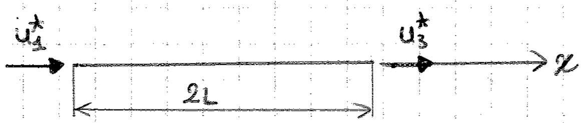
We apply the FEM algorythm to a simple situation, i.e. 1D problem:

we have a 1D object with variable cross section and two known displacements are applied to the edges of the beam.
Compute the axial stress in the bar, by using the F.E.M.



(I) DEFINITION OF THE DOMAIN

The domain has only one dimension and its length is $2L$.

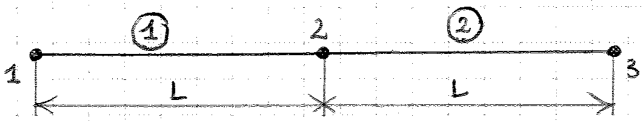


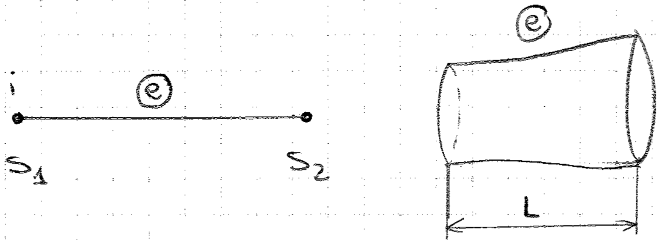
(II) ELEMENT DISCRETIZATION

The domain has to be divided into a number of finite elements.
In this case, we use 2 one-dimensional elements and have 2 nodes.
So, the finite element mesh has 2 elements and 3 nodes.

$$\text{Finite Element Mesh} \begin{cases} 2 \text{ elements} & ① \quad ② \\ 3 \text{ nodes} & 1 \quad 2 \quad 3 \end{cases}$$

The single generic element (e) has length L and the cross sections at nodes are S_1 and S_2.





$\{u\}_e$ = matrix of nodal displacements of the element ©

$[A]$ = matrix of coordinates of the nodes

Then we solve the vector $\{\alpha\}$ from the equation

$$\{\alpha\} = [A]^{-1}\{u\}_e$$

Coming back to the initial equation, we can have displacement in the elements

$$u(x) = \{\Phi\}^T\{\alpha\} =$$
$$= \{\Phi\}^T[A]^{-1}\{u\}_e = \{H\}^T\{u\}_e$$

The term H is the shape function, that represents element's geometry and shape.

$$\{H\}^T = \{\Phi\}^T[A]^{-1} =$$

$$= \begin{pmatrix} 1 & x \end{pmatrix} \begin{bmatrix} 1 & 0 \\ -\frac{1}{L} & \frac{1}{L} \end{bmatrix} = \begin{pmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{pmatrix}$$

$$\{H\}^T = \begin{pmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{pmatrix}$$

In this way, displacement within the element is function of NODAL DISPLACEMENTS and SHAPE of the element.

$$u(x) = \{H\}^T\{u\}_e = \begin{pmatrix} 1-\frac{x}{L} & \frac{x}{L} \end{pmatrix} \begin{pmatrix} u_i \\ u_j \end{pmatrix}$$

Then, we have to compute axial strain that, in 1D problems, is calculated through the following equation.

$$\varepsilon_x = \frac{du(x)}{dx} =$$

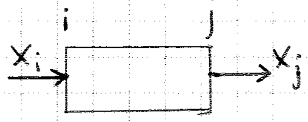
$$= \frac{d}{dx}\{\Phi\}^T[A]^{-1}\{u\}_e =$$

$$= \{\Phi'\}^T[A]^{-1}\{u\}_e = \{B\}\{u\}_e$$ → only Φ depends on x

At this point, we introduce a RELATIONSHIP BETWEEN NODAL FORCES X AND NODAL DISPLACEMENTS u_e within the single element.

$$\{X\}_e = [k]_e \{u\}_e \qquad k = \text{stiffness matrix}$$

In this case, we have 2 nodes and the relation is



$$\begin{pmatrix} X_i \\ X_j \end{pmatrix} = \begin{bmatrix} k_{ii} & k_{ij} \\ k_{ij} & k_{jj} \end{bmatrix} \begin{pmatrix} u_i \\ u_j \end{pmatrix}$$

How to define stiffness matrix?

It is something related to deformability and we apply the virtual work principle

Internal virtual work (strain energy) = External work, due to nodal forces and displacements

$$\int_{V_e} \{\bar{\varepsilon}\}_e^T \{\sigma\}_e \, dV_e = X_i \bar{u}_i + X_j \bar{u}_j$$

The kinematic contributes (u and ε) are overscored because they're virtual and not real.

We focus on single terms.

→ external virtual work

$$X_i \bar{u}_i + X_j \bar{u}_j = (\bar{u}_i \quad \bar{u}_j) \begin{pmatrix} X_i \\ X_j \end{pmatrix} = \{\bar{u}\}_e^T \{X\}_e$$

→ internal virtual work

generally, B is a matrix

$$\int_{V_e} \{\bar{\varepsilon}\}_e^T \{\sigma\}_e \, dV_e = \int_{V_e} [B]^T \{\bar{u}\}_e^T \cdot [C][B]\{u\}_e \, dV_e$$

We can take the virtual nodal displacement out of the integral.

$$\int_{V_e} \{\bar{\varepsilon}\}_e^T \{\sigma\}_e \, dV_e = \{\bar{u}\}_e^T \int_{V_e} [B]^T [C][B] \, dV_e \cdot \{u\}_e$$

$$[k]_e = \frac{S_e E}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$ Stiffness matrix for 1D element

In stiffness matrix, we can see geometry (cross section S_e) and deformability parametres (Young's modulus E).
Using this equation, we can fill the stiffness matrix with the single components.

$$k_{ii} = k_{jj} = \frac{S_e E}{L} \qquad k_{ij} = k_{ji} = -\frac{S_e E}{L}$$

Now, we have all the element's equations, ~~which~~ that include displacements, strains, stresses and nodal forces.

V) WRITING OF THE GLOBAL EQUATIONS

We have to assembly the separate element equations into a set of global equations by using the stiffness matrix.

Firstly, we write the relationship nodal forces - nodal displacements referring to all the mesh

$$\{X\} = [k]\{u\}$$

$$\begin{pmatrix} X_1 \\ X_2 \\ X_3 \end{pmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} \\ k_{12} & k_{22} & k_{23} \\ k_{13} & k_{23} & k_{33} \end{bmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \end{pmatrix}$$

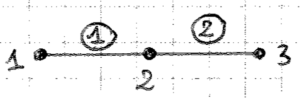
the mesh has 3 nodes

global stiffness matrix

We have just computed the stiffness matrix for the generic element.
Now, we need to define how the stiffness components of the single element will go to the global matrix.
In other words, from each single component, we'll built global matrix and we use the direct stiffness method.

This process is applied element by element.

Then we superimpose the 2 conditions and the global stiffness matrix of the mesh will have the sum of the components.



$$\begin{pmatrix} X_1 \\ X_2 \\ X_3 \end{pmatrix} = \begin{bmatrix} \frac{S_1 E}{L} & -\frac{S_1 E}{L} & 0 \\ -\frac{S_1 E}{L} & \frac{S_1 E}{L} + \frac{S_2 E}{L} & -\frac{S_2 E}{L} \\ 0 & -\frac{S_2 E}{L} & \frac{S_2 E}{L} \end{bmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \end{pmatrix}$$

Compared to the contributions, global stiffness matrix has only one different component, given by the sum of the 2 contributions.
This component represents the stiffness related to node 2, that belongs to 2 elements and has the contribution from both them.

VI) APPLICATION OF THE BOUNDARY CONDITIONS

We have two known displacements applied at the external nodes

$$u_1 = u_1^*$$

$$u_3 = u_3^*$$

The problem presents 6 unknowns (3 nodal forces + 3 nodal displacements) and 3 equations. With these boundary conditions, we add 2 other equations.
So, we have to add a further condition:

do we know something about forces?

At the external nodes, as we apply a displacement, we have a reaction which isn't known.
At node 2, as it is a internal node, in terms of external forces, we have

$$X_2 = 0$$

because we have a component on one side and a component on the other side that are opposite.

In this way, we have a 3x3 system that we can solve.

$$\begin{array}{c} ⓐ \\ ⓑ \\ ⓒ \end{array} \quad \begin{pmatrix} X_1 \\ 0 \\ X_3 \end{pmatrix} = \begin{bmatrix} \frac{S_1 E}{L} & -\frac{S_1 E}{L} & 0 \\ -\frac{S_1 E}{L} & \frac{S_1 E}{L} + \frac{S_2 E}{L} & -\frac{S_2 E}{L} \\ 0 & -\frac{S_2 E}{L} & \frac{S_2 E}{L} \end{bmatrix} \begin{pmatrix} u_1^* \\ u_2 \\ u_3^* \end{pmatrix}$$

→ element 2

$$\varepsilon_2 = \{B\}\{u\}_2 =$$

$$= \begin{pmatrix} -\frac{1}{L} & \frac{1}{L} \end{pmatrix} \begin{pmatrix} u_2 \\ u_3 \end{pmatrix} =$$

$$= -\frac{u_2}{L} + \frac{u_3}{L} =$$

$$= -\frac{1}{L} \cdot \frac{S_1 u_1^* + S_2 u_3^*}{S_1 + S_2} + \frac{u_3^*}{L} = \frac{S_1}{S_1 + S_2} \, \frac{u_3^* - u_1^*}{L}$$

$$\Rightarrow \sigma_2 = E \varepsilon_2$$

We can see that strains and stresses within the 2 elements are different and they are equal only if the elements have the same cross section.

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 41 di 218

Then, if the domain represents the ground, we have self-weight.
So, we have a BODY-FORCE, with 3 components.

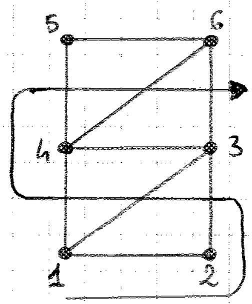
$$\{f^V\} = \begin{pmatrix} f_x^V \\ f_y^V \\ f_z^V \end{pmatrix}$$

2 Now, we see the FEM method's steps in a 2D-3D problem (we focus on 2D examples, in order to simplify).

I DEFINITION OF THE DOMAIN

II ELEMENT DISCRETIZATION

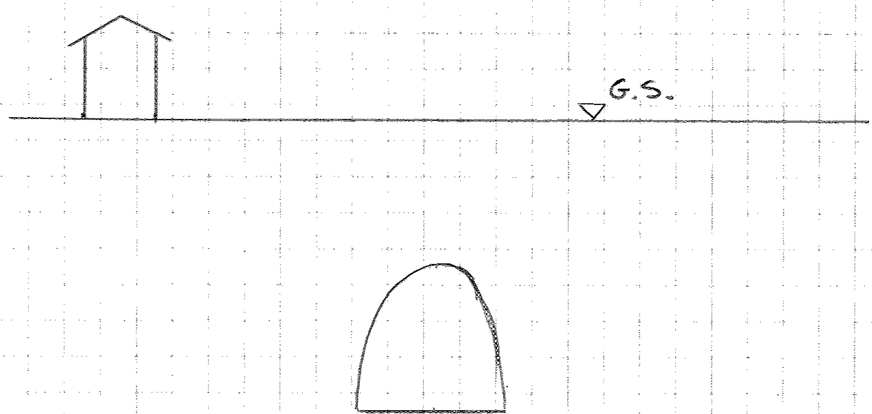
We discretize domain into a number of small elements, that will interact one with the other one thanks to the nodes.



Elements and nodes must be NUMBERED IN A SYSTEMATIC MANNER, following a certain path.
This aspect is relevant as the way we number the mesh influences the computational processe and a clever way of discretization can reduce computation time.

In 3D problems, building the mesh becomes an important aspect as it influences the quality of the results.
How can we BUILD THE MESH?

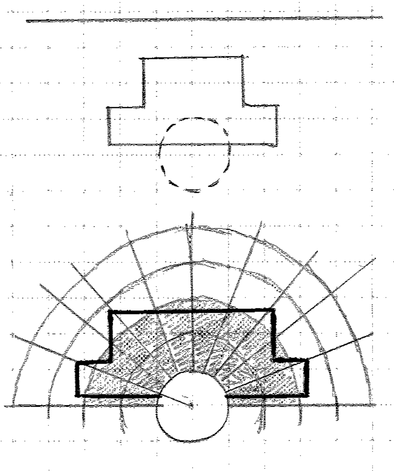
Let's consider a practical case: we want to excavate an horse-shoe shape tunnel in a ground. On its surface, there is a house.



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 43 di 218

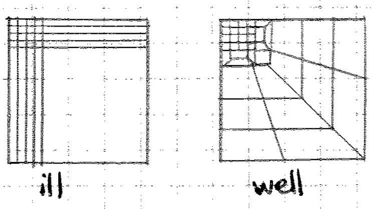
→ non homogeneous zones (e.g. separation between material layers)

When we have non homogeneous zones, we should put nodes along the contours from one zone to the other one. In this way, we'll not have problems in assigning parametres to the elements.



Sometimes, we can't follow this rule because it is too much complicated.
For instance, if we have a tunnel, we build a certain mesh in order to follow the shape and have quadrilateral elements.
Sometimes, we inject grout to consolidate the upper arch and have a reinforced arch after the excavation.
Yet, if we want to simulate this in a numerical method, the mesh doesn't fit the geometry of the problem very well as the contours go through the elements.
If we don't want to change the mesh, we can apply material properties of consolidated ground to some elements and build geometry in order to resemble it as much as we can.

→ we need denser mesh where high stress gradients are expected.
In this way, by increasing the size of elements while we go faraway from the problem, we can make bigger models as we have less elements (we optimise geometry).



We have to use SMOOTH TRANSITION from the small elements to the big ones.
If this condition is satisfied, we tell ~~abou~~ "well conditioned mesh". Otherwise, we have an "ill conditioned mesh".

Software are helpful to design the mesh, as it has authomatic mesh and gives suggestions on how to build it.

Nodal displacements are referred to as the unknown degrees of freedom and, for 2D problems, each node has 2 degrees of freedom

→ u displacement, in the x coordinate direction

→ v displacement, in the y coordinate direction

Then, we use element equations to derive stresses and strains within each element. These equations combine the compatibility conditions, the equilibrium and constitutive equations.

Strain state is given by the same equation used in 1D problem, but now we have more components.

$$\{\varepsilon\} = [B]\{u\}_e$$

Then, we apply constitutive equations, assuming an ILE behaviour, and we have stresses.

$$\{\sigma\} = [C]\{\varepsilon\}$$

Is convenient to rewrite this equation in a more general form, in order to take in account the conditions that we may have when we study the problem.

$$\{\sigma\} = [C]\left(\{\varepsilon\} - \{\varepsilon\}_i\right) + \{\sigma\}_i$$

$\{\sigma\}_i$ = INITIAL STRESS STATE MATRIX

At the beginning, we have no stress only if we are modelling a tri-axial test specimen.
Actually, we have to take in account of the lithostatic stress γz.
Moreover, if we are next to the ground surface, we feel the effect of external loads - these are not felt at high depth and here we come back to lithostatic stress.

⇒ we need to do a preliminary study to determine INITIAL CONDITIONS

$\{\varepsilon\}_i$ = INITIAL STRAIN MATRIX, less common (maybe due to thermal effects).

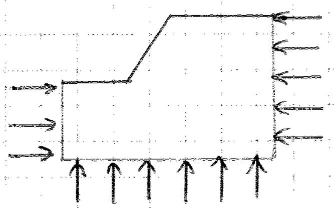
The global stiffness matrix is the sum of the elements' contribution.

$$[k] = \sum_e \int_{V_e} [B]_e^T [C]_e [B]_e \, dV_e$$

VI APPLICATION OF THE BOUNDARY CONDITIONS

We can have 3 different types of boundary conditions

→ **loading conditions**: this is the case of line loads and surcharge pressures.

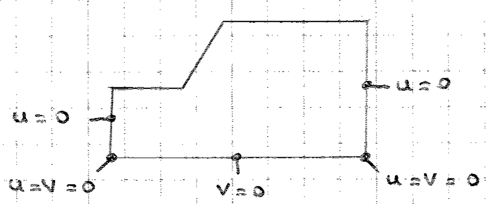


If ~~noda~~ point forces are prescribed, these can be assembled directly into the nodal forces; if we have other kinds of forces (distributed loads), these must be expressed as equivalent nodal forces.

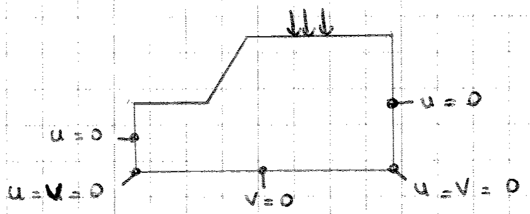
↓ we know stresses at the boundaries

→ **displacement boundary conditions** (the corresponding global equation can be eliminated during the solution process)

→ **loading and displacement conditions** (typical situation for tunnels and slope stability problems)



Displacement boundary conditions



Loading and displacement conditions

We can choose displacements or stresses as boundary condition.
If we choose displacements, model is going to settle.
If we choose stresses, we have to adopt an adequate distribution law (it is not always constant as the ground surface is not horizontal) and, in gravity, superior points move more than the other ones. So, we have to put rollers to allow consolidation and the model is going to settle again.

VII SOLUTION OF THE GLOBAL EQUATIONS

Global equations present a number of equations and unknowns N given by the relation

$$N = n \cdot \rho$$

n = number of nodes $\qquad$ ρ = number of unknowns per node

To solve them, we can use direct or iterative mathematical techniques.

$$\begin{pmatrix} u_i \\ u_j \\ u_k \\ v_i \\ v_j \\ v_k \end{pmatrix} = \begin{bmatrix} 1 & x_i & y_i & 0 & 0 & 0 \\ 1 & x_j & y_j & 0 & 0 & 0 \\ 1 & x_k & y_k & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_i & y_i \\ 0 & 0 & 0 & 1 & x_j & y_j \\ 0 & 0 & 0 & 1 & x_k & y_k \end{bmatrix} \begin{pmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{pmatrix}$$

$$\downarrow$$

$$\{u\}_e = [A]\{\alpha\}$$

In this way, we have A matrix.

Next step is inverting this matrix and compute displacement in the generic point in function of nodal displacements.

$$\underset{\text{displacement } 2\times 1}{\{u(x)\}} = \underbrace{[\phi]^T [A]^{-1}}_{\text{shape function } H, \text{ that contains geometry } 2\times 6} \underset{\text{nodal displacement } 6\times 1}{\{u\}_e}$$

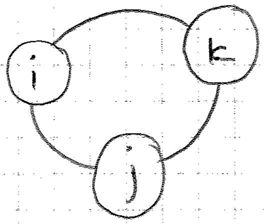
In particular, we obtain

$$u(x;y) = \frac{1}{2A}\left[(a_i + b_i x + c_i y)u_i + (a_j + b_j x + c_j y)u_j + (a_k + b_k x + c_k y)u_k\right]$$

$$v(x;y) = \frac{1}{2A}\left[(a_i + b_i x + c_i y)v_i + (a_j + b_j x + c_j y)v_j + (a_k + b_k x + c_k y)v_k\right]$$

where

$$a_i = x_j y_k - x_k y_j \qquad b_i = y_j - y_k \qquad c_i = x_k - x_j$$



The remaining coefficients $a_j, b_j, \dots b_k$ and c_k are obtained by a cyclic permutation of the subscripts in the order i, j and k

The strain in the generic point depends on nodal displacements by means of some constant terms

$\Rightarrow$ matrix B components don't depend on the position of the point within the element and it means that STRAIN IS CONSTANT IN ALL POINTS INSIDE THE ELEMENT.

