NUMERO: 2075A - ANNO: 2017

A P P U N T I

STUDENTE: Aimar Mauro

MATERIA: Metodi numerici per l'ingegneria geotecnica (parte II), Teoria + esercizi - prof. Barla, Musso

DISTINCT ELEMENT METHOD

Introduction: continuum and discontinuum

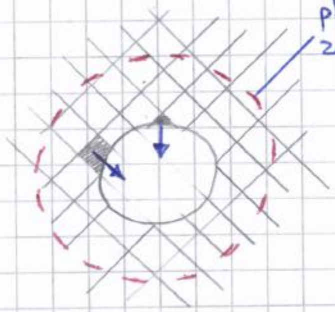
In rock mechanics, we can distinguish 3 cathegories of rock mass

→ CONTINUUM ROCK MASS, e.g. soil, soft rocks or stiff and massive rocks.

→ EQUIVALENT CONTINUUM ROCK MASS, e.g. jointed rock mass - it is so much jointed that is almost a soil

→ DISCONTINUUM ROCK MASS e.g. not so much jointed rock mass - spacing between joints has the same order of magnitude of the dimension of the geotechnical opera.

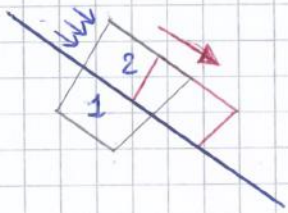
In the last case, we have to introduce new numerical methods, otherwise if we use methods of continuous framework, we will get the wrong answer.



For instance, a tunnel excavated in a continuous isotropic elasto-plastic media is expected to give rise to a plastic radius.

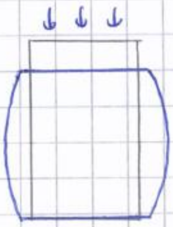
If rock mass has 2 different joint sets, by adopting a discontinuum approach, we can see same blocks that are going to move and fall down.
If a continuous approach is adopted, we will see only a plastic radius and not the fall of those blocks.

A first method is the LIMIT EQUILIBRIUM METHOD (LEM), which consists of computing the weight of the wedge and designing the force that a support system should apply.
Other systems are the Discrete Element Method (DEM) and the Finite-Discrete Element Method (FDEM)

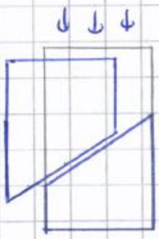


In a DISCONTINUUM MODEL, if a pressure is applied on block 2, this will slide along the joint. Moreover, the 2 blocks can be separated - sliding and separation can't happen in a continuum model.
So, interaction in a discontinuum model is more complicated because we have to take into account that they may slide, separate or also get in contact during the computation.

→ response of a laboratory specimen (phenomenological point of view)

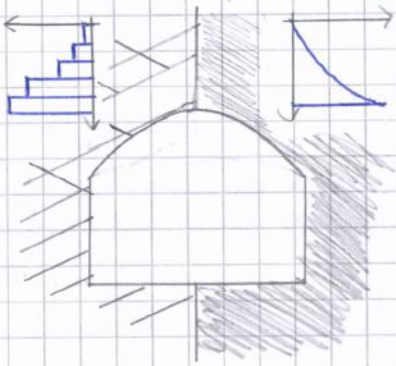


If we model the response of a laboratory specimen with a CONTINUUM MODEL, it will deform and assume a barrel shape. Its response will be characterised by stresses and strains.
This could be a good model for clays.



If a DISCONTINUUM MODEL is adopted, the specimen is divided by a surface into 2 elements, one sliding on the other.
The response will be characterised by forces and displacements.
This could be a good model for hard rock.

→ excavation response (geotechnical and structural point of view)

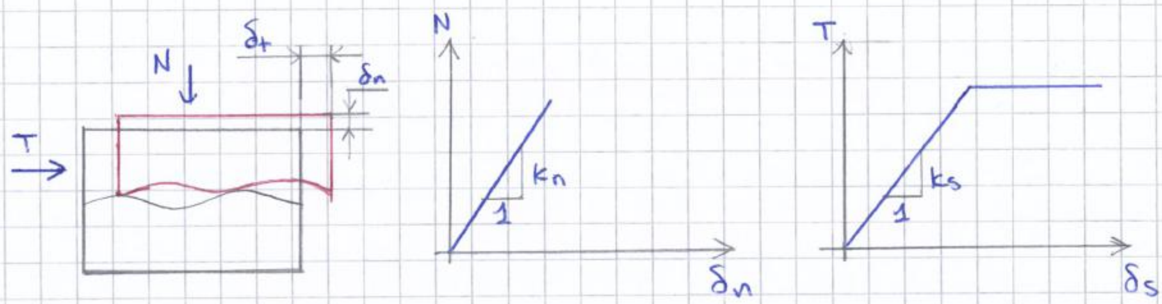


If we model the response to an excavation with the CONTINUUM APPROACH, the radial displacement will be maximum at the contour and then it decreases to zero in a continuous way.

By using DISCONTINUUM MODEL, some element will fall down and we will see big displacements at the contour, whereas other elements move less.

⇒ completely different responses

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 5 di 242



To determine these parameters, we use a DIRECT SHEAR TEST:

firstly, a normal load N is applied and we see a normal displacement δ_n. If the rock is stiff enough, this normal displacement is due to a contraction along the joint. From this, we get the joint deformability in normal direction and the normal stiffness k_n.
Once the normal load is applied, we start to apply the shear force T and we evaluate the shear displacement δ_s.

When there is sliding, the correspondent shear force will be the shear strength of the joint under the normal stress.
If we do many tests on different levels of normal stress, we get many points in the $\tau-\sigma_n$ plane and we can sketch a line.

If tests are referred to an artificial joint - very pulished -, this lines intersects the origin and the angle is called BASE FRICTION ANGLE φ_b.