For this reason, triangular elements are called constant strain triangular elements (CST).

Moreover, as each element has a constant strain, strain is not continuous and has discrete variations passing from an element to the other one.
Big elements imply big jumps in strain and it's not a good thing, so denser mesh is required where we expect high stress gradients.

Then, we pass to stresses by using the constitutive law (linear elasticity, in this case).

$$\{\sigma\} = [C]_e \{\varepsilon\} =$$
$$= [C]_e [B]_e \{u\}_e$$

If the element is homogeneous - in a good ripartition, it is -, also stress is constant within the specific element.

$\Rightarrow$ STRAIN AND STRESS ARE CONSTANT WITHIN EACH TRIANGULAR ELEMENT

Then we
Then we build the element equations, by computing the stiffness matrix.

$$k_e = \int_{V_e} [B]_e^T [C]_e [B]_e \, dV$$

In CST elements, B and C matrix are constant in the element and volume can be expressed in terms of the area and the thickness of the element - as it is a 2D element.

$$k_e = [B]_e^T [C]_e [B]_e \int_{V_e} dV =$$
$$= [B]_e^T [C]_e [B]_e \, At$$

$$[k]_e = [B]_e^T [C]_e [B]_e \, At$$ Stiffness matrix of a CST element

Then, we compute stiffness matrix

$$[k]_e = [B]_e^T [C]_e [B]_e A t$$

We assume a unit stiffness and use the linear elastic constitutive law

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} = \frac{1-\nu}{(1+\nu)(1-2\nu)} E \begin{bmatrix} 1 & \frac{1}{1-\nu} & 0 \\ \frac{1}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{pmatrix}$$

↳ constitutive law in plain strain (in 2D problems)

$$[k]_e = \frac{1}{2\sqrt{3}} \begin{bmatrix} -\sqrt{3} & 0 & -1 \\ 0 & -1 & -\sqrt{3} \\ \sqrt{3} & 0 & -1 \\ 0 & -1 & \sqrt{3} \\ 0 & 0 & 2 \\ 0 & 2 & 0 \end{bmatrix} \cdot \frac{(1-\nu)E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{1}{1-\nu} & 0 \\ \frac{1}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix} \cdot$$

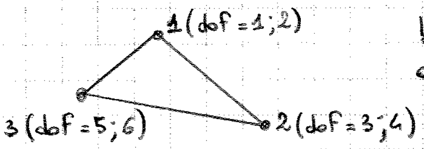
$$\cdot \frac{1}{2\sqrt{3}} \begin{bmatrix} -\sqrt{3} & 0 & \sqrt{3} & 0 & 0 & 0 \\ 0 & -1 & 0 & -1 & 0 & 2 \\ -1 & -\sqrt{3} & -1 & \sqrt{3} & 2 & 0 \end{bmatrix} \cdot \sqrt{3} \cdot 1 =$$

$$= \frac{(1-\nu)E}{4\sqrt{3}(1+\nu)(1-2\nu)} \begin{bmatrix} -\sqrt{3} & 0 & -1 \\ 0 & -1 & -\sqrt{3} \\ \sqrt{3} & 0 & -1 \\ 0 & -1 & \sqrt{3} \\ 0 & 0 & 2 \\ 0 & 2 & 0 \end{bmatrix} \begin{bmatrix} -\sqrt{3} & -\frac{1}{1-\nu} & \sqrt{3} & -\frac{1}{1-\nu} & 0 & \frac{2}{1-\nu} \\ -\frac{\sqrt{3}}{1-\nu} & -1 & \frac{\sqrt{3}}{1-\nu} & -1 & 0 & 2 \\ -\frac{1-2\nu}{2(1-\nu)} & -\frac{\sqrt{3}}{2}\frac{1-2\nu}{1-\nu} & -\frac{1-2\nu}{2(1-\nu)} & \frac{\sqrt{3}}{2}\frac{1-2\nu}{1-\nu} & \frac{1-2\nu}{1-\nu} & 0 \end{bmatrix} =$$

Then, we pass in computing global equations.

$$[k]_G \{u\}_G = \{R\}_G$$

We have to assembly stiffness matri es of each element and build a global stiffness matrix.



In a triangular element, we have 3 nodes for a total of 6 degrees of freedom (dof).

Global stiffness matrix can be written in terms of degrees of freedom and we have a 6 x 6 matrix.

$$[k]_G = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} \rightarrow$$

indices are referred to the degrees of freedom

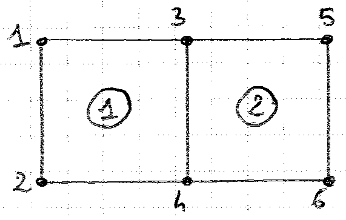
In order to obtain the global stiffness matrix, it is useful to use ~~the~~ a partitioned matrix, in which each element represents 4 different components - a 2x2 matrix - and its indices are referred to the nodes.

$$[k]_G = \left[\begin{array}{cc|cc|cc} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ \multicolumn{2}{c|}{K_{11}} & \multicolumn{2}{c|}{K_{12}} & \multicolumn{2}{c}{K_{13}} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ \hline k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ \multicolumn{2}{c|}{K_{21}} & \multicolumn{2}{c|}{K_{22}} & \multicolumn{2}{c}{K_{23}} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ \hline k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ \multicolumn{2}{c|}{K_{31}} & \multicolumn{2}{c|}{K_{32}} & \multicolumn{2}{c}{K_{33}} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{array}\right] = \begin{bmatrix} K_{11} & K_{12} & K_{13} \\ K_{21} & K_{22} & K_{23} \\ K_{31} & K_{32} & K_{33} \end{bmatrix}$$

$\downarrow$

indices are referred to the nodes

This method is general and can be applied to other problems, like quadrangular elements.



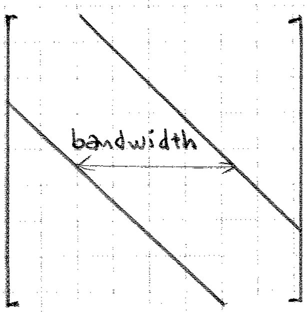
In this case, element stiffness matrices are 4x4 and, in writing the global equations, we have some locations on which there is superimposition → and sum contributions.

$$\begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} = \begin{bmatrix} k^1_{11} & k^1_{12} & k^1_{13} & k^1_{14} & 0 & 0 \\ & k^1_{22} & k^1_{23} & k^1_{24} & 0 & 0 \\ & & k^1_{33}+k^2_{33} & k^1_{34}+k^2_{34} & k^2_{35} & k^2_{36} \\ & & & k^1_{44}+k^2_{44} & k^2_{45} & k^2_{46} \\ & \text{symm} & & & k^2_{55} & k^2_{56} \\ & & & & & k^2_{66} \end{bmatrix}$$

In conclusion, we can observe some properties of global equations

→ THE GLOBAL STIFFNESS MATRIX IS SYMMETRIC and the non zero components arise only from the connection between degrees of freedom through the elements.
Thus, for each row, the last nonzero element corresponds to the highest degree of freedom to which the degree of freedom represented by the raw is connected.

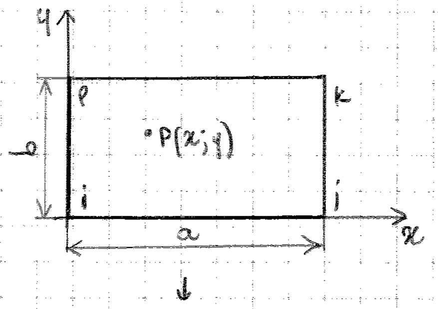
→ THE GLOBAL STIFFNESS MATRIX IS generally SPARSE - has many zero terms - AND BANDED - the non-zero components are concentrated along the diagonal.



So, we can define the bandwidth, that is the part of the matrix which has non-zero elements.
This width depends on MESH'S NUMERATION and this aspect is fundamental because in computers we store only the non-zero elements (usually, the diagonal and upper triangle, due to symmetry) and, with a small bandwidth, we have to store less data.

⇒ we have to choose a numbering in order to minimize bandwidth (for instance, I should avoid that, in the same element, 2 nodes have 2 very different numbers)

Adopting displacement as primary variable, we can write the element equations.



the sides of the element have the same direction as xy axes

Considering one generic QUAD element, we have to define how the primary variable varies within the element.
So, we approximate displacement through a polynomial form, which order is given by the number of nodes.

$$u(x;y) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 xy$$

$$v(x;y) = \alpha_5 + \alpha_6 x + \alpha_7 y + \alpha_8 xy$$

The relationship can be rewritten in matricial form.

$$\begin{pmatrix} u(x;y) \\ v(x;y) \end{pmatrix} = \begin{bmatrix} 1 & x & y & xy & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & x & y & xy \end{bmatrix} \begin{pmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \\ \alpha_7 \\ \alpha_8 \end{pmatrix}$$

$$\{u(x)\} = [\phi]^T \{\alpha\} \quad (1)$$

Same equation of 1D element

Then, the equation is written in relation to nodal points and the relative displacements.

$$\begin{cases} u_i = \alpha_1 + \alpha_2 x_i + \alpha_3 y_i + \alpha_4 x_i y_i = \alpha_1 \\ v_i = \alpha_5 + \alpha_6 x_i + \alpha_7 y_i + \alpha_8 x_i y_i = \alpha_5 \\ u_j = \alpha_1 + \alpha_2 x_j + \alpha_3 y_j + \alpha_4 x_j y_j = \alpha_1 + a\alpha_2 \\ v_j = \alpha_5 + \alpha_6 x_j + \alpha_7 y_j + \alpha_8 x_j y_j = \alpha_5 + a\alpha_6 \\ u_k = \alpha_1 + \alpha_2 x_k + \alpha_3 y_k + \alpha_4 x_k y_k = \alpha_1 + a\alpha_2 + b\alpha_3 + ab\alpha_4 \\ v_k = \alpha_5 + \alpha_6 x_k + \alpha_7 y_k + \alpha_8 x_k y_k = \alpha_5 + a\alpha_6 + b\alpha_7 + ab\alpha_8 \\ u_\ell = \alpha_1 + \alpha_2 x_\ell + \alpha_3 y_\ell + \alpha_4 x_\ell y_\ell = \alpha_1 + b\alpha_3 \\ v_\ell = \alpha_5 + \alpha_6 x_\ell + \alpha_7 y_\ell + \alpha_8 x_\ell y_\ell = \alpha_5 + b\alpha_7 \end{cases}$$

The shape function is the following

$$[H(x;y)]^T = [\Phi(x;y)]^T \; [A]^{-1}$$

$$= \begin{bmatrix} 1 & x & y & xy & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & x & y & xy \end{bmatrix} \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ -\frac{1}{a} & \frac{1}{a} & 0 & 0 & 0 & 0 & 0 & 0 \\ -\frac{1}{b} & 0 & 0 & \frac{1}{b} & 0 & 0 & 0 & 0 \\ \frac{1}{ab} & -\frac{1}{ab} & \frac{1}{ab} & -\frac{1}{ab} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{1}{a} & \frac{1}{a} & 0 & 0 \\ 0 & 0 & 0 & 0 & -\frac{1}{b} & 0 & 0 & \frac{1}{b} \\ 0 & 0 & 0 & 0 & \frac{1}{ab} & -\frac{1}{ab} & \frac{1}{ab} & -\frac{1}{ab} \end{bmatrix}$$

$$= \begin{bmatrix} 1-\frac{x}{a}-\frac{y}{b}+\frac{xy}{ab} & \frac{x}{a}-\frac{xy}{ab} & \frac{xy}{ab} & \frac{y}{b}-\frac{xy}{ab} & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-\frac{x}{a}-\frac{y}{b}+\frac{xy}{ab} & \frac{x}{a}-\frac{xy}{ab} & \frac{xy}{ab} & \frac{y}{b}-\frac{xy}{ab} \end{bmatrix}$$

$$= \frac{1}{ab} \begin{bmatrix} (a-x)(b-y) & x(b-y) & xy & y(a-x) & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & (a-x)(b-y) & x(b-y) & xy & y(a-x) \end{bmatrix}$$

In the shape function, we have the element's dimensions a and b and the variables x and y connected to ~~the~~ geometry.

To write the element stiffness matrix, it is necessary to write strains in the element.
As the element has 2 dimensions, the components of strain state are

$$\varepsilon_x = \frac{\partial u}{\partial x} \qquad \varepsilon_y = \frac{\partial v}{\partial y} \qquad \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

The strain vector will be

$$\{\varepsilon\} = [B]\{u\}_e$$

where B is the strain matrix, given by the expression

Then, we can compute element stiffness matrix, by using the relationship

$$[k]_e = \int_A [B]^T [C] [B]\, t \, dx\, dy$$

In plane strain condition, thickness t is equal to ~~1~~ and the constitutive law is the following

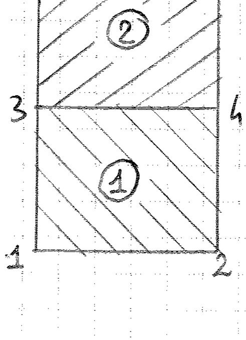
$$[C] = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1 & \frac{\nu}{1-\nu} & 0 \\ \frac{\nu}{1-\nu} & 1 & 0 \\ 0 & 0 & \frac{1-2\nu}{2(1-\nu)} \end{bmatrix}$$

The element stiffness matrix is

$$[k]_e = \frac{tF}{ab} \begin{bmatrix} \frac{1}{3}(b^2+c_3a^2) & \frac{1}{3}\left|c_3\frac{a^2}{2}-b^2\right| & -\frac{1}{6}(b^2+c_3a^2) & \frac{1}{3}\left|\frac{b^2}{2}-c_3a^2\right| & (m+c_3)\frac{ab}{4} & (m-c_3)\frac{ab}{4} & -(m+c_3)\frac{ab}{4} & -(m+c_3)\frac{ab}{4} \\ & \frac{1}{3}(b^2+c_3a^2) & \frac{1}{3}\left|\frac{b^2}{2}-c_3a^2\right| & -\frac{1}{6}(b^2+c_3a^2) & (c_3-m)\frac{ab}{4} & -(m+c_3)\frac{ab}{4} & (m-c_3)\frac{ab}{4} & (c_3-m)\frac{ab}{4} \\ & & \frac{1}{3}(b^2+c_3a^2) & \frac{1}{3}\left|c_3\frac{a^2}{2}-b^2\right| & -(m+c_3)\frac{ab}{4} & (m+c_3)\frac{ab}{4} & (m+c_3)\frac{ab}{4} & (m+c_3)\frac{ab}{4} \\ & & & \frac{1}{3}(b^2+c_3a^2) & (m-c_3)\frac{ab}{4} & (c_3-m)\frac{ab}{4} & (c_3-m)\frac{ab}{4} & -(m+c_3)\frac{ab}{4} \\ & & & & \frac{1}{3}(b^2+c_3a^2) & \frac{1}{3}\left|\frac{a^2}{2}-c_3b^2\right| & -\frac{1}{6}(a^2+c_3b^2) & \frac{1}{3}\left|c_3\frac{b^2}{2}-a^2\right| \\ & \text{symm} & & & & \frac{1}{3}(a^2+c_3b^2) & \frac{1}{3}\left|c_3\frac{b^2}{2}-a^2\right| & -\frac{1}{6}(a^2+c_3b^2) \\ & & & & & & \frac{1}{3}(a^2+c_3b^2) & \frac{1}{3}\left|\frac{a^2}{2}-c_3b^2\right| \\ & & & & & & & \frac{1}{3}(a^2+c_3b^2) \end{bmatrix}$$

$$F = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)} \qquad m = \frac{\nu}{1-\nu} \qquad c_3 = \frac{1-2\nu}{2(1-\nu)}$$

Then, we write global equations in symbolic form.



In terms of numbering of the nodes, there are 6 nodes and so we expect a 6×6 matrix - this is a partitioned matrix.

In general, it ~~wi~~ assumes the following form.

To finish the problem, it is necessary to focus on boundary conditions, especially on loading conditions.
We can have

→ POINT NODAL FORCES $\{R\}$;

→ BODY NODAL FORCES:

equivalent nodal forces are calculated through the expression

$$\{R\}_v = \int_S [H]^T \{f\}^v ds$$

→ SURFACE NODAL FORCES:

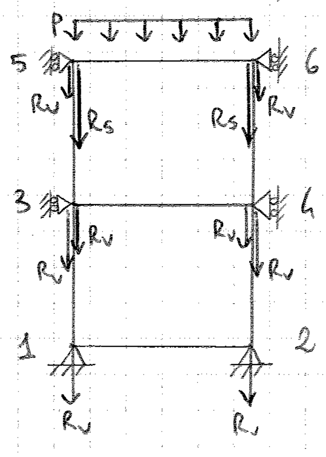
equivalent forces at the nodes are calculated through the expression

$$\{R\}_S = \int_S [H]^T \{f\}^S ds$$

Body forces and surface forces are divided as nodal forces because the solution of the problem is given by the equation

$$\{X\} = [k]\{u\}_e$$

and, in order to have nodal displacements, we have to identify the forces X at the nodes.
In the conversion to nodal forces, the shape function H is involved because in this problem geometry has a role.



Thus, there is a certain repartition of surface forces and body forces

→ pressure p is distributed only to 2 nodes - the nodes where it is applied.

→ the body force is applied on all elements, so it is divided among all nodes.

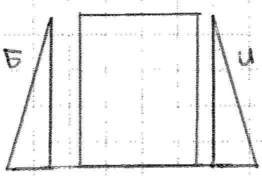
So

→ SELF WEIGHT ρg

$$\{R\}_V = \int_V [H]^T \{\rho g\}\, dV =$$

$$= \int_V \frac{1}{ab} \begin{bmatrix} (a-x)(b-y) & 0 \\ x(b-y) & 0 \\ xy & 0 \\ y(a-x) & 0 \\ 0 & (a-x)(b-y) \\ 0 & x(b-y) \\ 0 & xy \\ 0 & y(a-x) \end{bmatrix} \begin{pmatrix} 0 \\ \rho g \end{pmatrix} t\, dx\, dy =$$

$$= \int_V \frac{1}{ab} \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ (a-x)(b-y)\rho g \\ x(b-y)\rho g \\ xy\rho g \\ y(a-x)\rho g \end{pmatrix} t\, dx\, dy = \begin{pmatrix} 0 \\ 0 \\ 0 \\ 0 \\ \rho g t \frac{ab}{4} \\ \rho g t \frac{ab}{4} \\ \rho g t \frac{ab}{4} \\ \rho g t \frac{ab}{4} \end{pmatrix}$$



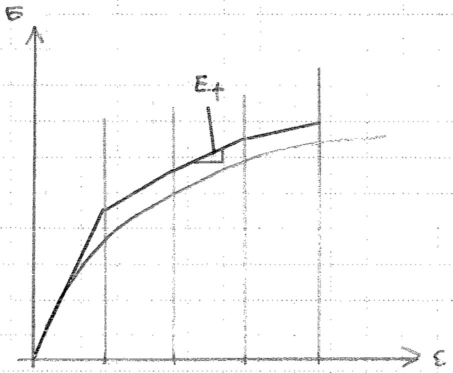
At the end, stresses and displacements will not be uniform - even if a uniform load is applied - but vary in a linear way, due to the presence of the body force.