If normal joints, with rough surfaces, are tested, results are different and we define a non-linear shape, that is the BARTON CRITERION.
This criterion requires the evaluation of 3 parameters

→ JRC, that represents joint roughness and it is obtained from direct observation of the joint

→ JCS, that is related to Schmidt hammer and, in general, it is similar to unconfined compressive strength of intact rock

→ φ_b, from direct shear test

It means that the equation of motion is solved by an explicit approach - i.e. variables are function of time - and from it we find forces and displacements at the interfaces of the blocks.
In this approach, within a timestep, we solve the equations of motion and constitutive equations, by assuming that velocities and accelerations are constant within the timestep.
Of course, the timestep has to be sufficiently small that, during a single step, disturbances can't propagate between one discrete element and its immediate neighbours as this would need a larger time to occur

The flow chart of explicit approach for DEM is the following.

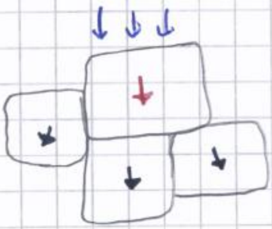
1) For each timestep, we compute forces for each block, due to stresses, point or body forces applied.
   Velocities and displacements can vary linearly, while stresses and strains remain constant in a single element, as in the FDM.

2) Equation of motion are integrated to determine new values of velocity and the displacements, based on the applied forces.

3) Strain rates are computed from the velocities

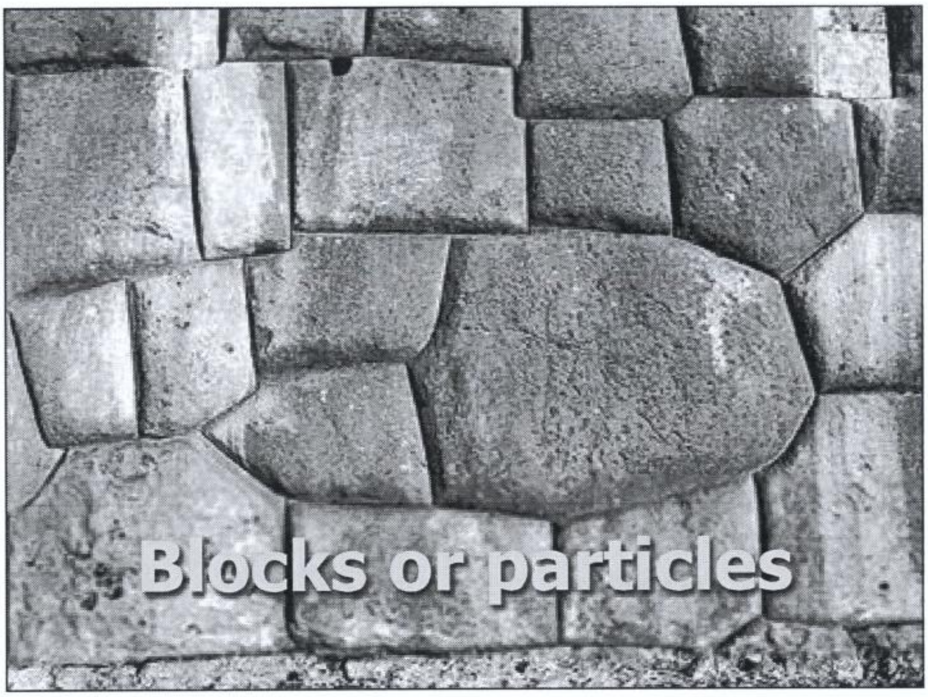
4) By using constitutive equations, stresses are computed from strain rates so that Forces on the blocks can be computed during subsequent timestep.

In this way, we can simulate the propagation of the perturbation within the blocky system:

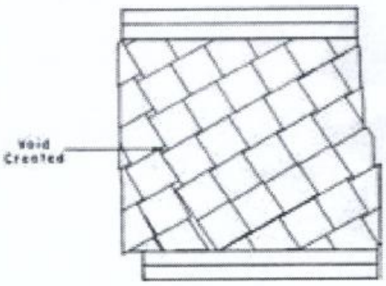


by applying a pressure on the system, we get a certain stress level.
From this, we compute strains and displacements of blocks. Timestep is so short that perturbation is applied only on the first element and there is a velocity only on it.
Then, from velocity, we compute the pressure applied from the first element through joints to the neighbourings elements.
In the next step, we get new velocities

In this way, we see hvelocity propagation in the elements.



BLOCKS OR PARTICLES



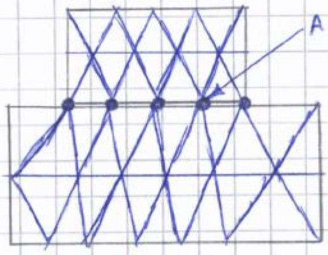
3 Kind of blocks

In the method, we can use 2 kinds of blocks.

On one side, we can use rigid blocks:

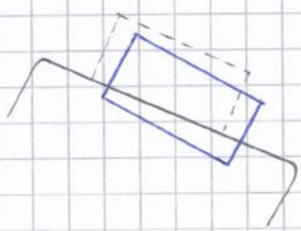
in this case, displacements in a block assembly occur only along sliding planes.

Rigid blocks interact by two contact points at the boundary of the common segment.

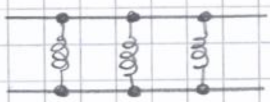


On the other side, also deformable blocks can be used.
In this case, intact block material deformation is not negligible compared to displacements along discontinuities.
Blocks are discretized with finite elements and discrete elements interact thanks to special nodes. These nodes are special because along the common segment there are nodes belonging to one discrete element and nodes belonging to the other discrete element, that occupy the same position - one node belongs to 3 finite elements and the other node, in the same position, belongs to other 3 finite elements - node A - and we have different forces and displacements in that point.

Moreover, FDM models allow OVERLAPPING of blocks and this is useful to simulate NORMAL STIFFNESS OF THE JOINT.