Practically, ANY GEOTECHNICAL PROBLEM INVOLVES A NONLINEAR BEHAVIOUR, unless the stress state is very low - rare situation.

Iterative methods

1 Iterative methods represent a way to modify FEM in order to consider nonlinearity.

The main principle of these methods is adopting a multilinear model to describe the constitutive behaviour.



The constitutive law is divided into a number of segments in order to follow as possible the real behaviour.
In this way, the constitutive law is described in incremental terms and, for each increment, a different constitutive law is assumed.
By consequence, for each increment, we have a different matrix - that depends on elastic constants.

$$\{\Delta\sigma\} = [C_t]\{\Delta\varepsilon\} \quad \text{or} \quad \{\Delta\sigma\} = [C_s]\{\Delta\varepsilon\}$$

C_t is defined by tangent parameters E_t, ν_t
C_s is defined by secant parameters E_s, ν_s

In iterative methods, the nonlinearity of stress-strain law causes the governing FEM equations

$$[k]_G\{u\}_G = \{R\}_G \quad \rightsquigarrow \text{global equation}$$

to be reduced to a INCREMENTAL FORM OF FINITE ELEMENT GLOBAL EQUATIONS.

$$[k]_G^i\{\Delta u\}_G^i = \{\Delta R\}_G^i$$

Δu_G^i = vector of incremental nodal displacements
ΔR_G^i = vector of incremental nodal forces
k_G^i = incremental global stiffness matrix
i = increment number

These equations are applied for each increment and the final solution - final displacement - will be obtained by summing the results of each increment.

The presence of this error has not to be interpreted as a problem in all situations:

for instance, if the applied load R_{ap} is equal to 1 MPa and the unbalanced load is 0,1 Pa, the error is small compared to load and the given displacement is a satisfying result.

→ the main parameter that governs iterations is the ratio unbalanced load - applied load.

If the ratio between these quantities is not adequate, we can fix at the displacement u_1 and compute a new stiffness matrix k_1

obviously, the computation is made element by element because the stress state may be different and it would mean different Young modulus and different element stiffness matrix. Then, these matrices are combined into a global stiffness matrix.

Then, global equations are applied one more time.

$$\{u\}_2 = [k]_1^{-1} \{R\}_{ap} \qquad \rightarrow k_1 \neq k_0$$

Now, we get a different displacement and we still have a difference with respect to the constitutive law, but the unbalanced ~~law~~ load is smaller.

If we want to continue, we apply the same process and, with the i-th iteration, the displacement is

$$\{u\}_i = [k]_i^{-1} \{R\}_{ap}$$

and the unbalanced load is smaller.

When the iterative method should be stopped?

We should stop it when THERE IS NO MUCH DIFFERENCE FROM ONE ITERATION TO THE NEXT ONE

→ in terms of displacements

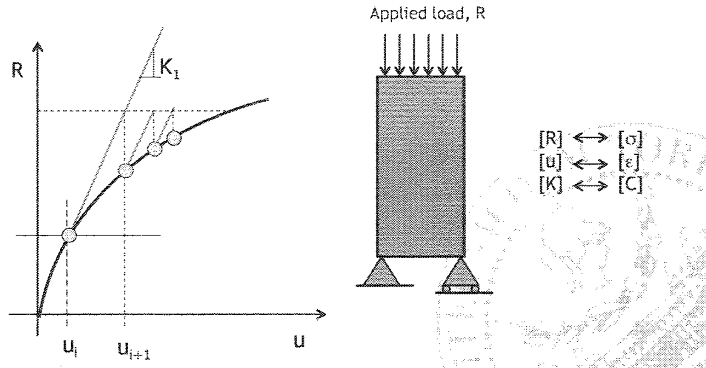
$$u_n - u_{n-1} \sim 0$$

→ in terms of unbalanced load

$$\Delta R_n - \Delta R_{n-1} \sim 0$$

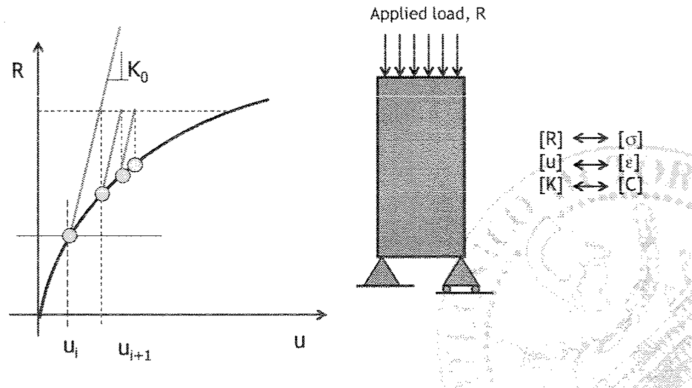
MODIFIED N-R METHOD

With initial tangent in increment.



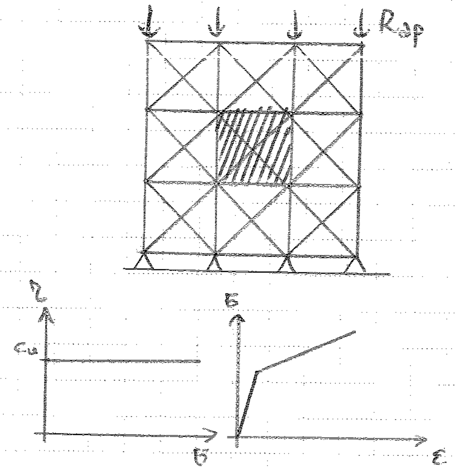
MODIFIED N-R METHOD

With initial problem tangent.



In the context of solid mechanics the method is also known as the STRESS TRANSFER or INITIAL STRESS APPROACH.

Ex (APPLICATION OF ITERATIVE METHODS : ELASTO-PLASTIC MATERIAL)



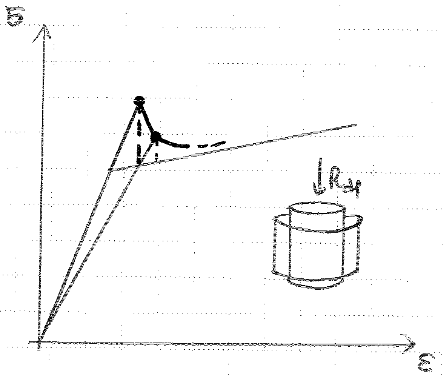
A specimen subjected to an uniaxial load is discretize into a number of triangular elements and, at the bottom, the following boundary conditions are given.

$$u = 0 \quad v = 0$$

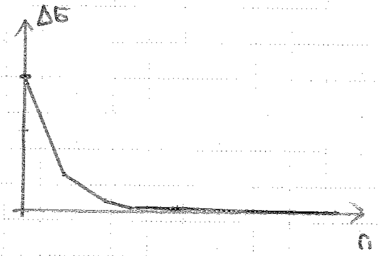
In the middle of the specimen, there is a material with an elasto-plastic behaviour, with failure given by Tresca criterion.

$$\tau_{max} = c_u$$

We apply the iterative method.

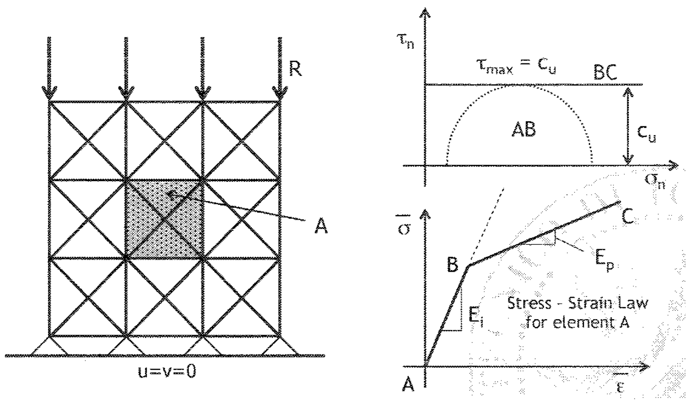


In the I iteration, we assume the initial parameters and, applying the Force, we can compute the stress and then the strain.
The I iteration will give a certain unbalanced load.
In the II iteration, we work with a smaller Young modulus and we get a lower stress. As the force applied is the same, the specimen has to deform itself and assume a bigger area.
Doing many iteration, we get closer to the constitutive law.

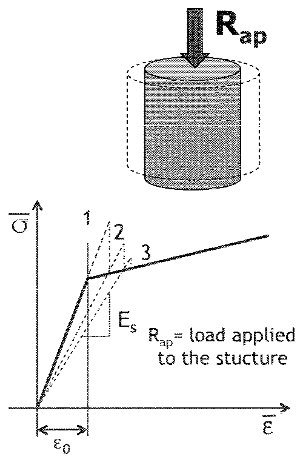


If we plot the error in relation to the number of iterations, it varies and reduces very quickly at first, then it reduces slowlier and gets to a constant value after a number of iterations.

EXAMPLE – UNIAXIAL LOADING OF A SPECIMEN



EXAMPLE – UNIAXIAL LOADING OF A SPECIMEN



E and ν will be some complicated but known functions of the stress $\bar{\sigma}$ or strain $\bar{\varepsilon}$ components

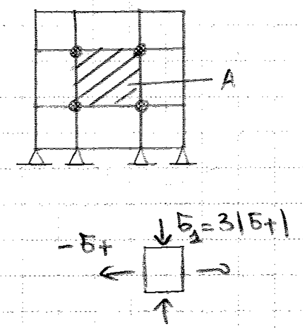
$$\bar{\varepsilon} = \frac{2}{3}\sqrt{(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2}$$

$$\bar{\sigma} = \frac{1}{3}\sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}$$

Zienkiewicz & Cheung 1967

(*) first applications of FEM to the solution of nonlinear problems in stress analysis

Ex (APPLICATION OF ITERATIVE METHOD : NON - TENSION MATERIAL)



A specimen is subjected to a Brasilian test and it is discretized in triangular elements. In this specimen, there is a material with non - tension behaviour, i.e. its behaviour is linear elastic in compression and it has no tensile strength.

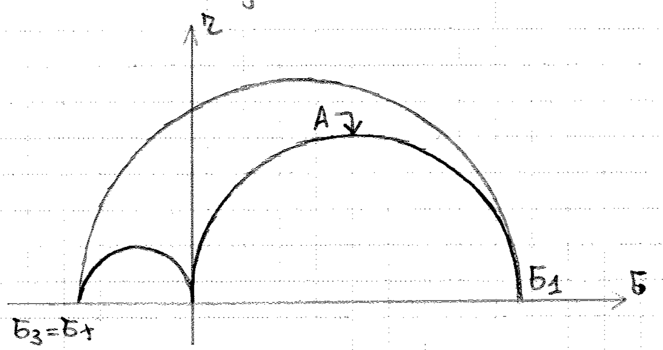
When the load is applied, the specimen is expected to fail with tensile failure along the vertical axis.

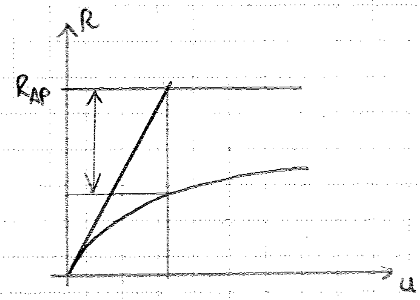
Focusing on stress state at the center of the specimen, theoretically it would present

$$\sigma_3 = -\sigma_t$$
$$\sigma_1 = 3\sigma_t$$

σ_t = tensile strength

This state of stress can't be carried on the element, because it has no tensile strength and so there will be an unbalanced load.

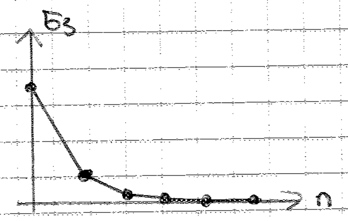




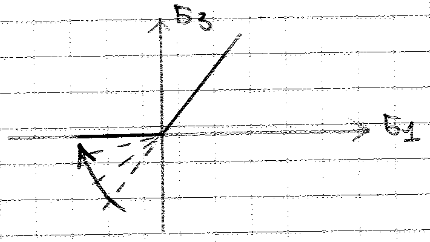
In particular, doing the analysis with Mohr circles, we can define the one representing the theoretical stress state and the element carries only a part - it carries all σ_1 and not tensile stresses.
By consequence, there is an additional stress state that can't be carried by the element and it represents a non-equilibrated stress ③.

We can notice that this is a similar situation with respect to the one seen in the iterative method, where a non-equilibrated load was defined.

This unbalanced stress, as it can't be carried by the central element, will be transferred to the adiacent elements.
So, we have to quantify this unbalanced stress and, in order to do this, we can write the equations of stress state when the plane reference system is changed.

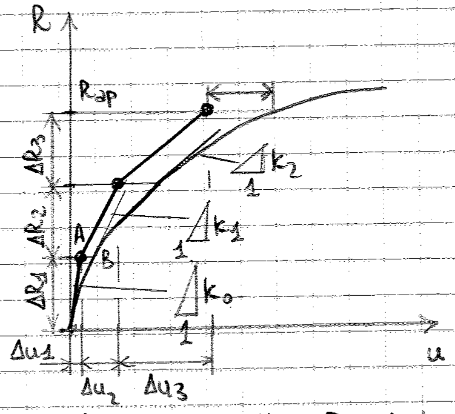


At the end of the first steps, the FEM model may again be subjected to tensile stresses, which are to be transferred to the surrounding elements.
The process is iterated up to obtaining $\sigma_3 = 0$ - no tension.



Incremental method (or tangent stiffness method)

It is an other approach to deal with non linear behaviour of geomaterials.



Given a nonlinear constitutive behaviour, written in terms of forces and displacements, we apply the load.

In iterative method, the load is fully applied. In incremental method, the load is divided in a number of increments, that could be of different size.

Firstly, the first increment $\{\Delta R\}_1$ is applied and the correspondent displacement $\{\Delta u\}_1$ is computed by using the finite element formulation, with the stiffness matrix evaluated in the origin.

$$\{\Delta u\}_1 = [k]_0^{-1} \{\Delta R\}_1$$

Yet, only part of the full load is applied and there is also an unbalanced load.

So, we apply the second increment $\{\Delta R\}_2$.
In this case, the stiffness matrix used is the one corresponding to point B - we know the level of the load here and the constitutive law and we get the elastic modulus and the stiffness matrix.
The displacement $\{\Delta u\}_2$ will be

$$\{\Delta u\}_2 = [k]_1^{-1} \{\Delta R\}_2$$

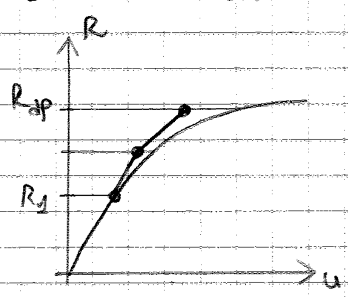
In this case, on the force - displacement plane, we are moving from point A along a line which is parallel to the tangent of the constitutive

PROPERTIES → in iterative method, in the first iteration we are far from real solution and more we go, more we get closer to it.
In incremental method, more we go and worse is the approximation.

→ as in iterative method, the stiffness matrix is computed element by element at each step, as the elements may have different stress states.

→ the accuracy of the solution depends on the number of increments and their size

OBSERVATION



In geotechnical applications, we usually don't start from the origin but from a level of load R_1.
So, the incremental method will be applied in the range of load from R_1 to R_{dp}.

STEPS OF INCREMENTAL METHOD

(I) Computation of stiffness matrices for the elements and then of the global stiffness matrix for the model.

(II) Division in load increments.

(III) Application of an increment and computation of displacement - and variation of stresses and strains, derived as secundary quantities in the element equations

(IV) Computation of the new deformability parameters, of the new stiffness matrices for the elements and of the new global stiffness matrix

(V) Continue until all increments aren't concluded.

III) Based on the elastic strain increment $\{\Delta\varepsilon\}_{1,el}$, the increment of elastoplastic stress is computed, according to the elasto-plastic behaviour.

$$\{\Delta\sigma\}_{1,ep} = [C_{ep}]\{\Delta\varepsilon\}_{1,el}$$

IV) The different between elastic increment and elasto-plastic increment is computed and it represents the UNBALANCED STRESS

$$\{\Delta\sigma\}_1 = \{\Delta\sigma\}_{1,el} - \{\Delta\sigma\}_{1,ep}$$

This is a stress increment that we would like to apply to the specimen but we can't.
So, it is similar to the unbalanced load we have defined in non linear methods but this unbalanced stress is computed in the PLASTIC FIELD, whereas the other value is defined in a non linear elastic field.
In other ~~ter~~words, we are applying the same kind of concept in plastic field.

This unbalanced stress is transferred to the neighbouring elements and, to do this, we have to pass from stresses to nodal forces.

$$\{R\}_1 = \int_{V_e} [H]^T \{\Delta\sigma\}_1 \, dV$$

In this way, this stress is transferred by applying the vector of nodal forces to the nodes.

V) We have a new initial stress and, having these nodal forces, we apply the FEM equations and we compute the new displacements, strains and stresses - elastic stresses, plastic stresses and their difference.

$$\{\Delta u\}_1 = [K]^{-1}\{R\}_1$$

$$\{\Delta\varepsilon\}_2 = [B]\{\Delta u\}_1$$

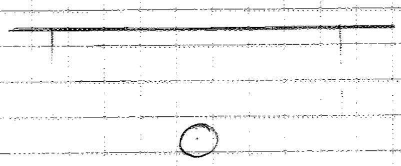
$$\{\Delta\sigma\}_2 = \{\Delta\sigma\}_{2,el} - \{\Delta\sigma\}_{2,ep} = [C]\{\Delta\varepsilon\}_2 - \{\Delta\sigma\}_{2,ep}$$

Now, the unbalanced stress is smaller.

(VI) The cycle is repeated until when the unbalanced ~~u~~stress is small enough, i.e. the unbalanced stress is smaller than a given number we have definid previously.

We can notice that the method is similar to iterative approach but now we work above the yield limit.

Ex (CIRCULAR TUNNEL)



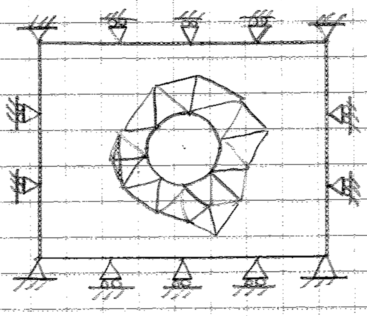
Given a ~~ear~~deep circular tunnel excavated into a ground with an isotropic in-situ stress.

$$\sigma_1 = \sigma_3 = 10\ MPa$$

$$E = 20\ GPa$$

$$c = 1,5\ MPa \qquad \varphi = 30^\circ$$

Evaluate stresses, strains and radial displacements - circumferential displacements are null due to symmetry - adopting ILE behaviour and then elasto-plastic behaviour.