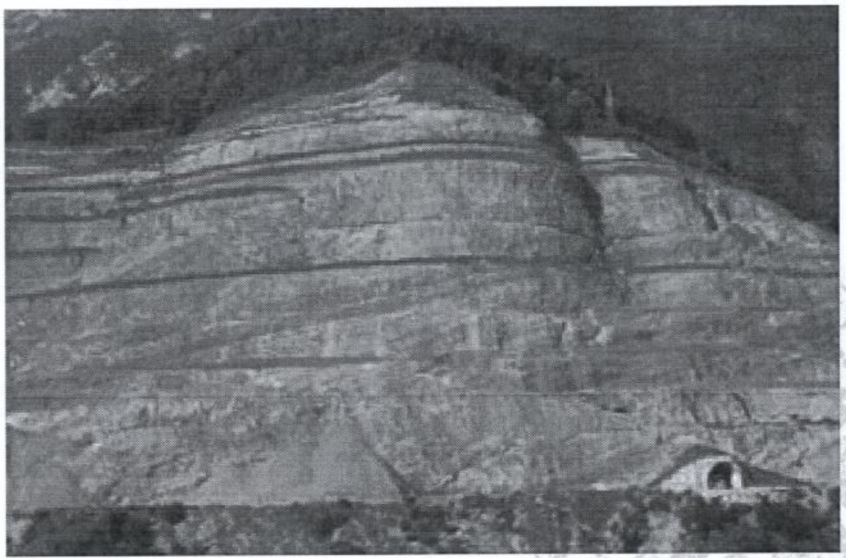
If joint is very rigid, nothing will happen if a load is applied.
With a joint with a finite normal stiffness, elements will overlap a bit.



In this case, nodes of different elements are superposed from the geometrical point of view - i.e. same coordinates - and each couple of nodes is connected by springs.
Springs mean deformability but, in order to introduce deformability, we have to allow one node to go below the other one, that is allow overlapping, from the geometrical point of view.

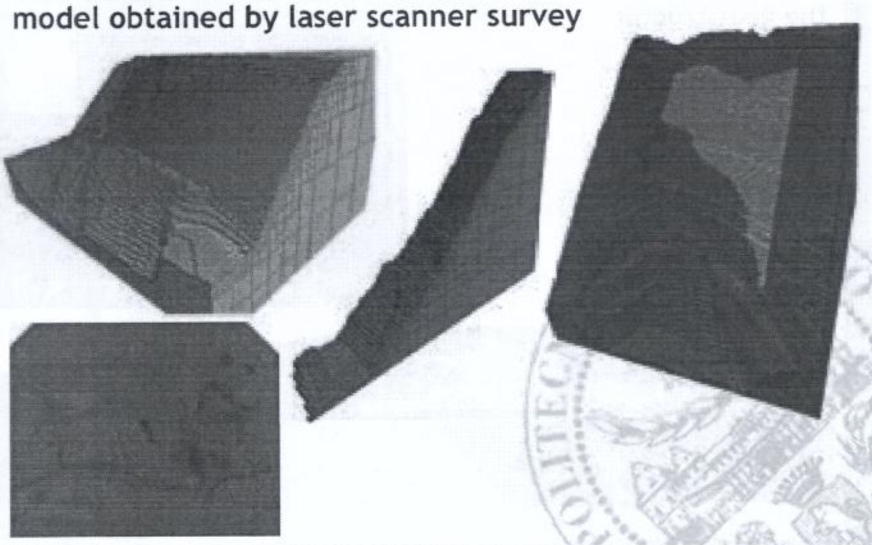
With this aspect, we can simulate normal interaction.

GEOMETRY



GEOMETRY

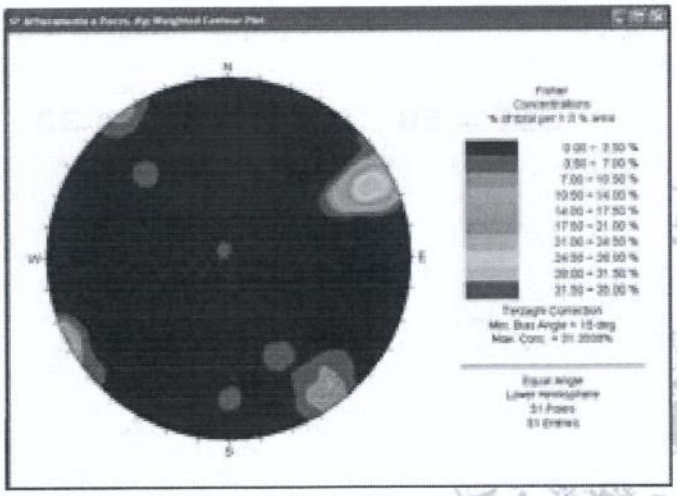
Geometry generated from a digital elevation model obtained by laser scanner survey



GEO-STRUCTURAL SURVEY

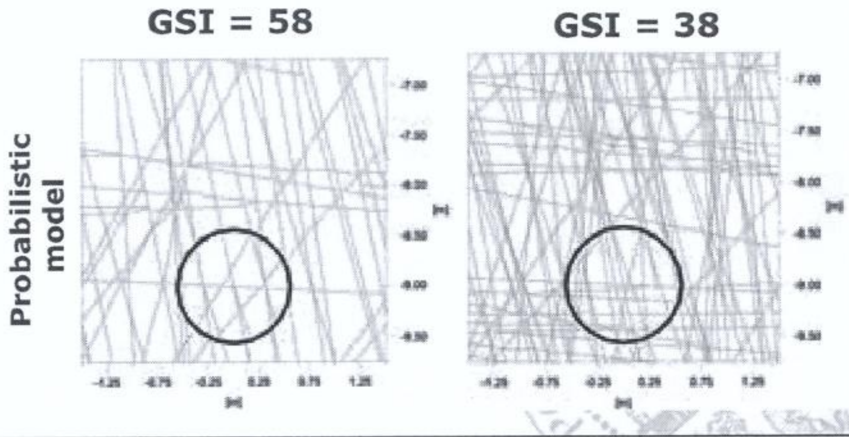
STEREOMAP

Analyse them...