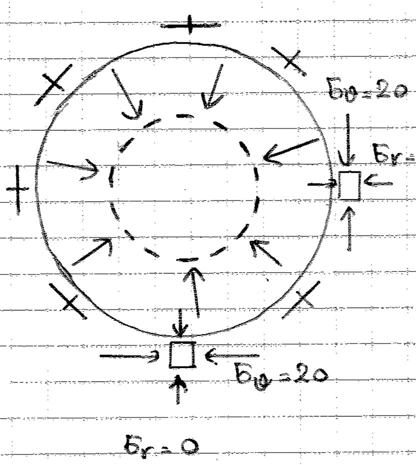


Adopting FEM, the first step consists of closing the domain by applying the boundary conditions in terms of displacements - rollers and hinges.
Alternatively, we could apply stress boundary conditions, if we know stress state in situ.

Then the domain is discretised, by using triangular elements.

Then the material properties are assigned and we assume that there is a homogeneous rock mass.

We start the computation for ILE behaviour.



Our prediction, coming from the analytical solution, is that principal stresses are

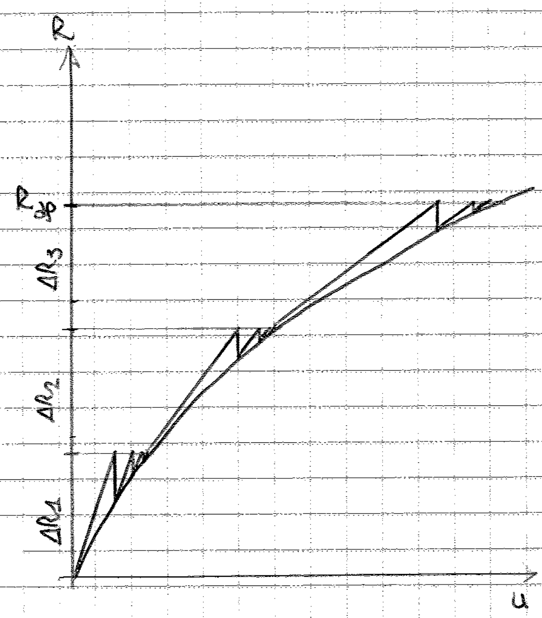
$$\sigma_\theta = 20\ MPa \qquad \sigma_r = 0\ MPa$$

at the contour of excavation.
If we refer to horizontal and vertical stresses, there is a rotation of stresses around the excavation

→ on springline $\quad \sigma_v = 20\ MPa \quad \sigma_h = 0\ MPa$

→ on crown $\quad \sigma_v = 0\ MPa \quad \sigma_h = 20\ MPa$

Iterative and incremental approach (MIXED METHOD)



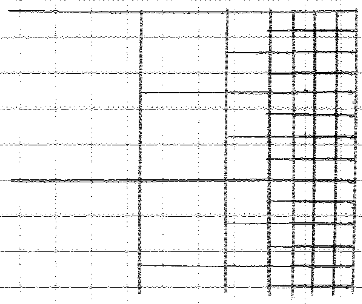
A non linear material is subjected to a load R_{dp}.

The mixed method consist of dividing the load in a number of increments - as in the incremental method.
Then, each increment is applied but the result is far from real solution due to the error.
In order to minimize it, we can apply the iterative approach, assuming that the increment applied is the total load.
In this way, the unbalanced load is reduced and, once we are close enough, the second increment is applied and then the iterative approach is applied again.

Thus, in the mixed method within each incremental step the iterative approach is applied to refine the solution and reduce the error.

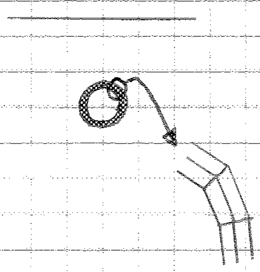
This approach, with respect to iterative method, requires a bigger number of iteration but the unbalanced load is much smaller at each step and the results are better.

⇒ DISCRETISATION OF STRUCTURAL ELEMENTS MAY CAUSE PROBLEMS IN DOMAIN DISCRETISATION



Actually, the mesh could be modified by adopting small QUAD elements close to the diaphragm wall and, furtherly, changing in bigger QUAD elements.

In this way, the shape of the elements remains square and it is a good thing because, in this kind of structure, they help to keep a simple geometry.



A similar operation can be done for a tunnel lining, by adopting QUAD elements to reproduce the shape.
In this way, we can also model the geometry of a reinforced area, where parameters like stiffness and cohesion are improved - e.g. by using fiber glass.

We can also notice that, by discretising the structural element with only one big element, we could get only the displacements at the boundaries and, due to the hypothesis of linear variation of the displacement field within the element, we don't have a good description of stresses and strains in the element.
Indeed, from an element we derive nodal displacements and, with a big element, we will know only what happens at intradox and extradox.

⇒ in order to reproduce structural behaviour, usually $3 \div 4$ LINES OF ELEMENTS are adopted

FIELD OF APPLICATION → ok for piles, diaphragm walls, linings and retaining walls

→ BAD to reproduce bolts, cables and anchors in the ground.

ADVANTAGE: using this method, we only get stresses, whereas in the II method we will get stresses, bending moment and axial force.
The distribution of stresses is very similar to the actual one and internal actions are obtained from stresses with an additional steps.

3 Method 2

In some situations, the dimensions of the structural elements are small compared to the overall geometry and modelling them with 2D continuum elements would give a very ~~sm~~ large number of elements or elements with unacceptable aspect ratios.

Actually, many times we are not interested in full distribution of stresses within the structural elements, but only on the DISTRIBUTION OF INTERNAL ACTIONS as

→ bending moment

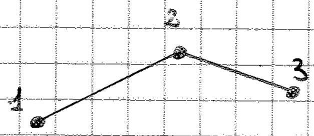
→ axial force

→ shear force

The advantage is that we can avoid full discretisation of the structure because we can adopt some special finite elements formulated by collapsing one or more dimensions into zero:
These elements are formulated directly in terms of bending moments, axial forces and shear forces and their associated strains, in order to get the quantities of engineering interest in a direct way.

We can distinguish many tipes of these special finite elements.

→ Beam elements



They are 1D ELEMENTS WITH 3 DEGREES OF FREEDOM at each node, that are x-translation, y-translation and rotation.
These elements can be joined together with one another and/or the grid.

Beam elements are used to represent a structural member in which bending resistance and limited bending moment are important, as diaphragm walls and tunnel linings.

The equation that governs their behaviour is the following one.

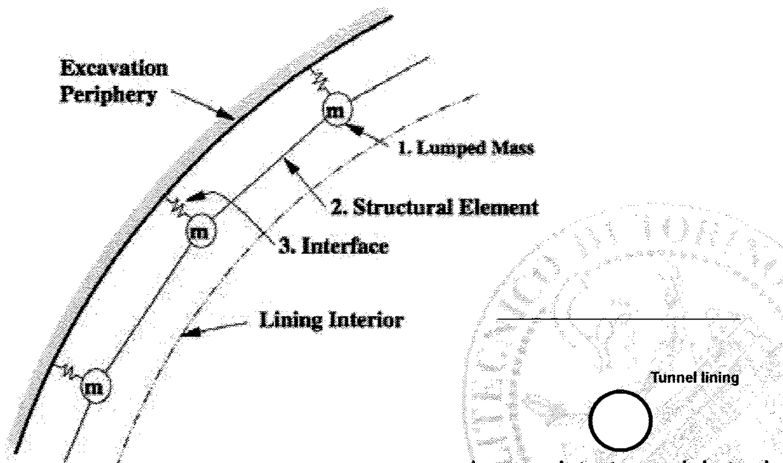
$$\begin{pmatrix} \varepsilon \\ \chi \\ \gamma \end{pmatrix} = [B] \begin{pmatrix} u \\ v \\ \vartheta \end{pmatrix}$$

Strain-displacement relationship

In this case, the 2 dimensions are collapsed to 1 dimension, as the beam element has one length and 2 nodes.

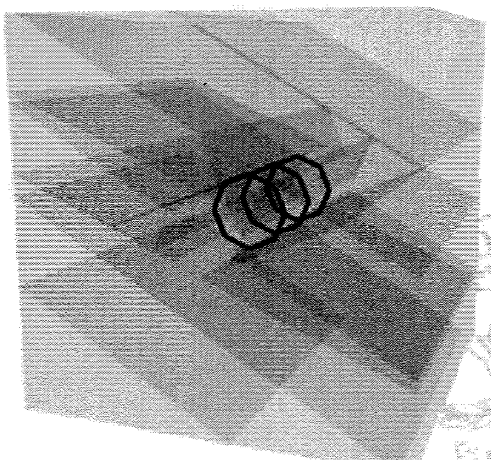
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 95 di 218

BEAM ELEMENT



Appropriate to model steel sets in tunnels

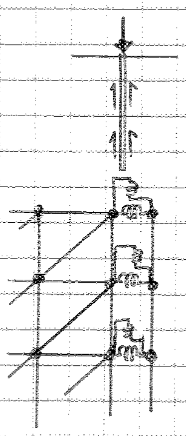
BEAM ELEMENT



Tunnel with beam elements (3D model)

→ pile elements

They are less used in commercial softwares because their behaviour can be reproduced with a beam element with a vertical load.



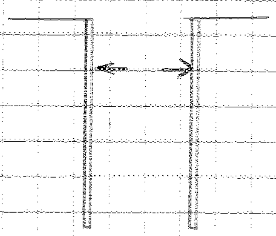
~~On~~ the other side, pile interacts with the ground with the ground due to friction force and we need something to represent this axial force.

Thus, pile elements are 2D elements that can transfer normal and shear forces and bending moments to the grid.
So, piles offer the combined features of beams and cables and shear forces act parallel to the element and normal forces perpendicular to the element.

Pile elements are designed to represent the behaviour of foundation piles.

→ support members

They are intended to model props, but they are not very common elements.



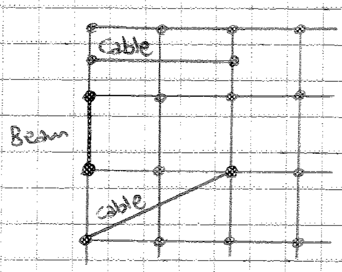
They have a very simple formulation and they basically apply two forces on the two sides.
In other words, they are springs connected between two boundaries and have no independent degrees of freedom because they simply impose forces on the boundaries to which they are connected.

A support member may also have a width associated with it and, in this case, it behaves as if it were composed of several parallel members spread out over the specified width.

In conclusion, to introduce interaction in FEM, there are 2 ways.

→ representing the proper geometry and discretizing the structural component with finite elements.

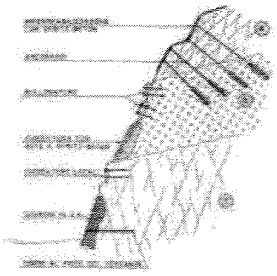
→ using special elements, losing the correct geometry.

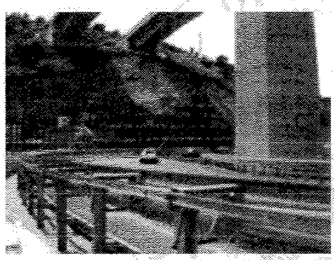


These elements have to be connected with the grid.
For instance, a beam has two points where the structural element is connected to the ground.
A cable, on the other side, has not necessarily to be directly connected to the nodes.

CABLE ELEMENT

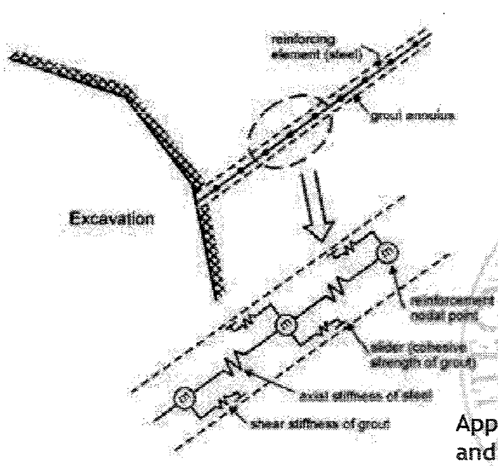
Cable elements are one-dimensional axial elements that may be anchored at a specific point in the grid (point-anchored) or grouted so that the cable element develops forces along its length as the grid deforms. Cable elements can yield in tension or compression, but they cannot sustain a bending moment. If desired, cable elements may be initially pre-tensioned. Cable elements are used to model a wide variety of supports for which tensile capacity is important, including rock bolts, cable bolts and tiebacks.

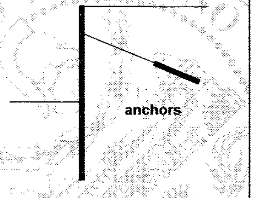




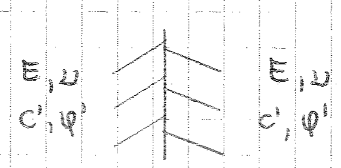
CABLE ELEMENT

Cable elements allow to include the shear strength along the cables.

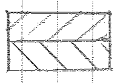




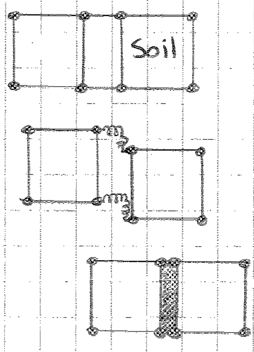
Appropriate to model anchors and bolts



Parameters for the soil - structure interface can be obtained via laboratory tests on the interface, e.g. direct shear testing.
Yet, this approach is expensive and time-consuming and, in laboratory, it is difficult to reproduce real behaviour.
For this reason, sometimes so a value is assumed, by taking something in between of the properties of the 2 materials.



Once we have defined the constitutive behaviour, we can introduce the interface element.



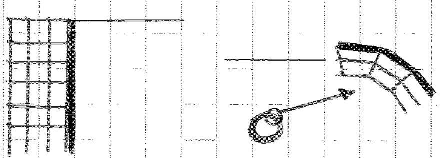
We can distinguish several types of interface elements.

→ FINITE ELEMENTS - continuum elements

→ normal and shear SPRINGS, that can only model deformability - so, the behaviour is linear elastic

→ ZERO THICKNESS INTERFACE ELEMENTS.

Interface elements are very used in method 1 in correspondence of the common nodes, where there is an interaction between structure and ground.



In this way, we can introduce sliding of nodes and reproduce a behaviour that resembles reality.

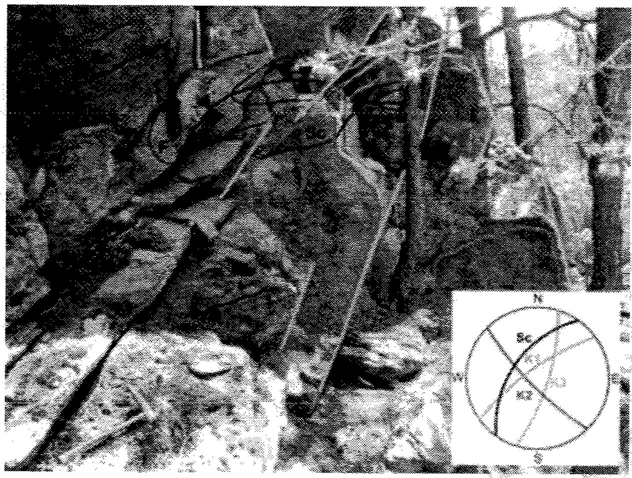
$$F_s = \frac{R}{E}$$

and, as we can estimate actions, we can compute strength parameters. In this situation, the problem is more complex and, with a software, we try to infer parameters.

These parameters are applied into a new model in which structural elements are introduced according with method II - cable elements. Only exception is the retaining wall, which is simulated with its actual geometry through finite elements.
The results show that there are some superficial areas where displacement is big. This means that this solution is not enough and we can solve the problem by placing a steel net.

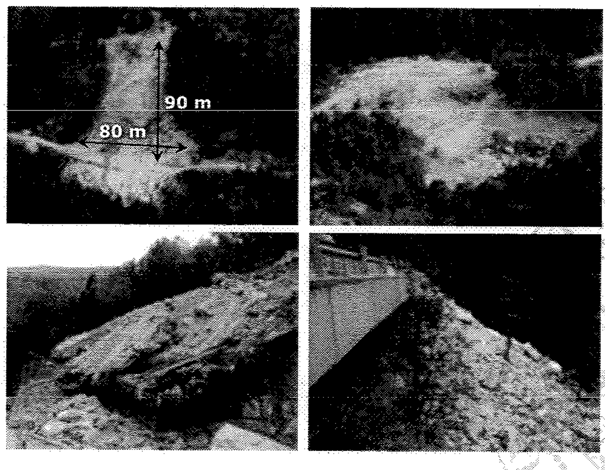
Once this system has been placed, we compute the safety factor and we can notice that it has been improved by 0,3.