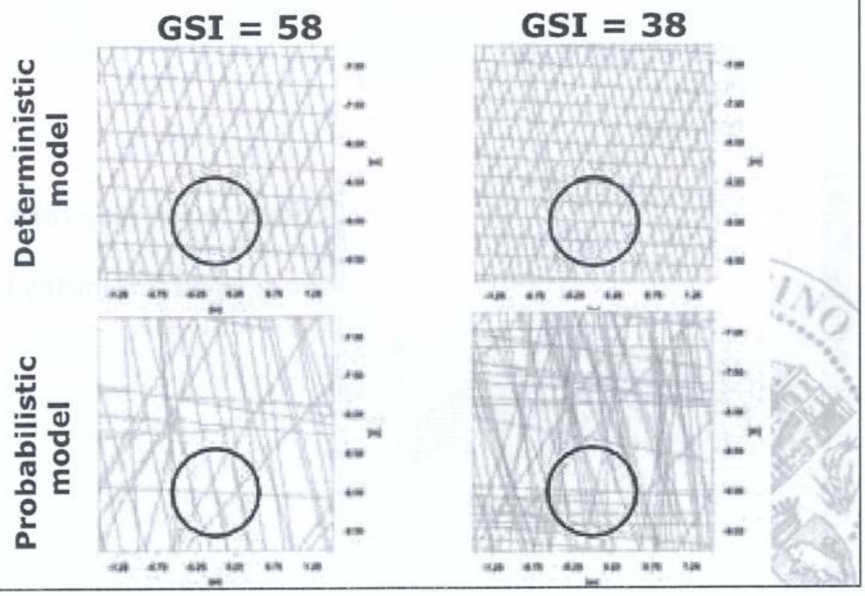


BUILDING A DEM MODEL

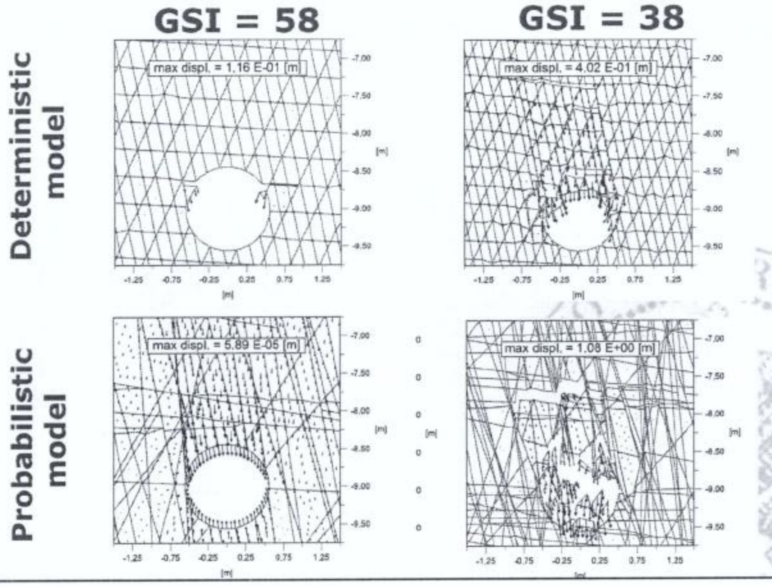
Alternatively, we may use statistical parameters (mean values of the parameters and standard deviation).



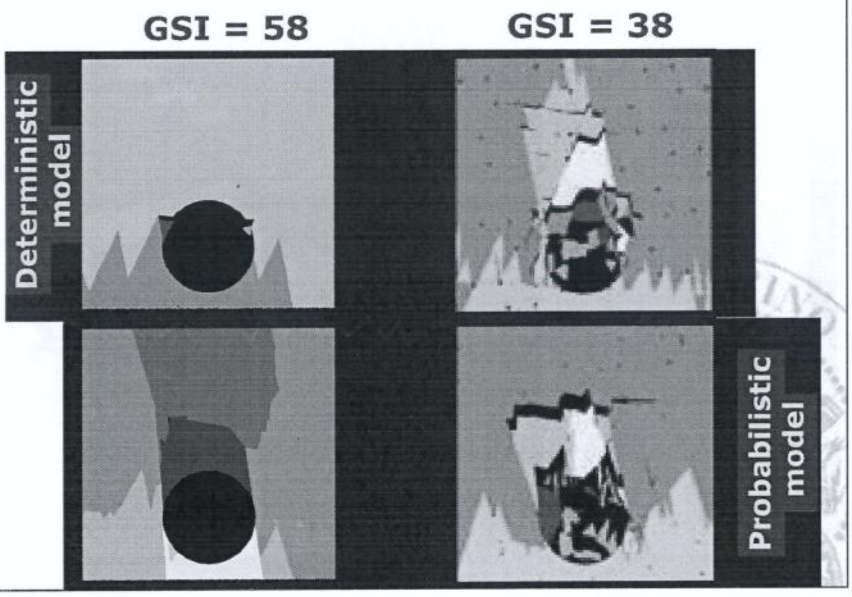
BUILDING A DEM MODEL



INTERPRETING RESULTS



INTERPRETING RESULTS



(II) Definition of boundary conditions

The model usually has a smaller size with respect to the one of FEM and boundary conditions are applied with the same technique of FEM.

(III) Definition of material properties

(IV) Definition of the initial stress state

(V) Definition of stages

(VI) Computation, that takes long time in DEM

(VII) Interpretation of results, especially in terms of displacement vectors.

These connections are able to resist

- → tension / compression
- → shear
- → torsion
- → bending

up to a certain breaking limit.
So, bonds have certain strain and resisting properties, usually described with an elastic perfectly plastic as, once the bond is broken, it is lost and particles interact as separate ones.
In this situation, cohesion is lost and interaction is given by micro-friction.

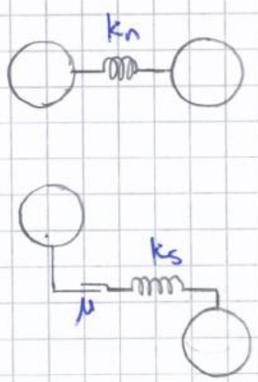
Thus, we are also able to simulate fractures on the rock.

Moreover, in case of more than 3 particles, we can adopt an infinitively rigid and resistent glue among them - i.e. it can't break - and the system is going to become a super-particle built by 3 parts connected by a rigid glue and with a different shape from the sphere.

This system is called clump, that is a super-particle of arbitrary shape with infinitively stiff and strong inter-granular contacts.
The system behaves as a rigid body and the deformability of the model comes from the connection with other clumps or particles.

This system, as it can change a lot its shape, can be used to simulate at a micro-scal the real particles distribution, for instance in sands, which has an influence on properties.

3 Particle contact

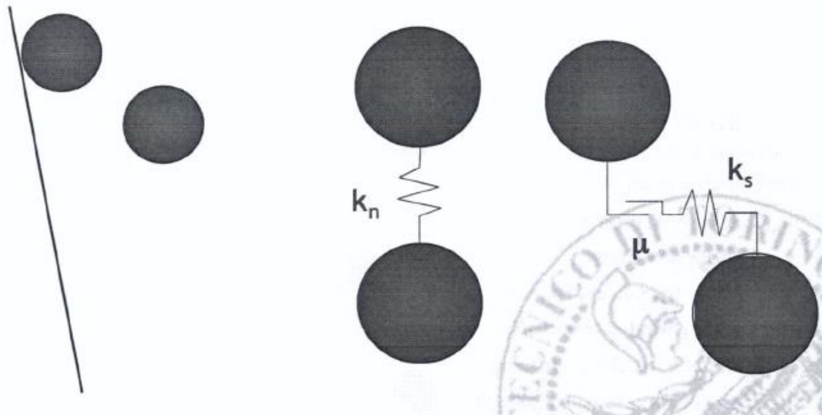


In the interaction between particles

- → normal stiffness is simulated by overlapping
- → shear stiffness and shear resistance are simulated by means of springs and a slider, which resistance is characterized by the parameter φ'.

$$\mu = \tan\varphi'$$

PARTICLE CONTACT



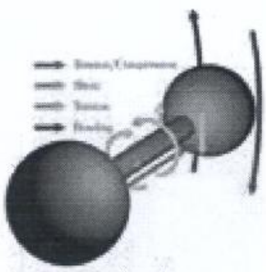
PARTICLE-WALL CONTACT

PARTICLE-PARTICLE CONTACT

PARTICLE BOND

Particle bonds are inter-particle connections able to resist

- tension/compression,
- shear,
- torsion and
- bending



up to a certain breakage limit. Once broken, the bond connection is lost permanently. Particle bonds and related models can be used to model soils, „glued particles", particle breakage, crack formation in beams („lattice beam model") etc...

CLUMPS

Each clump consists of a set of overlapping particles with inter-granular contacts infinitively stiff and strong that act as a rigid body with a deformable boundary

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 27 di 242

$$D_R = \frac{e_{max} - e}{e_{max} - e_{min}}$$

As the model of particle DEM is defined in a 2D domain, we have to define a 2D value of porosity, based on 2D voids:

by inverting the relationship of relative density, we can evaluate the 2D void index

$$e_{2D} = e_{2D,max} - D_R\left(e_{2D,max} - e_{2D,min}\right)$$

The values $e_{2D,min}$ and $e_{2D,max}$ are respectively the maximum and the minimum void ratio that we obtain with the maximum and the minimum patching of particles and they are 2D values.

From it, by applying the relationship porosity - void ratio, we get the porosity in 2-dimensional conditions.

$$n_{2D} = \frac{e_{2D}}{1 + e_{2D}} \qquad \text{2D - porosity}$$

(II) Micro parameters

There are 3 parameters that govern the interaction between 2 different particles

→ normal stiffness of the contact k_n

→ shear stiffness of the contact k_s

→ particle friction coefficient μ

If particles are glued and a bond is created, we have to add other parameters.