ROCK LANDSLIDE



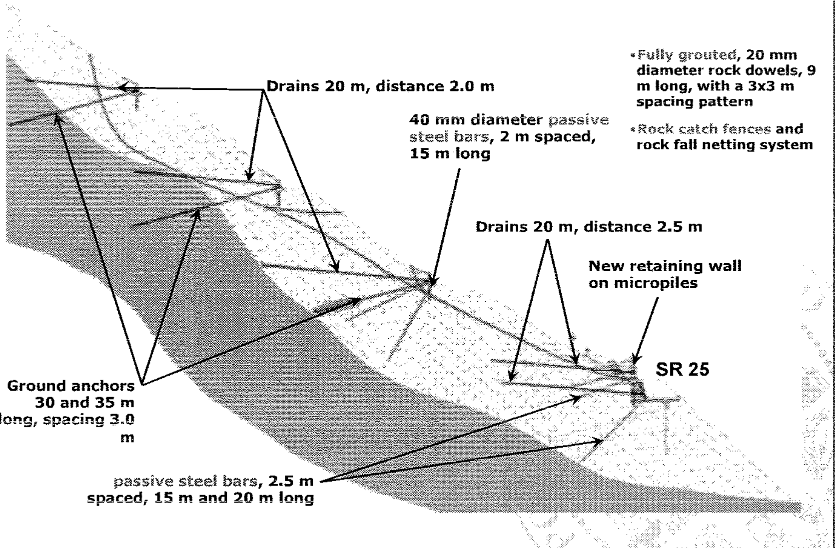
Discontinuities in the Ruitor Unit (picture taken from SW).
K1, K2; K3: main joint sets;
Sc: schistosity;
F: isoclinal fold

ROCK LANDSLIDE

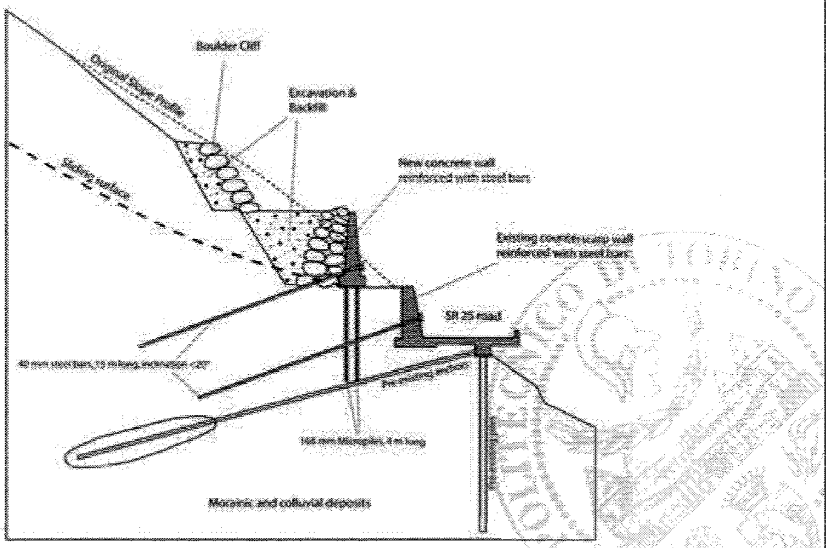


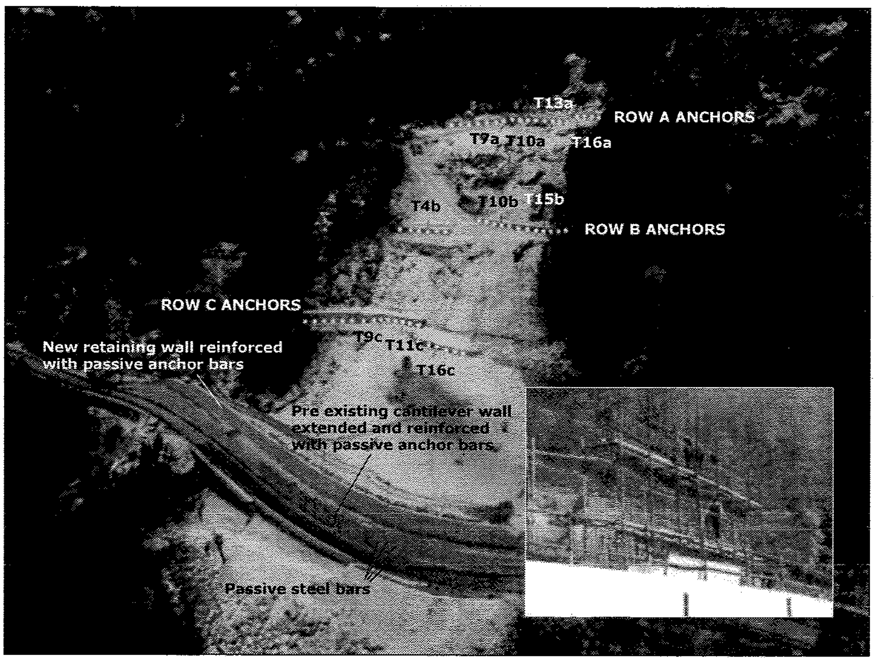
- Debris sliding (Cruden & Varnes, 1996)
- Volume: 35.000 m^3
- Triggering after rain falls (270 mm between march and april 2009)
- Snow melting contribution
- Important damages to the retaining wall of the Regional Road SR25 (the only connection between the Valgrisenche valley and the Aosta valley)

REINFORCEMENT DESIGN



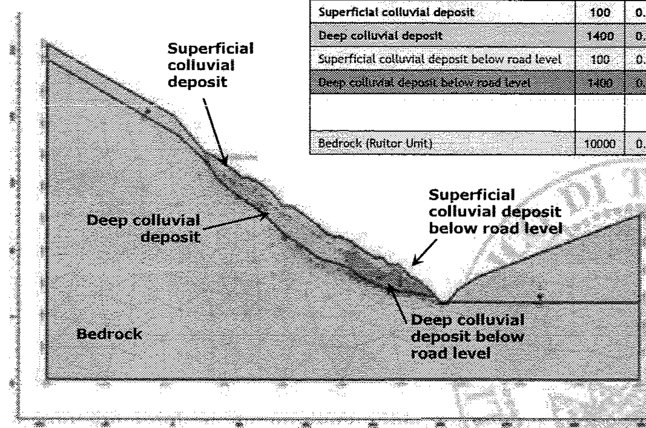
DETAIL OF THE CONCRETE WALL





THE FEM MODEL

	E [MPa]	ν [-]	c' [kPa]	φ [°]
Superficial colluvial deposit	100	0.3	10	27
Deep colluvial deposit	1400	0.3	70	31
Superficial colluvial deposit below road level	100	0.3	50	27
Deep colluvial deposit below road level	1400	0.3	100	31
			UCS [MPa]	GSI
Bedrock (Ruitor Unit)	10000	0.3	75	70



Suggestions to build a FEM model

To introduce the strategies to build a nice FEM model, we follow the steps of FEM

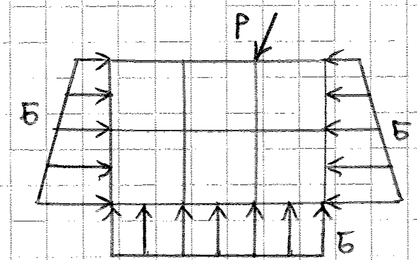
I Discretizing

The problem is discretized, creating the finite element mesh.
Very often, we get ~~the~~ assistance by commercial softwares, that have an automatic generating system of mesh.
Yet, this is not always the best way.

II Definition of boundary conditions

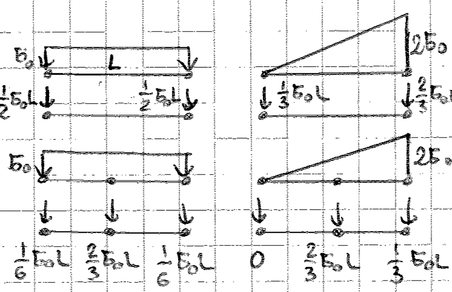
Boundary conditions can be defined in terms of displacements, by applying roller and hinges at the boundaries.

Otherwise, we can apply pressure as a boundary condition - stress boundary conditions.
This condition has to be converted into equivalent nodal forces with the expression



$$\int_{S_0} [H]_0^T \{f^S\}_0 \, dS$$

and these force can be easily computed for simple stress distributions over elements with 2 nodes or 3 nodes.
This computation is performed automatically in many softwares.
We also have the possibility to apply nodal forces.



III Assignment of material properties to the elements

To each element, material properties are assigned, with an appropriate constitutive law and parameters.

The first approach is better because sometimes it is not easy to define in-situ stress state.

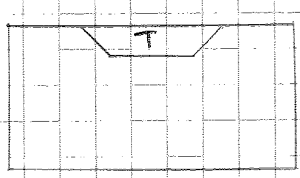
V Computational stages

Computational stages are useful to simulate CONSTRUCTION SEQUENCE.

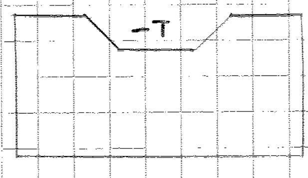
In geotechnical applications, we can define two tipes of construction.

→ excavation

It is simulated through the removal of part of the model.
How we can do this on a computation point of view?



Firstly, we define a part of the model and we set initial stress state.
The part to be excavated will have a certain weight and, due to this weight, it will transfer same forces to the surrounding elements.
If we remove this part, we will also remove these forces and the remaining part will go up.



So, we have to compute forces applied by the excavated area to the contour line and then THESE FORCES ARE APPLIED WITH THE OPPOSITE SIGN, whereas the material is "removed" by deactivating its elements.

Indeed, if we deactivate element, no displacements or changes in stress occur and we have to apply forces opposite to the internal stresses in the soil mass that act on the excavated surface before the part is removed.

These nodal forces are given by the following expression

$$\{R_0\}_i = \frac{1}{m} \sum_{j=1}^{m} \{R_0\}^j$$

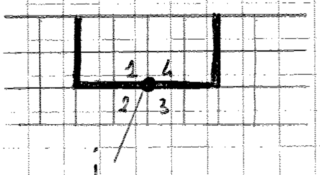
where

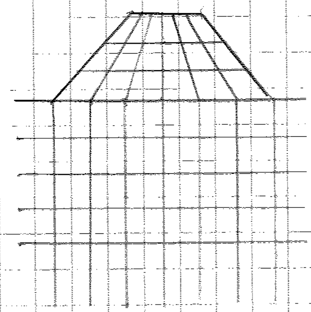
$$\{R_0\}^j = \int_{V_e} [B]_e^T [C]_e [B]_e \{u\}_e \, dV$$

i = node on the excavation surface

j = elements around node i

m = number of elements around node i





For instance, to simulate the embankment construction, all elements in the embankment are deactivated up to the stage when it is to be constructed.
At first increment, all elements in the first layer are reactivated - i.e. added to the active mesh at the beginning of the increment.
These elements are assigned a costitutive model and self weight forces are computed, from which we evaluate nodal equivalent forces that will be added to the increment.
In this way, we evaluate the incremental displacements.

These displacement are due to self-weight and the ones calculated for the nodes in the constructed element are zeroed, as we use them only to compute the variation of stress.
Then, stress adjustments are made.

The final result is obtained by accumulating the results for each increment of the analysis.

VI Computation

Many softwares show in output the computational steps and the details of the iteration process, i.e. tolerance and number of steps, that gives the indication about when the process is stopped. Usually, default values are good.

VII Interpretation

We have to analyze and validate results comparing them to predictions or monitoring data.

This formulation allows us to build a FEM model with commercial softwares.

CONSTITUTIVE BEHAVIOUR OF GEOMATERIALS

When we build the element equations in FEM, we have to determine stress state and, to do this, we need to define a constitutive model.

Models can't be perfect, but can be adequate for the purpouse, so we need to select a good constitutive model, that proposes a realistic representation and reproduces real behaviour.
If it is possible, we always should try to simplify and stay in elasticity field, as computation is easier and the model is simple to interpret.
Unfortunately, in geotechnic engineering 80% of times we go into elasto-plasticity field.

Elasticity field: linear elastic behaviour

1 This behaviour can be described with a linear relationship between the stress tensore σ_{ij} and the strain tensor ε_{ij}.

$$\varepsilon_{ij} = D_{ijkl}\,\sigma_{kl}$$

D = compliance tensor

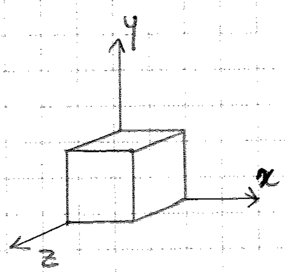
If we write the inverse relationship

$$\sigma_{ij} = C_{ijkl}\,\varepsilon_{kl}$$

C is the stiffness tensor, i.e. the inverse of the compliance tensor.

We will see that in an isotropic, linear elastic medium the 36 constants in the tensor can be reduced down to 2 parameters only.

We can also write the relationship in an other way.



In 3D space, we have normal and angle strain components in the strain tensor and normal and shear stress components in the stress tensor.

$$\{\varepsilon\}_p = \begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{pmatrix} \qquad \{\sigma_p\} = \begin{pmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{pmatrix}$$

$$[D] = \begin{array}{c} \begin{array}{cccccc} & C & & & E & \end{array} \\ \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} & D_{15} & D_{16} \\ & D_{22} & D_{23} & D_{24} & D_{25} & D_{26} \\ & & D_{33} & D_{34} & D_{35} & D_{36} \\ & & & D_{44} & D_{45} & D_{46} \\ & \text{symm} & & & D_{55} & D_{56} \\ & & & & & D_{66} \end{bmatrix} \begin{array}{c} \\ \\ \\ \\ D \\ \\ \end{array} \end{array}$$

A B

2 Orthotropic material

It is a material where there is NO DIRECT COUPLING BETWEEN NORMAL AND SHEAR COMPONENTS IN DIFFERENT DIRECTIONS.
~~It~~
It means that material has 3 symmetry planes and the relationship changes.

$$\frac{\varepsilon_{xx}}{\sigma_{xx}} = \frac{1}{E_{xx}} \qquad \frac{\varepsilon_{xx}}{\sigma_{yy}} = -\frac{\nu_{yx}}{E_{yy}} \qquad \frac{\gamma_{xy}}{\tau_{xy}} = -\frac{1}{G_{xy}}$$

↳ similar equations are valid for ε_{yy}, ε_{zz}, γ_{xz}, γ_{yz}

In this case, direct coupling between normal components is given by a constant E_{ii}, whereas indirect coupling of normal components is given by the parametre ν_{ij}, that gives the strain in orthogonal direction.
To fill direct coupling of shear block, we introduce a shear modulus G_{ij}.

E_{ii} = YOUNG MODULUS

ν_{ij} = POISSON COEFFICIENT

G_{ij} = SHEAR MODULUS

In this case, we have the following matrix.

$$[D] = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & -\frac{\nu'}{E'} & 0 & 0 & 0 \\ & \frac{1}{E} & -\frac{\nu'}{E'} & 0 & 0 & 0 \\ & & \frac{1}{E'} & 0 & 0 & 0 \\ & & & \frac{2(1+\nu)}{E} & 0 & 0 \\ & \text{symm} & & & \frac{1}{G'} & 0 \\ & & & & & \frac{1}{G'} \end{bmatrix}$$

4 Isotropic geomedia

In this case, we have ISOTROPY IN ALL DIRECTIONS.
So, Young modulus and Poisson coefficients in different directions are equal and, using the relationship between shear modulus and the other constants, we can say that there are only 2 indipendent constants.

$$E_1 = E_2 = E_3 = E$$

$$G_{12} = G_{23} = G_{31} = G$$

$$\nu_{21} = \nu_{32} = \nu_{31} = \nu$$

$$\frac{1}{G} = \frac{2(1+\nu)}{E}$$

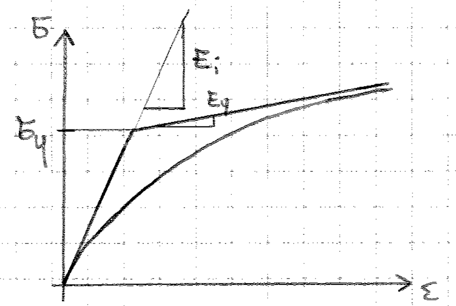
The matrix assumes the following form.

$$[D] = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & -\frac{\nu}{E} & 0 & 0 & 0 \\ & \frac{1}{E} & -\frac{\nu}{E} & 0 & 0 & 0 \\ & & \frac{1}{E} & 0 & 0 & 0 \\ & & & \frac{2(1+\nu)}{E} & 0 & 0 \\ & & & & \frac{2(1+\nu)}{E} & 0 \\ & & & & & \frac{2(1+\nu)}{E} \end{bmatrix}$$

For this reason, isotropic model allows the cheapest and easiest approach.

3 Bilinear elastic model

It is a model similar to the multilinear model, but simpler.



At the beginning, we consider a linear elastic behaviour with the Young modulus at the origin - slope is given by the tangent to real curve at the origin.
Above a certain value, we adopt a ~~diff~~ linear law with a different slope, given by the asinthote.

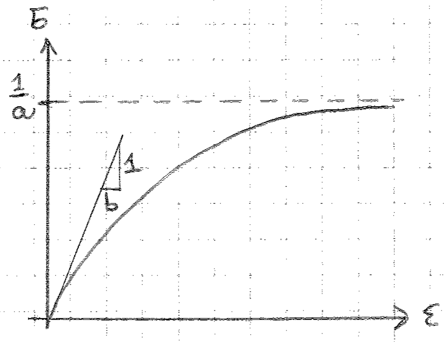
$$\begin{cases} \{\Delta\sigma\} = [C]_i \{\Delta\varepsilon\}, & \sigma < \sigma_y \\ \{\Delta\sigma\} = [C]_y \{\Delta\varepsilon\}, & \sigma > \sigma_y \end{cases}$$

In this model, Poisson's ratio is assumed constant ($\nu \sim 0,3$), because it has only a little influence and its variation ratio is little - whereas Young Modulus determines stresses and strains in a problem.

OBSERVATION: bulk modulus K and shear modulus G undergo the same decrease, as they depend directly on Young modulus E.

4 Hyperbolic model

It is one of the best NONLE models and it's implemented in softwares.

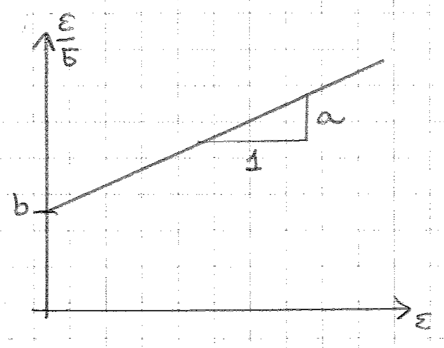


This constitutive model is defined by the equation

$$\sigma = \frac{\varepsilon}{b + a\varepsilon}$$

a = parameter related to the asinthote
b = slope in the origin
} these parameters govern the shape

By changing the axis, we can write a constitutive law plot in a linear way



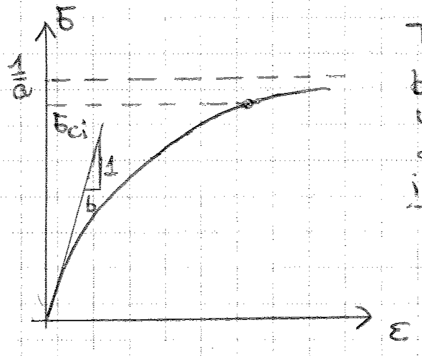
$$\frac{\varepsilon}{\sigma} = b + a\varepsilon$$

This relationship reminds Mohr-Coulomb criterion and a is the slope, whereas b is the intersection with y axis.

By substituting in equation ①, we obtain the following relationship.

$$E_t = \underbrace{\frac{1}{b}}_{E_i\ (②)}(1 - a\sigma)^2 =$$

$$= E_i\left(1 - \frac{R_f \sigma}{\sigma_{ci}}\right)^2$$

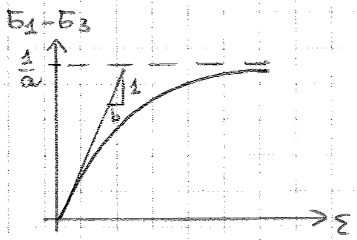


Thus, we are using an elastic constitutive model but now we have added also information about strength - we inser resistance in an elastic formulation - and the effect is that slope - tangent modulus - is a function of strength.

$$E_t = E_i\left(1 - \frac{R_f \sigma}{\sigma_{ci}}\right)^2$$

Now, in geotechnical engineering we usually deal with DEVIATORIC STRESSES:

so, we can write hyperbolic model in terms of deviatoric stresses



$$\frac{\varepsilon}{\sigma_1 - \sigma_3} = b + a\varepsilon$$

In this case, term $1/a$ represents the ultimate deviatoric stress.

In this case, we can define the ultimate deviatoric stress by using the Mohr-Coulomb failure criterion.

$$(\sigma_1 - \sigma_3)_f = \frac{2\sigma_3 \sin\varphi + 2c\cos\varphi}{1 - \sin\varphi}$$

Putting this formulation in equation ①, we get the elastic tangent modulus in relation to the deviatoric stress at failure

$$E_t = \underbrace{\frac{1}{b}}_{E_i\ (②)}(1 - a\sigma)^2 =$$

$$= E_i\left[1 - \frac{R_f(\sigma_1 - \sigma_3)}{\underbrace{(\sigma_1 - \sigma_3)_f}}\right]^2$$

inside here, we have the Mohr-Coulomb parameters.

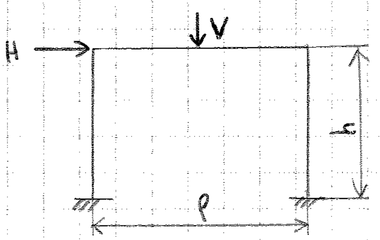
Elasto-plasticity models

The issue is complicated and the aim is to introduce the main constitutive models used to describe plastic behaviour.
We remember that models have a precise aim:

physical reality is complicated and it is very difficult to have a complete view of reality.
For these reason, a model is not a mirror of reality but considers same specific issues of real behaviour - making some mistakes - that are helpful to us.

Introduction

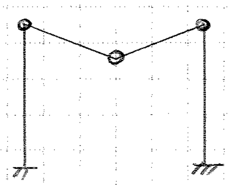
1 Let's focus on a simple case, that is the analysis of the COLLAPSE OF A PORTAL FRAME.



If we consider a portal frame subjected to 2 puntual forces, the structure can undergo collapse, when these forces assume 2 specific values.

We have 3 possible mechanisms of collapse.

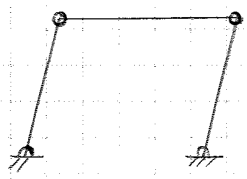
→ beam mechanism



The forces are ~~so~~ such that 3 plastic hinges develop at the top of the portal.
There's a value of vertical force applied to this portal that generates this condition

$$\frac{Vl}{M_p} = 8 \qquad M_p = \text{plasticization moment}$$

→ sway mechanism



Now, plastic hinges are at the bottom and the top of the portal and the mechanism occurs when horizontal force has the following value.

$$\frac{Hh}{M_p} = 4$$

→ combined mechanism

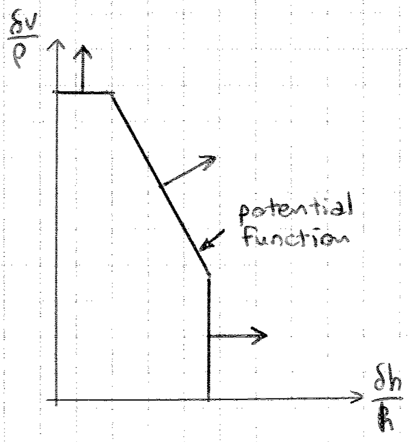
In this case, both the horizontal and the vertical force play a role in the collapse.

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 129 di 218

→ in case of combined mechanism, due to geometry and the position of plastic hinges, mechanism is more complicated and the ratio between displacement is 1 : 2.

$$\frac{\frac{\delta v}{\rho}}{\frac{\delta h}{h}} = \frac{1}{2}$$

These displacements are NOT REVERSIBLE, because this is a collapse and portal is becoming a mechanism. So, they don't recover when we take out the forces.



If we focus on the direction of displacements, we'll get a plot into which we put vertical and horizontal displacements in different conditions

→ in beam mechanism, we have only vertical displacements

→ in sway mechanism, we have only horizontal displacements

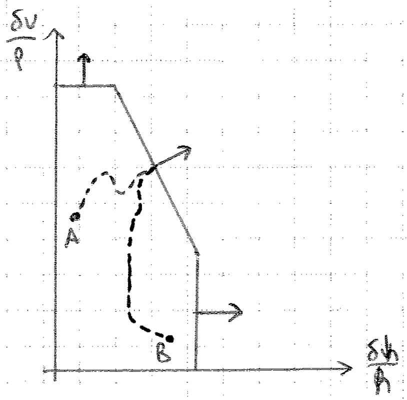
→ in combined mechanism, the ratio is 1 : 2.

Having these displacement, we can describe them with a potential function:

it is a mathematical function which gradient is a vector - in this case, the displacement vector - normal to the function itself.

So, in each mechanism of collapse, we have defined displacements and we can build the potential function, remembering that its normal is parallel to the displacement.

PROPERTIES



→ if we start from different load conditions and go to the same point of the potential function - e.g. the surface of the combined mechanism -, the direction of the displacement ~~will~~ always will be the same.

⇒ there is a no load-path dipendence of the displacement

It may sound strange, because when we apply normal stresses, we're used to see volume strains and not shear strains - same issue is valid if we apply shear stresses -, but here this question has no importance.
The issue is also valid in the load plane $V\rho/M_p - Hh/M_p$.

These aspects - nonlinearity, irreversibility, stress path dependence - need to be taken into account by constitutive equations aiming at providing prediction of the geomaterials under applied loads.
These models don't represent each aspect and are based only on some features of the response seen in experimental observations.
The quality of predictions depends fundamentally on the ability of defining a suitable idealization for real materials, for the class of loading paths of practical interest.

With reference to the behaviour of a specimen in tri-axial test, we can define some stress and strain variables

→ Volumetric strain: it's the variable chosen to define strain, because volume changes affect mechanical properties in soils.
It's given by the sum of principal strains and, in axisymmetric conditions, we have

$$\delta\varepsilon_p = \delta\varepsilon_a + 2\delta\varepsilon_r$$

→ mean stress

we need to link strain increment and stress variables and effective mean stress is chosen as volumetric stress variable.
It's given by the sum of principal stresses divided by 3 and, in axi-symmetric conditions (like a triaxial test), we have

$$p' = \frac{\sigma_a' + 2\sigma_r'}{3}$$

In this way, from a thermodynamical point of view, we can define the work related to the volumetric change.

$$\delta W_p = p' \, d\varepsilon_p$$

→ deviatoric stress:

to describe the behaviour of a geomaterial, we have also to describe changes in shape and we use deviatoric stress.
In axisymmetric conditions, it's equal to

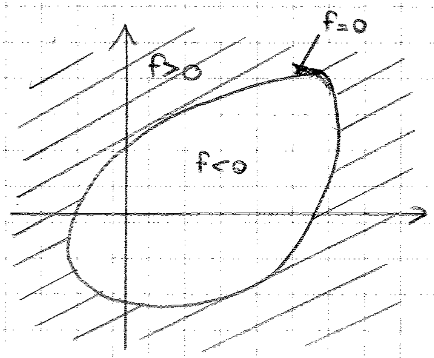
$$q = \sigma_a' - \sigma_r'$$

→ deviatoric strain (or distorsional strain)

it expresses change in shape.

3 Yield function

We have to define the yield function plotted in function of q and p'.



From a mathematical point of view, there are 3 possibilities

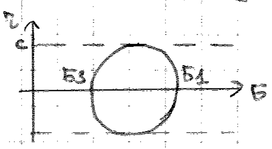
→ if $f < 0$, we fall into the elastic domain from a physical point of view

→ if $f = 0$, we have the collapse condition and we are on the border of the domain

→ if $f > 0$, it corresponds to an impossible state.

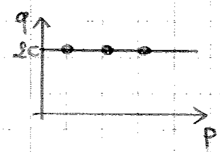
The yield function can be defined with reference to the different failure criteria

→ TRESCA CRITERION, useful in undrained conditions.



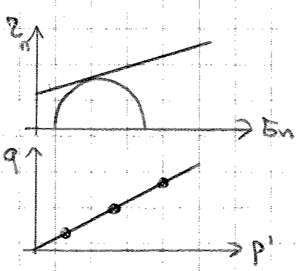
$$F(\sigma) = (\sigma_1 - \sigma_3) - 2c$$

→ VON MISES CRITERION, useful in undrained conditions



$$f = q - 2c$$

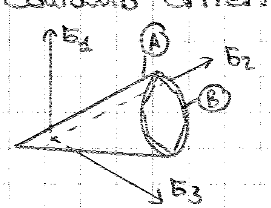
→ MOHR - COULOMB CRITERION



$$f = c' + |\sigma_n' \tan\varphi'| - \tau_n$$

$$f = q - Mp'$$

→ DRUCKER AND PRAGER CRITERION, that is the 3D version of Mohr-Coulomb criterion and it is plotted in function of principal stresses.

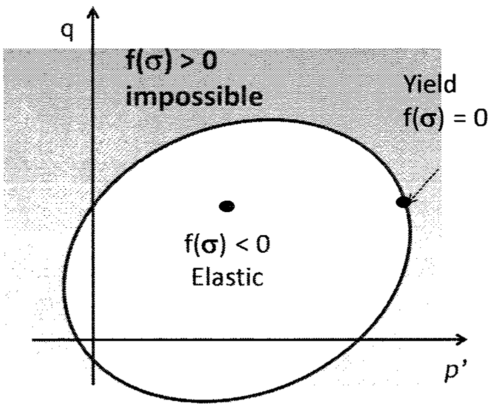


$$F(\{\sigma\};[k]) = \alpha I_1 + \sqrt{J_2} - k$$

Actually, Drucker and Prager model is quite different and its represented by the cone A, while Mohr - Coulomb is represented by the hexagon B.

Structure of elastic perfectly plastic models

Yield function –graphical representation of yield – yield surface

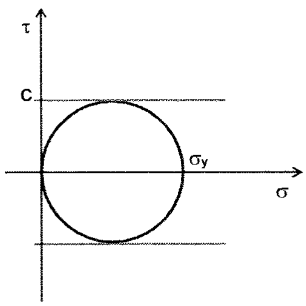


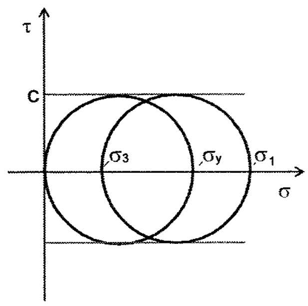
2D and 3D yield functions

2D Yield functions - TRESCA

$f = \max|\sigma_i - \sigma_j| - 2c = 0$ is we adopt the convention $\sigma_1 > \sigma_2 > \sigma_3$

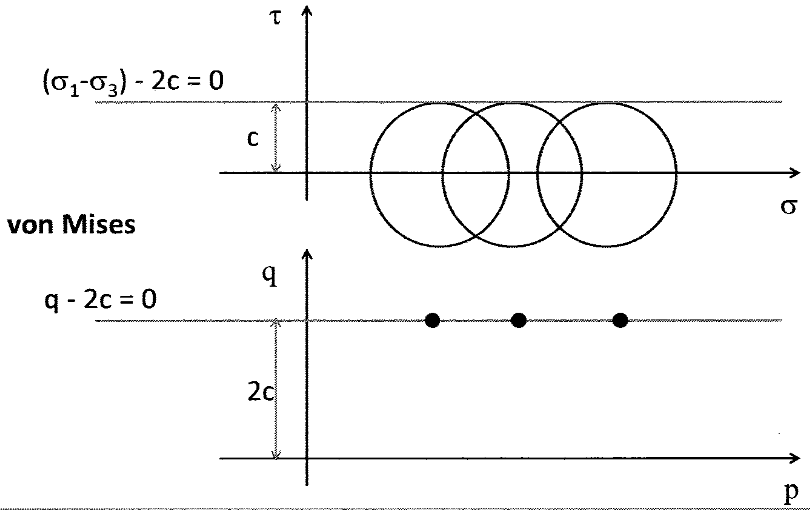
$$f = (\sigma_1 - \sigma_3) - 2c = 0$$





2D and 3D yield functions

Tresca in axysimmetry conditions



2D and 3D yield functions

3D Yield functions – von Mises

In terms of principal stresses:

$[(\sigma_1-\sigma_2)^2 + (\sigma_2-\sigma_3)^2 + (\sigma_3-\sigma_1)^2] = 8c^2$

In terms of invariants

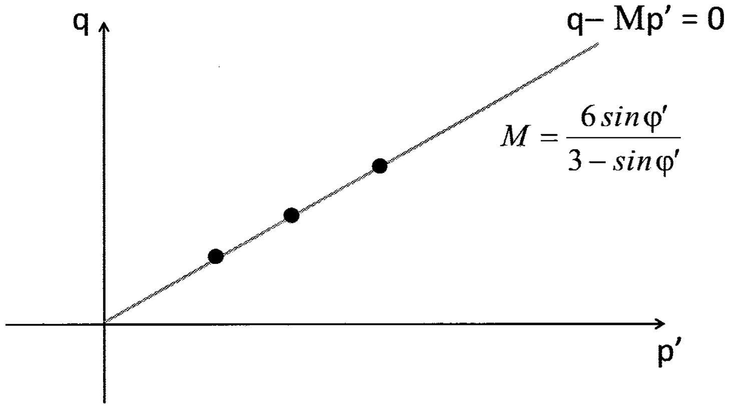
$q^2 = k$

In terms of normal and shear stresses:

$J_2=1/6\ [(\sigma_x-\sigma_y)^2 + (\sigma_y-\sigma_z)^2 + (\sigma_z-\sigma_x)^2]+\tau^2_{xy} +\tau^2_{yz} +\tau^2_{xz} = k^2$

Structure of elastic perfectly plastic models

Mohr Coulomb criterion (in 2D)

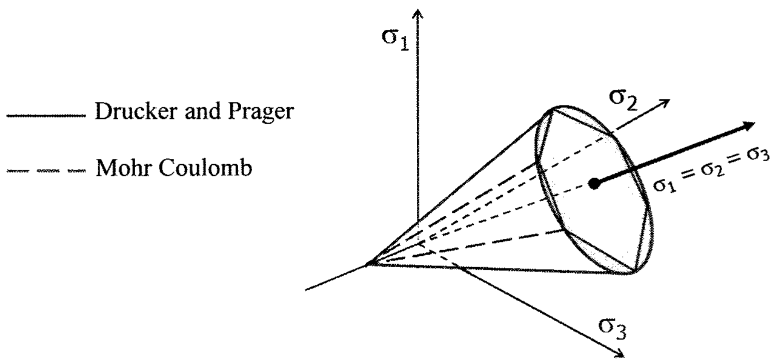


2D and 3D yield functions

3D Yield functions – Drucker and Prager and Mohr Coulomb

$F([\sigma],[K]) = \alpha I_1 + \sqrt{J_2} - k = 0$

with: I_1 first stress invariant; J_2 second invariant of stress deviator; α and k parameters which can be written in terms of c and φ



The condition that, once stress state achieves the yield surface, it remains there, is called consistency condition and it is represented by the equation

$$dF = \frac{\partial F}{\{\partial \sigma\}}\{\partial \sigma\} = 0$$

In elasto-plastic field, total strain is given by 2 components

$$\{\delta\varepsilon\} = \{\delta\varepsilon\}^e + \{\delta\varepsilon\}^p \qquad (1)$$

and the increase of the stress state is given by elasto-plastic stiffness matrix.

$$\{\delta\sigma\} = [D]^{ep}\{\delta\varepsilon\} \qquad (2)$$

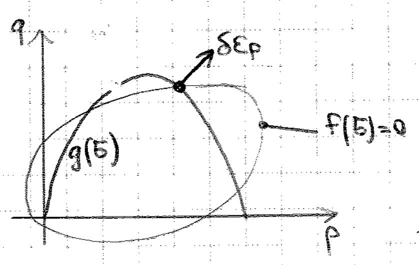
Plastic potential function

Plastic potential function is a function which gradient gives the direction of the plastic strain and define it means to define a FLOW RULE.

In elasto-plasticity, it is defined but expressing the increase in plastic strain as an unknown scalar μ multiplied by the plastic potential function - its derivative.

$$\{\delta\varepsilon\}^p = \mu \frac{\partial g(\{\sigma\})}{\{\partial\sigma\}} \qquad (3)$$

As μ is unknown, we have only the direction of plastic strain and not its size.



On a graphical representation, if we consider a point along the yield function, it will have a certain plastic strain occurring $\delta\varepsilon^p$. Elasto-plasticity tells us that the plastic strain is directed orthogonally to a certain function, that is the potential function g.

In order to know the value of the plastic strain, we can ~~use~~ combine the definition of total strain, that is the sum of elastic and plastic strains (1)

$$\{\delta\varepsilon\} = \{\delta\varepsilon\}^e + \{\delta\varepsilon\}^p \qquad (1)$$

$$= [D]\{\partial\varepsilon\} - \frac{[D]\frac{\partial g}{\{\partial\sigma\}}\frac{\partial f^T}{\{\partial\sigma\}}[D]}{\frac{\partial f^T}{\{\partial\sigma\}}[D]\frac{\partial g}{\{\partial\sigma\}}}\{\partial\varepsilon\} =$$

→ vector $\partial g/\{\partial\sigma\}$ is moved before the ratio because the ratio μ is a scalar

$$= \left([D] - \frac{[D]\frac{\partial g}{\{\partial\sigma\}}\frac{\partial f^T}{\{\partial\sigma\}}[D]}{\frac{\partial f^T}{\{\partial\sigma\}}[D]\frac{\partial g}{\{\partial\sigma\}}}\right)\{\partial\varepsilon\}$$

The expression has the same form of

$$\{\partial\sigma\} = [D]^{ep}\{\partial\varepsilon\}$$

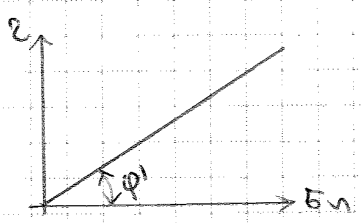
so the expression in the brackets represents the elasto-plastic stiffness matrix.

$$[D]^{ep} = [D] - \frac{[D]\frac{\partial g}{\{\partial\sigma\}}\frac{\partial f^T}{\{\partial\sigma\}}[D]}{\frac{\partial f^T}{\{\partial\sigma\}}[D]\frac{\partial g}{\{\partial\sigma\}}} \qquad (7)$$

In conclusion, when an elasto-plastic model is used, we have to check whether stress state is inside the yield surface or not.
In the first case, we use only the elastic stiffness matrix.
In the second case, we use the elasto-plastic stiffness matrix that derives from a combination of the plastic potential function and the yielding function.

Mohr-Coulomb model and elastic-perfect-plasticity

Mohr-Coulomb model is the most used failure criterion in geotechical engineering.

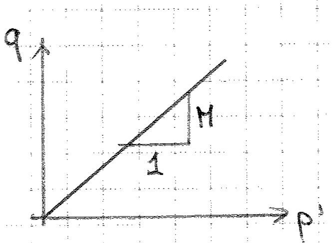


This model is typically used in the $\tau-\sigma$ space and it is given by a slope inclined of $\tan\varphi'$

$$\tau = \sigma_n \tan\varphi'$$

This criterion can also be expressed in terms of ratio of principal stresses

$$\frac{\sigma'_a}{\sigma'_r} = \frac{\sigma'_1}{\sigma'_3} = \frac{1+\sin\varphi'}{1-\sin\varphi'}$$



or in terms of ratio between deviatoric stress and mean stress

$$\frac{q}{p'} = M = \frac{6\sin\varphi'}{3-\sin\varphi'}$$

If we assume there is no volume strain in the plastic mechanism, we'll have $M^* = 0$.