→ bond normal stiffness k^n

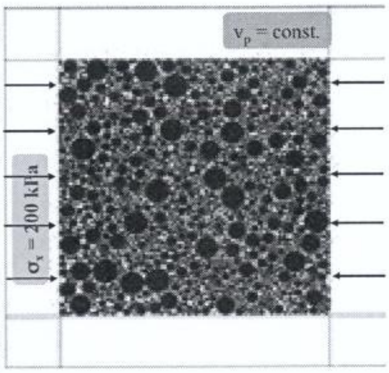
→ bond shear stiffness k^s

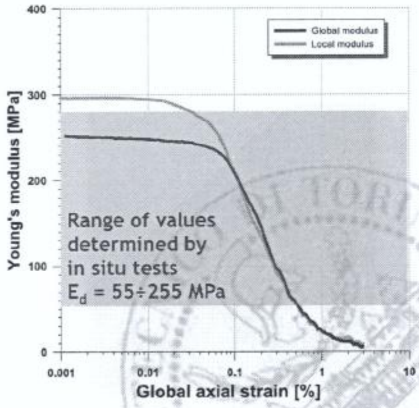
→ bond tensile strength σ_c

→ bond shear strength τ_c

→ bond radius λ

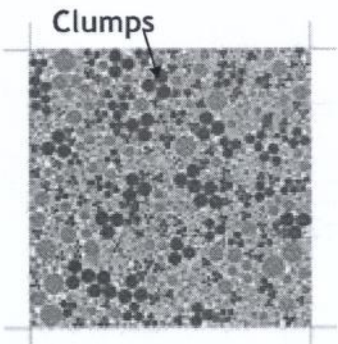
SIMULATING LAB TESTS



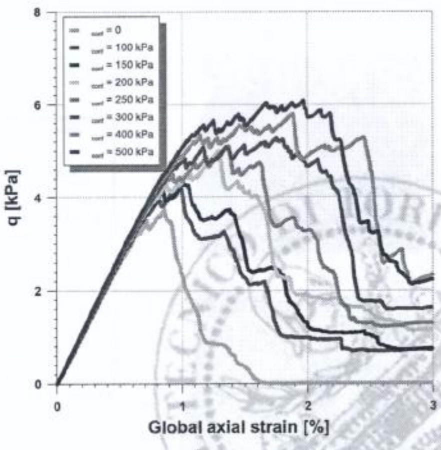


MICRO PARAMETERS

To increase strength one may introduce clumps



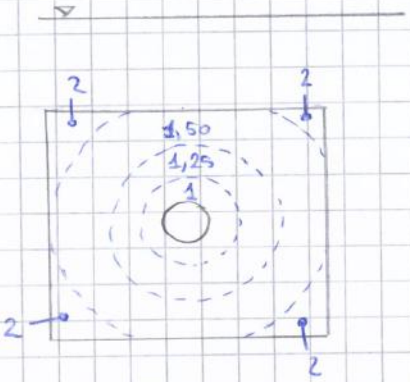
Synthetic specimen



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 31 di 242

5 Building the particle model

(I) Generation of particles and walls



For instance, in the case of excavation of a small diameter tunnel, the ground is reproduced by introducing grains and voids.
Then, boundary conditions are set not by placing hinges but by setting wall elements:

wall elements can't move or are movable so that normal stress is kept as constant – the velocity of displacement of the wall is controlled. This last approach is more realistic, as it reproduces confining stress.

The model is generally divided in circular areas in order to, going further from the excavation, multiply the size of particles and reduce the total number of particles.
Indeed, the proper grain size is used in a small area closed to the excavation – high detailed model – and, going further, all diameters of particles are multiplied by a constant and in the other areas the number of particles will be smaller, otherwise the model would be too heavy.

(II) Definition of micro parameters of walls and particles

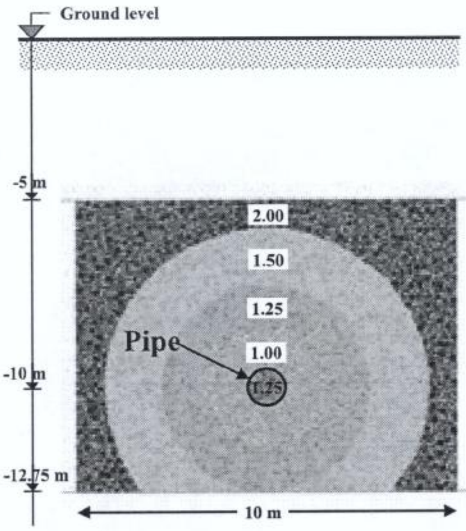
(III) Setting of the initial stress state

(IV) Definition of stages

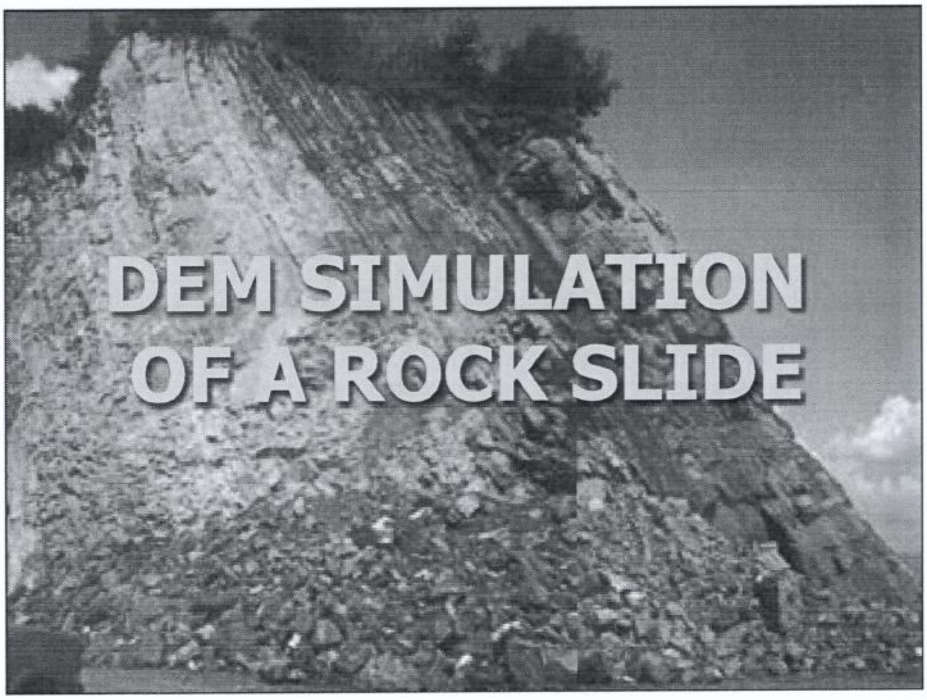
(V) Computation

(VI) Interpretation

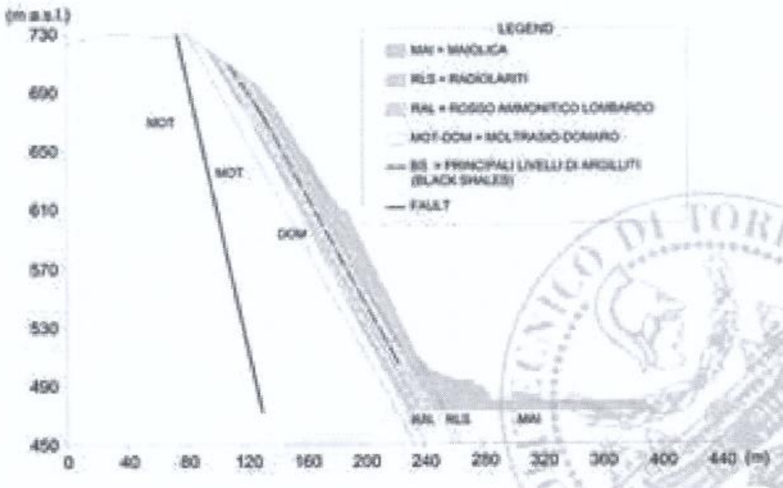
BUILDING A MODEL



1. Generate particles and walls + boundary conditions.
2. Define properties of particles and walls
3. Set the initial stress state
4. Define stages
5. Compute
6. Interpret

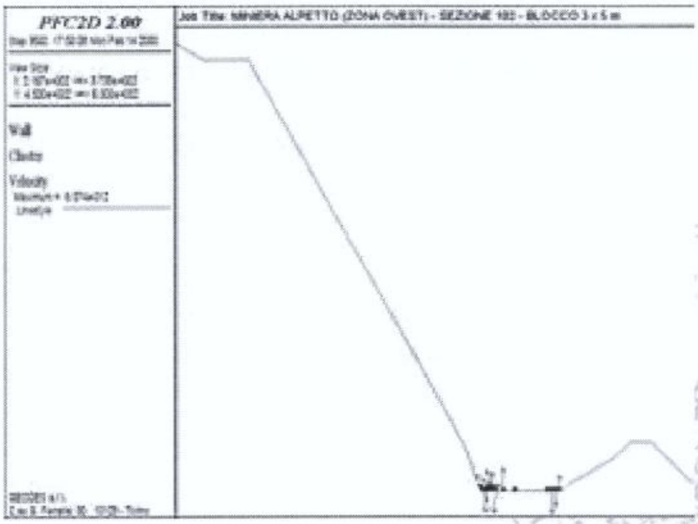


CROSS SECTION A

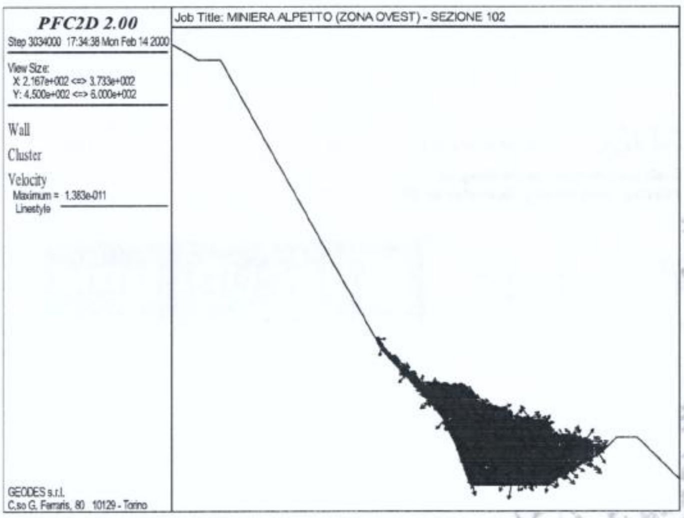


DEM MODEL

DEM MODEL SINGLE BLOCK



DEM MODEL



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 39 di 242

6 ADVANTAGES → possibility to simulate complete detachments and new contacts during computation

→ use for the evolution of a landslide and not only for triggering (="innesco") - it was not possible with FEM, as it works in continuous mechanics.

DISADVANTAGES → computer demanding

→ when particle element method is used, it is necessary to determine microparameters and a calibration process is needed

7 DEM softwares

→ UDEC and 3DEC, that work with blocks

→ PFC = Particle Flow Code

→ DEM Solutions (Freeware), which works with particles and it is applied in mining field

→ Mercuri DPM

→ Rocky

The FDM used an explicit approach to evaluate the solution and, by consequence, the solving equation will be the following.

$$kx + M\ddot{x} + C\dot{x} = p$$

In this approach, damping is important and it is expressed through a viscous damping k_s, proportional to stiffness.

$$k_s = 2h\sqrt{\frac{E}{\rho}} \qquad h = \text{element size}$$

The stability of the solution is guaranteed by the selection of an appropriate time step.

2 INGREDIENTS OF FDEM

- → deformability of the material, as in FEM
- → fracture model has to be implemented because it tells when a crack is created – transition from FEM to DEM
- → contact detection and contact interaction because, once a crack has been created, they have to be characterised, as in DEM.

Thus, structure of FDEM is characterised by FEM, DEM and their interaction.
It resembles the ELPLA model, where elastic behaviour and plastic behaviour "interact" through the yielding surface.

In this specific case, there is no plastic behaviour of elements but they get cracked and separated. Then, each element will have an elastic behaviour and, in some way, plastic behaviour is simulated by propagation of fractures.

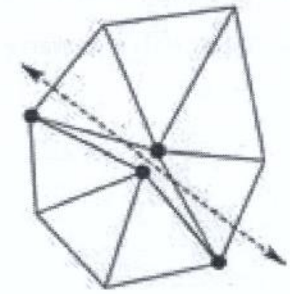
FRACTURE PROCESS

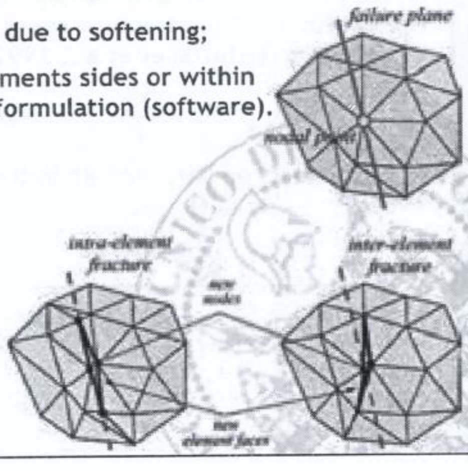
- In the context of the combined finite-discrete element method, transition from continua to discontinua is done through fracture and fragmentation process:
  - Mode I crack - opening mode: when the tensile strength normal to the fracture plane is exceeded;
  - Mode II crack - sliding mode the shear strenght the fracture plane exceeded.