So, the term M^* represents the RATIO BETWEEN VOLUMETRIC STRAIN AND DEVIATORIC STRAIN.
From a mathematical point of view, M^* defines the slope of the family of lines that define plastic potential function; from a physical point of view, it is a dilatancy (ratio volumetric strain - deviatoric strain) and we can write it as

$$M^* = \frac{6\sin\psi}{3-\sin\psi} \qquad \psi = \text{angle of dilation}$$

Once we have defined the slope of the plastic potential function, we can find its expression by using equations (3) and (9)

$$\{\delta\varepsilon\}^p = \mu \frac{\partial g}{\{\partial\sigma\}} \qquad (3)$$

$$\begin{pmatrix} \delta\varepsilon_p^p \\ \delta\varepsilon_q \end{pmatrix} = \mu \begin{pmatrix} \frac{\partial g}{\partial p'} \\ \frac{\partial g}{\partial q} \end{pmatrix} = \mu \underbrace{\begin{pmatrix} -M^* \\ 1 \end{pmatrix}}_{(9)}$$

From this, we can derive the plastic potential function.

$$g(\{\sigma\}) = g(p'; q) = q - M^* p + k \qquad (10)$$

Once we have defined the plastic potential function, we can write the STIFFNESS MATRIX, that gives the relationship between stress increase and strain increase.
We start from the general expression of the elasto-plastic stiffness matrix (4).

$$\{\delta\sigma\} = [D]\{\delta\varepsilon\} - \mu[D]\frac{\partial g}{\{\partial\sigma\}} \qquad (4)$$

Elastic stiffness matrix can be written by using bulk and shear modulus K & G.

$$[D] = \begin{bmatrix} K & 0 \\ 0 & 3G \end{bmatrix}$$

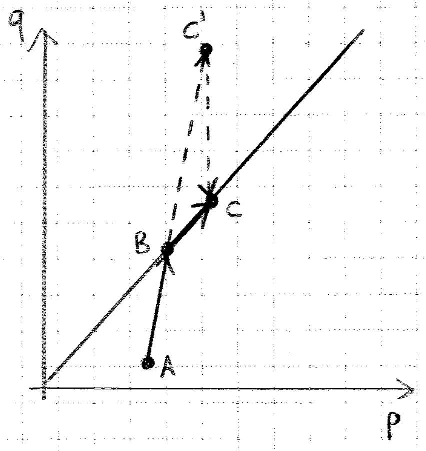
$$[D]^{ep} = \underbrace{\begin{bmatrix} k & 0 \\ 0 & 3G \end{bmatrix}}_{A} - \frac{1}{kMM^* + 3G} \underbrace{\begin{bmatrix} MM^*k^2 & -3M^*Gk \\ -3MGk & 9G^2 \end{bmatrix}}_{B} \quad (15)$$

Elasto-plastic stiffness matrix presents 2 components

→ A, that is the elastic stiffness matrix

→ B, that is a part only related with plasticity - it activates only when there is yielding.

Interpreting it from a practical point of view, we can see it as given by an ELASTIC PREDICTOR and a PLASTIC CORRECTION.



Let's imagine an initial state A in qp' plane.

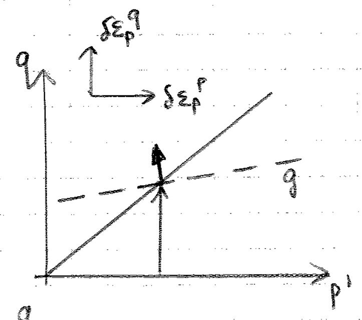
When we apply a load, we follow a certain stress path and initially we get only an elastic strain until state B, that lies along the yield surface. Then, if we continue to load, we fall into the elasto-plastic field.
If we compute with the elastic stiffness matrix only, we will go in the point C' but this point is not acceptable because it falls in the non allowed zone. So, point C' - obtained from an elastic prediction - is an impossible stress state and we have to apply the plastic correction. This brings the stress state back to the yield surface, at the point C. At the end, the path follows a curved shape and we have a different slope with respect to the initial zone.

The plastic correction includes elastic parameters (k, G), dilatancy (M^*) and Mohr-Coulomb parameter (M).

Then, elasto-plastic stiffness matrix can also be written in a different way.

$$[D]^{ep} = \frac{3Gk}{kMM^* + 3G} \begin{bmatrix} 1 & M^* \\ M & MM^* \end{bmatrix} \quad (16)$$

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 149 di 218

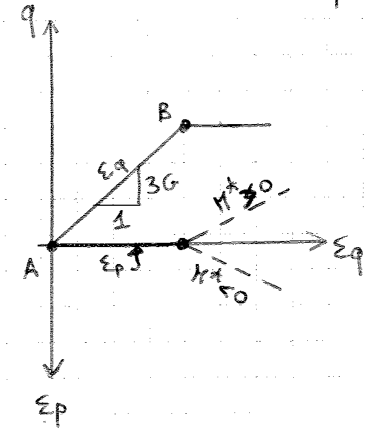


Deviatoric strain increases with a ratio of 3G

$$\varepsilon_q^e = \frac{q}{3G}$$

When we get to point B, plastic strain will depend on M^*, due to the relation (8).

$$\frac{\delta\varepsilon_p^p}{\delta\varepsilon_q^p} = -M^*$$



If $M^* = 0$, as plastic strain vector is orthogonal to the curve, plastic volume strain will be zero.
If $M^* > 0$, we'll get a negative plastic volume strain, i.e. a volume increases.

So, if we want to characterise a material that has compression at yielding, we will put

$$M^* < 0$$

Ex

A silty soil in drained conditions ~~was~~ is characterised, in its behaviour, with the elastic perfectly plastic Mohr-Coulomb model and the following parameters are got.

→ elastic parameters $E = 12{,}5$ MPa $\nu = 0{,}25$

→ yield function $\varphi' = 30°$

→ plastic potential function $\psi = 0°$

Firstly, evaluate the elastic and the elasto-plastic stiffness matrices.

Elastic matrix is used when stress state is above the yield function and it is given by the expression

$$[D] = \begin{bmatrix} k & 0 \\ 0 & 3G \end{bmatrix}$$

Bulk modulus k is

$$k = \frac{E}{3(1-2\nu)} = \frac{12{,}5}{3(1-2\cdot 0{,}25)} [\text{MPa}] = 8{,}33 \text{ MPa}$$

where $\tilde{D}$ is the elastic stiffness matrix or the elasto-plastic one.

We will progress step by step and, to choose which matrix to use, we evaluate the yield function

$$f = q - Mp'$$

→ if $f < 0$, we will use elastic stiffness matrix D.

→ if $f = 0$, we will use elasto-plastic stiffness matrix D^{ep}.

We can also work with the stress ratio

$$\eta = \frac{q}{p'}$$

If the ratio η is smaller than M, the behaviour will be elastic; otherwise if η is equal to M, the behaviour will be elasto-plastic.

So, we can set a table with different columns.

→ volume strain ε_p, which is all zero because no volume strain is applied.

→ deviatoric strain, that starts from zero and it is increased step by step, with steps of $2 \cdot 10^{-3}$.

→ increase in mean stress $\delta p'$ and increase in deviatoric stress δq:

when we are in elastic field - i.e. $f < 0$ or $\eta < 1{,}2$ -, these increases are calculated by using the elastic stiffness matrix.

$$\begin{pmatrix} \delta p' \\ \delta q \end{pmatrix} = \begin{bmatrix} K & 0 \\ 0 & 3G \end{bmatrix} \begin{pmatrix} \delta \varepsilon_p \\ \delta \varepsilon_q \end{pmatrix} \quad \rightarrow \{\delta \sigma\} = [D]\{\delta \varepsilon\}$$

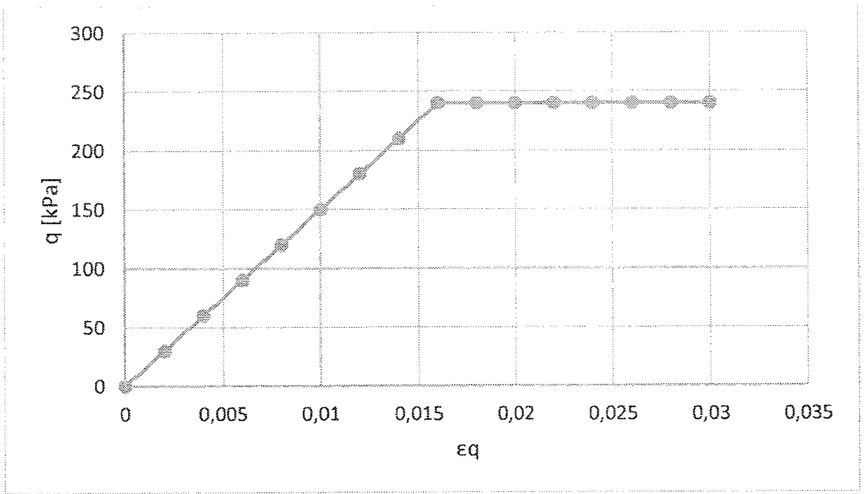
$$\Rightarrow \begin{aligned} \delta p' &= K \delta \varepsilon_p = 8'330 \,[kPa] \cdot 0 = 0 \text{ kPa} \\ \delta q &= 3G \delta \varepsilon_q = 15'000 \,[kPa] \cdot 0{,}002 = 30 \text{ kPa} \end{aligned}$$

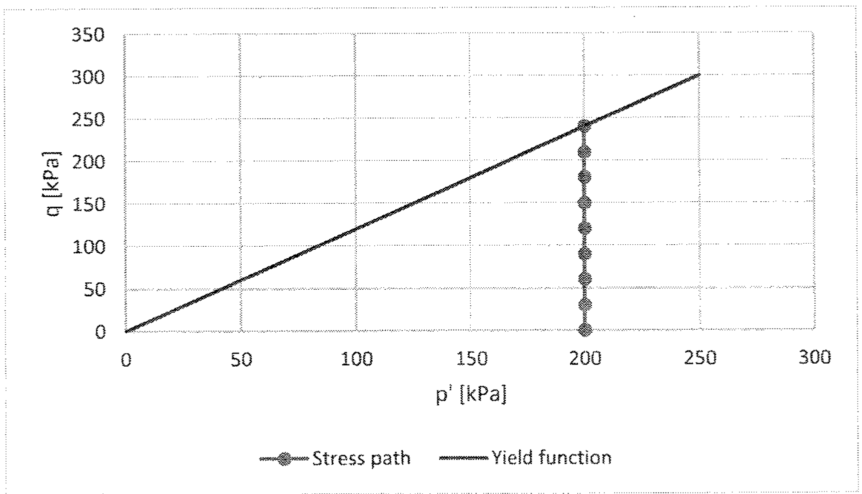
So, everytime we apply the elastic stiffness matrix, we have an increase of deviatoric stress equal to 30 kPa.

When we use elasto-plastic stiffness matrix, the stress increases are

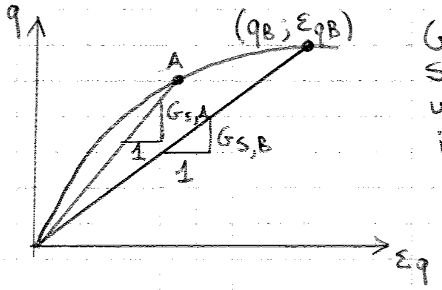
$$\begin{pmatrix} \delta p' \\ \delta q \end{pmatrix} = \begin{bmatrix} 8{,}33 & 0 \\ 10 & 0 \end{bmatrix} \begin{pmatrix} \delta \varepsilon_p \\ \delta \varepsilon_q \end{pmatrix} \quad \rightarrow \{\delta \sigma\} = [D]^{ep} \{\delta \varepsilon\}$$

εp	εq	δp'	δq	p'	q	f	η	M
0	0	0	0	200	0	-240	0	1,2
0	0,002	0	30	200	30	-210	0,15	1,2
0	0,004	0	30	200	60	-180	0,3	1,2
0	0,006	0	30	200	90	-150	0,45	1,2
0	0,008	0	30	200	120	-120	0,6	1,2
0	0,01	0	30	200	150	-90	0,75	1,2
0	0,012	0	30	200	180	-60	0,9	1,2
0	0,014	0	30	200	210	-30	1,05	1,2
0	0,016	0	30	200	240	0	1,2	1,2
0	0,018	0	0	200	240	0	1,2	1,2
0	0,02	0	0	200	240	0	1,2	1,2
0	0,022	0	0	200	240	0	1,2	2,2
0	0,024	0	0	200	240	0	1,2	3,2
0	0,026	0	0	200	240	0	1,2	4,2
0	0,028	0	0	200	240	0	1,2	5,2
0	0,03	0	0	200	240	0	1,2	6,2





Finally, we have to determine the secant shear modulus corresponding at the different strain steps.



Given a curve of response of the material, secant shear modulus is the slope of the secant line which connects a point to the origin and is calculated as

$$G_s = \frac{q - q_0}{\varepsilon_q - \varepsilon_{q,0}}$$

OBSERVATION: in this case, there is no plastic volume strain because if we use the definition of plastic ~~potential~~ strain (3), we get

$$\delta\varepsilon_p^p = \mu \frac{\partial g}{\partial p'}$$

$$\delta\varepsilon_q^p = \mu \frac{\partial g}{\partial q}$$

In Mohr-Coulomb model, potential function is

$$g(p'; q) = q - M^* p + k \qquad (10)$$

$$\Rightarrow \begin{aligned} \delta\varepsilon_p^p &= \mu \frac{\partial g}{\partial p'} = \mu \cdot (-M^*) = -\mu M^* \\ \delta\varepsilon_p^q &= \mu \frac{\partial g}{\partial q} = \mu \cdot 1 = \mu \end{aligned}$$

In this case, M^* is zero so also plastic volume strain will be zero.

Ex

A material is characterised by the following parameters.

→ elastic parameters $E = 12{,}5$ MPa $\quad \nu = 0{,}25$

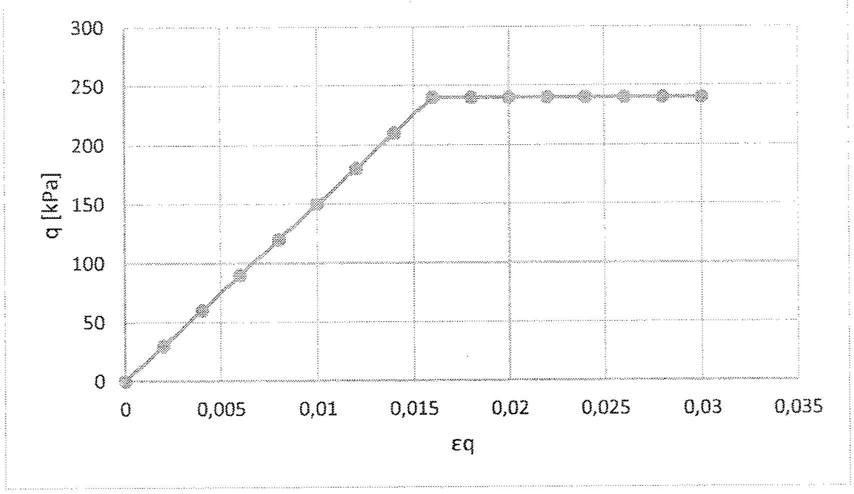
→ yield function $\varphi' = 30°$

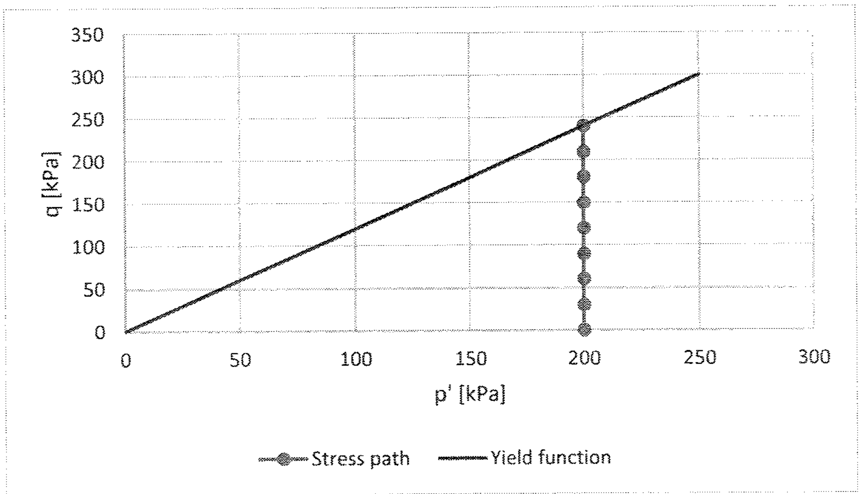
→ plastic potential function $M^* = 0{,}5$

Evaluate the response of the material by a stress path into which we start from an isotropic stress state

$$p' = 200 \text{ kPa} \qquad q = 0 \text{ kPa}$$

εp	εq	δp'	δq	p'	q	f	η	M
0	0	0	0	200	0	-240	0	1,2
0	0,002	0	30	200	30	-210	0,15	1,2
0	0,004	0	30	200	60	-180	0,3	1,2
0	0,006	0	30	200	90	-150	0,45	1,2
0	0,008	0	30	200	120	-120	0,6	1,2
0	0,01	0	30	200	150	-90	0,75	1,2
0	0,012	0	30	200	180	-60	0,9	1,2
0	0,014	0	30	200	210	-30	1,05	1,2
0	0,016	0	30	200	240	0	1,2	1,2
-0,001	0,018	0	0	200	240	0	1,2	1,2
-0,002	0,02	0	0	200	240	0	1,2	1,2
-0,003	0,022	0	0	200	240	0	1,2	1,2
-0,004	0,024	0	0	200	240	0	1,2	1,2
-0,005	0,026	0	0	200	240	0	1,2	1,2
-0,006	0,028	0	0	200	240	0	1,2	1,2
-0,007	0,03	0	0	200	240	0	1,2	1,2





© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 159 di 218

$$\delta\varepsilon_p = -0{,}5\,\delta\varepsilon_q = -0{,}5 \cdot 0{,}002 = -0{,}001$$

⇒ by setting a dilatancy equal to 0,5, the material tends to expand at yield

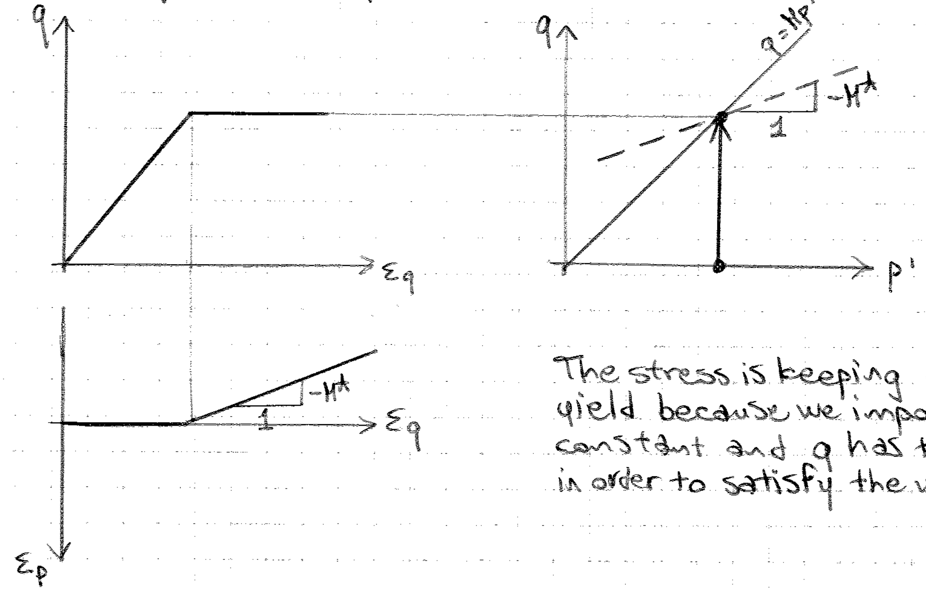
So, if we compute the quantities ε_p, ε_q, etc. (like the previous example) and we built a table like the following one

ε_p	ε_q	$\delta p'$	δq	p'	q	f	η
⋮	⋮	⋮	⋮	⋮	⋮	⋮	⋮

(~~figure~~

we can see than, in elastic field, ε_p is zero and, in plastic field, there is an increase of volume.

Then, we plot the response.



The stress is keeping constant at yield because we impose that p' is constant and q has to be constant in order to satisfy the yield condition.

If we divide the stress path into steps of deviatoric strain equal to $2 \cdot 10^{-3}$, in elastoplastic field the increases of stresses are

$$\delta p' = 3{,}12 \cdot 0{,}002 = 6{,}2 \text{ kPa}$$

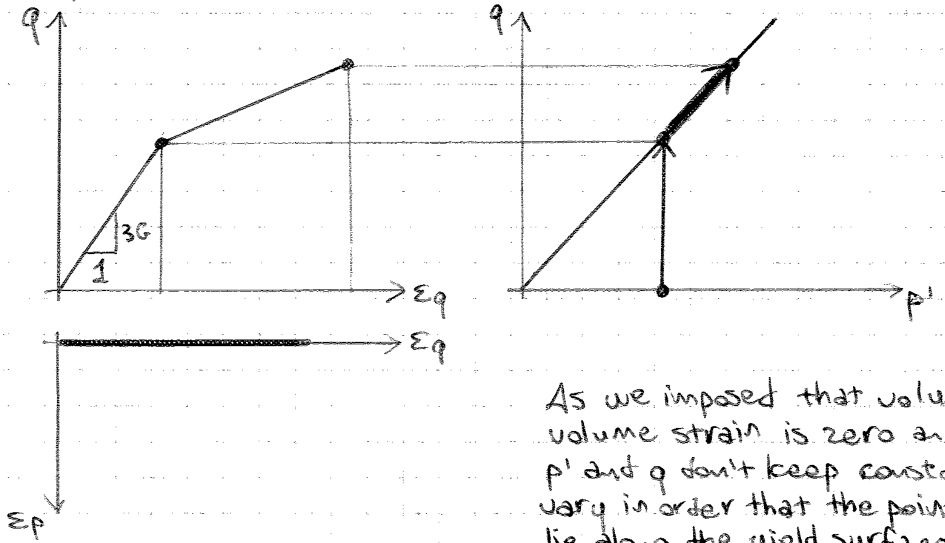
$$\delta q = 3{,}75 \cdot 0{,}002 = 7{,}5 \text{ kPa}$$

We can notice that the ratio between these increments is

$$\frac{\delta q}{\delta p'} = \frac{7{,}5}{6{,}2} \sim 1{,}2 = M$$

because the stress state representative point, at yield, has to move along the yield surface, which slope is M.

Finally, we plot the response.



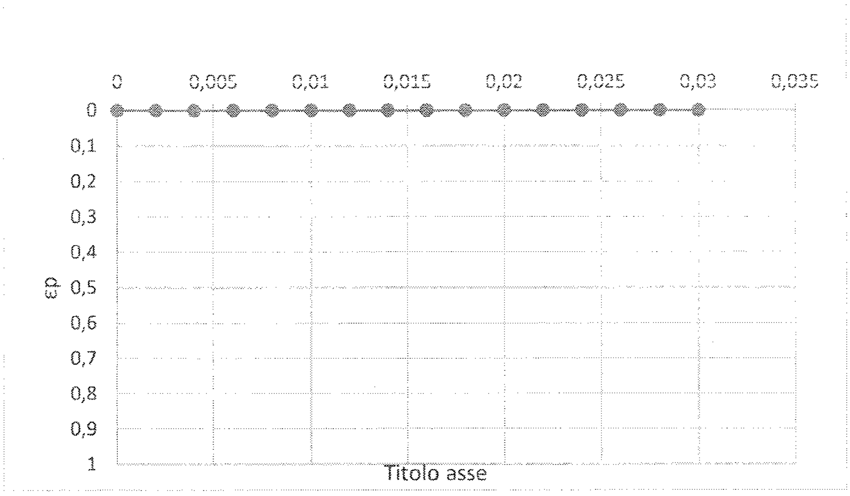
As we imposed that volume is constant, volume strain is zero and, at yielding, p' and q don't keep constant but they vary in order that the point continues to lie along the yield surface

In soils, a condition where volume is constant is represented by UNDRAINED CONDITION

⇒ that is the expected behaviour of a soil in undrained conditions

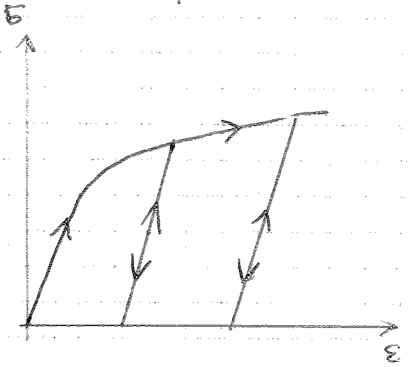
OBSERVATION

These answers are not the real behaviour because they are evaluated basing on a value of ψ that we have decided on the base of experience. Indeed, <u>results depend on the model used and we have to choose the one that represents in the best way the problem.</u>



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 165 di 218

This aspects can also be appreciated in stress-strain plot



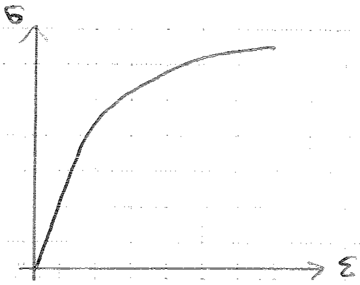
→ there are different levels of stress where plastic strain starts in the different cycles.

→ during the unloading, ideally there is only a linear elastic behaviour.
It means that there is something reversible

So, as said in the elastic perfectly plastic models, when stress state is within the yield surface, the behaviour will be elastic.

The main aspect is that yielding hasn't a meaning of failure in this case and the geometry of yield function varies

⇒ hardening is the increase of yield function and it is related to the fact that the material becomes more resilient with plastic strains.



From a mathematical point of view, hardening is described with the following yield function

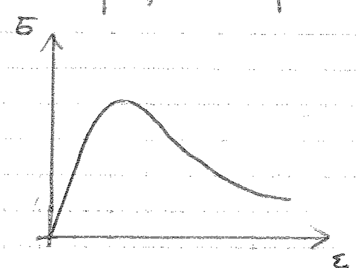
$$f(\sigma ; \chi)$$

χ = hardening parameter

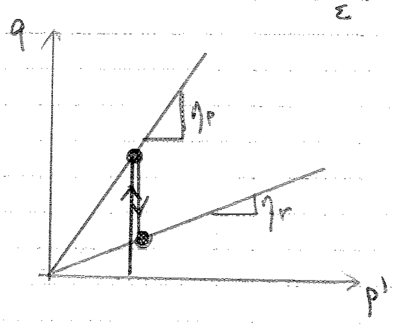
When $f = 0$, plastic strain occurs.
So, the yield function is no more constant in p'q space and it evolves.

There are different kinds of hardening and we will deal with ISOTROPIC HARDENING.

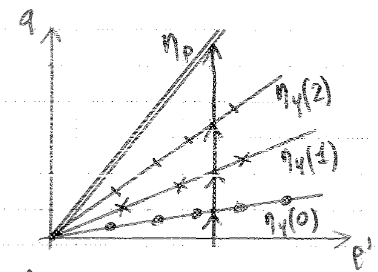
Another condition is softening, typical behaviour of overconsolidate clays, shear joints and dense sands.



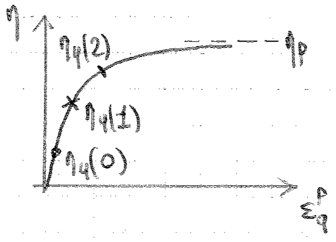
This is a condition where there is a DECREASE OF THE YIELD SURFACE and, by loading and unloading, we get to a smaller stress level when plastic strain increases.



A typical example is a material with a peak shear angle and a residual shear angle.
If we apply a stress path, at the beginning the material will show a big strength.
Then there is a decrease and peak is not allowed anymore - we are in residual conditions and the points above its limit aren't allowed -, so the stress path will go down.



If we simulate a test, at the beginning we have an elastic behaviour.
Then, we reach the yield surface and, when plastic strain increases, we can evaluate different values of η_y.

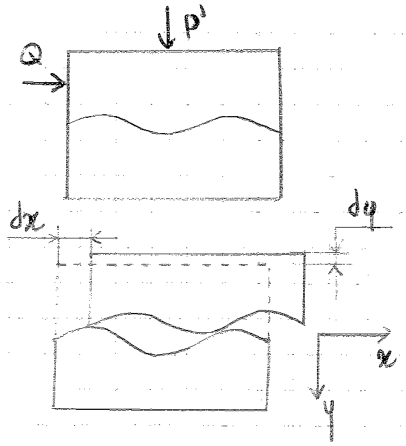


If we compare the plastic deviatoric strain ε_q^p with the stress ratio η, we'll get a certain curve that represents the hardening law.

The value of η_y has an upper limit, given by the failure value

Once we have defined the hardening law, we need a PLASTIC POTENTIAL FUNCTION, that gives the direction of plastic strains and so the relationship between plastic volume strain and plastic deviatoric strain.

In this case, the expression of this function derives from an experiment about dilatancy.



A specimen in a shear box is subjected to a vertical force P' and a horizontal force Q - shearing force.
As the specimen presents an ondulate interface, when shearing force is applied, it gets to a condition with horizontal and vertical displacements.

If we compute displacements in a certain way, we notice that dy is negative.
This behaviour is typical of dense sands where, in order to shear, the two parts have to climb one over the other.

Taylor, in 1948, evaluated the work done on the shear box.

$$\delta W = P'dy + Q dx$$

This energy is assumed to be dissipated by a friction mechanism.

$$\delta W = \mu P' dx \qquad \mu = \text{friction coefficient}$$

Comparing the work applied and the work dissipated, we get

$$P'dy + Q dx = \mu P' dx$$

By comparing these two equations, we get the following conditions.

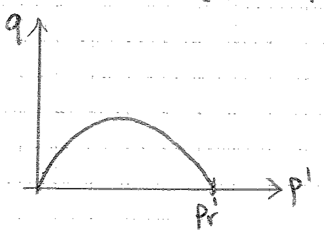
$$\frac{\partial g}{\partial p'} = M - \eta .$$

$$\frac{\partial g}{\partial q} = 1$$

We integrate these equation and we get the potential function.

$$g(p';q) = q - Mp' \ln\left(\frac{p_r'}{p'}\right)$$

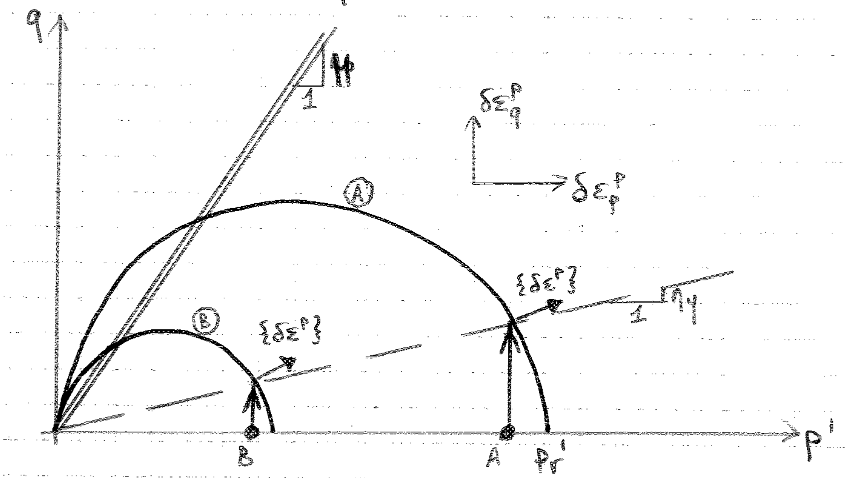
$$g(p';q) = q - Mp' \ln\left(\frac{p_r'}{p'}\right) \qquad (21)$$



The potential function has a certain shape and p_r' is a parameter that defines its size.

potential function ~~are~~ is defined with an arbitrary constant

How can be applied this potential function?

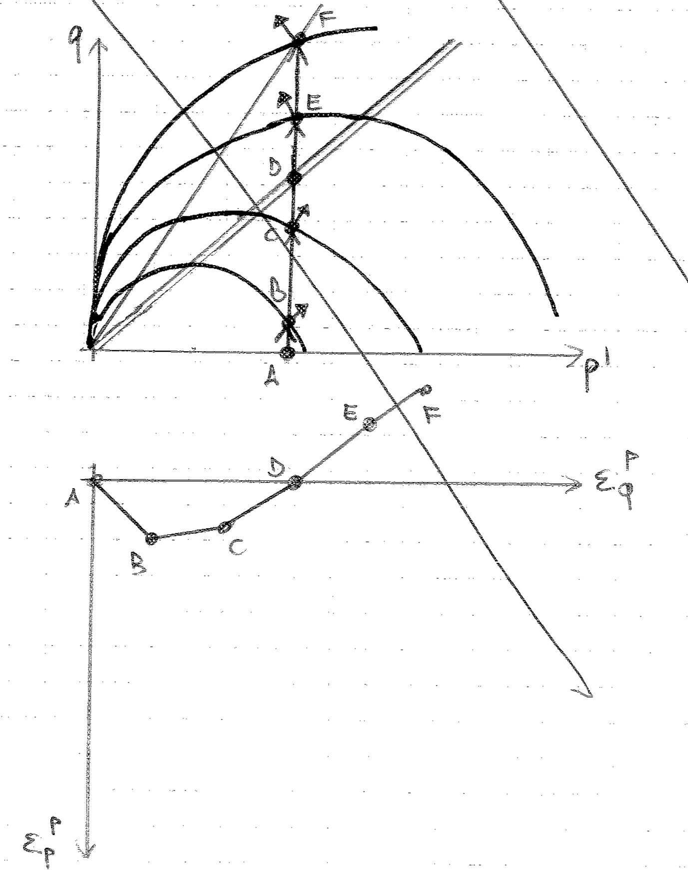


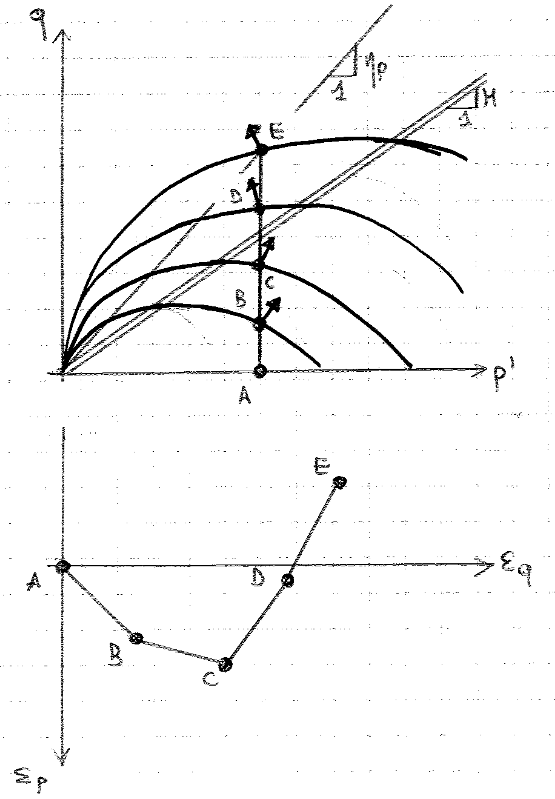
Starting from point A, we apply a shearing load and we touch the yield surface.
Here, we have the potential function curve A and the direction of the plastic strains is orthogonal to this shape. In this way, we define ε_p^p and ε_q^p and we notice that material is contracting ($\delta\varepsilon_p^p > 0$).

If we start from point B, once we get to the yield surface, we have plastic strains.
These can't be evaluated immediately, as curve A doesn't pass there.
Remembering that potential function is defined with an arbitrary constant, we can define a new plastic potential function by changing p_r'. It will have the same shape, but smaller.

→ $\eta_p > M$

Starting from point A, once we get at yield surface, we have positive volume ~~str~~ plastic strain.
Then, at point C, we have negative volume strain and it means that material begins to increase its volume.





In conclusion, the ingredients used to define an elasto-plastic model.

→ ELASTIC STRESS-STRAIN RELATIONSHIP

→ YIELD FUNCTION

→ PLASTIC POTENTIAL FUNCTION

→ HARDENING FUNCTION

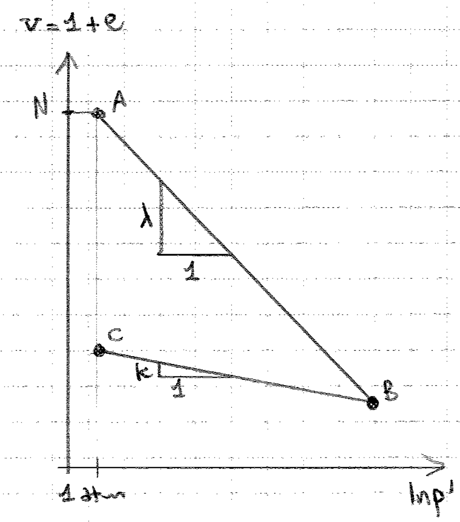
In defining the stiffness matrix, a fundamental element is the CONSISTENCY CONDITION, based on the value χ (it is coincident with η_y), that indicates that, when there are plastic strains, we always will move on the yield surface.

Modified cam clay model

1 This model belongs to the class of CRITICAL STATE MODELS

2 Critical state

To define the critical state, we have to start from an evidence about the compression behaviour of soils.



A remolded soil - reconstituted soil - specimen is loaded into a condition of ISOTROPIC LOAD.
It means that p' is changing and q is nil.

$p' = var. \qquad q = 0$

The response is represented in a diagram, where

→ y axis = specific volume v

$v = 1 + e \qquad e$ = void ratio

→ x axis = natural logarithm of mean stress p'.

The soil is reconstituted:

the soil is made like a powder, a little bit of water is added and we get a mud. In this way, the history of the soil is erased.

The stress path starts at the atmospheric pressure

$p' = 1$ kPa

and there the soil has a relatively high specific volume, called N.

Then the specimen is loaded with an isotropic load and its behaviour, in the plane v - lnp', is represented by a straight line, which slope is indicated by λ.
The parameter λ is similar to the geotechnical parameter of compression index C_c.

The load proceed from point A to point B and then the soil is unloaded and the isotropic stress is reduced.
The behaviour is represented by the straight line BC, with slope k.

So, the behaviour can be divided into 2 lines

→ AB, called isotropic compressive line (ICL)