FRACTURE PROCESS

Combined single and smeared crack model (Munjiza et al., 1999):

- Elastic behaviour up to the strength limit given by a Mohr-Coulomb criterion with a tension cut off;
- Gradual decrease of strength due to softening;
- Fracture is possible along elements sides or within elements, depending on the formulation (software).

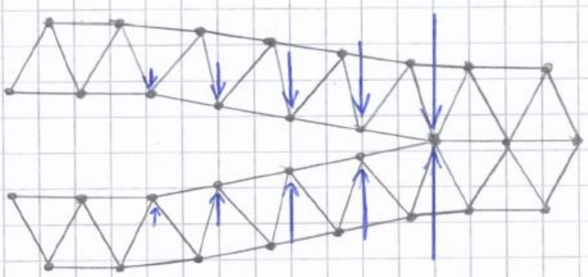


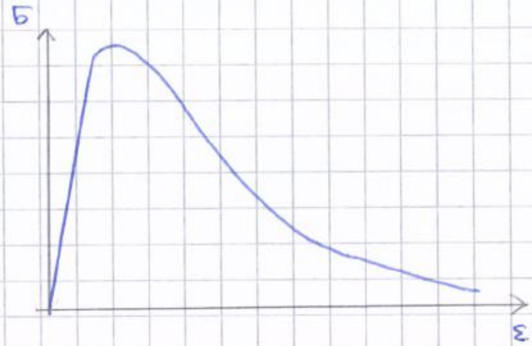


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 45 di 242

As regards MECHANICAL BEHAVIOUR, we can identify an elastic softening plastic behaviour:

there is an elastic behaviour up to the strenght limit, generally given by a Mohr-Coulomb criterion with a tension cut-off.
Then, fracture is created and there is a gradual decrease of resistance due to softening.





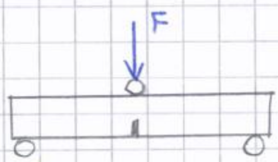
This model suits the model's behaviour because, at the beginning, elements are connected and mesh is continuous.
Then, a crack starts - e.g. opening crack - when the strength limit is reached (before, the behaviour is elastic).
Opening is not a sudden operation because more elements are gradually detached and stress state reduces due to softening, until the full opening.
So, FDEM uses an approach with smooth transition in fracture propagation.

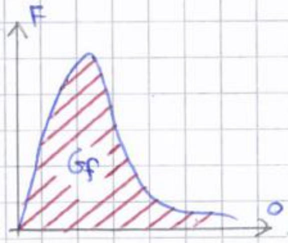
↳ FDEM parameters

We have to identify many parameters

→ properties of the intact material

We need the deformability parameters E and ν in order to identify the elastic part and tensile strength σ_t and shear strength c_i, φ_i (i = intact).





Moreover, we have to identify fracture energy (new parameter) G_f, that is the energy involved in the fracture process.
This parameter is determined thanks to the equation

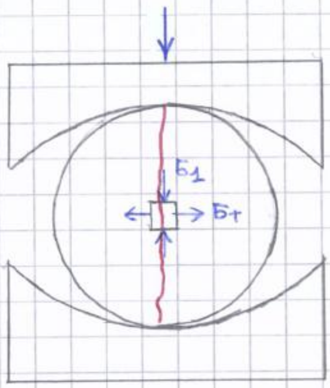
$$G_f = \frac{k_{Ic}^2}{E}$$

$k_{Ic} = 0,271 + 0,107\,\sigma_t$ Fracture toughness

otherwise, it is determined by SENB lab test.

6 Validation

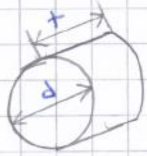
In building the model, we identify a problem with a closed-form solution or a laboratory test and we repeat the same conditions in the model.
The results will be compared, in order to validate the model.



For instance, on a brasilian test a load is applied along the diagonal of a cylinder with small height.
The specimen will break by opening because tensile strength is exceeded along the vertical direction in correspondence of the central element.
Here, at failure, the stress state is characterised

$$\sigma_1 \sim 3\sigma_t$$

where σ_t is given by



$$\sigma_t = \frac{2P}{\pi t d}$$

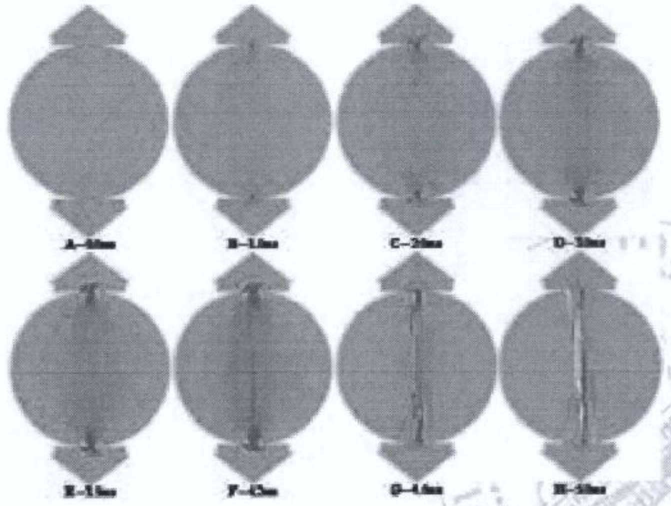
Here, the initial crack is created and then it propagates.

Reproducing it in a FDEM model, we can see the formation of the crack and its propagation.

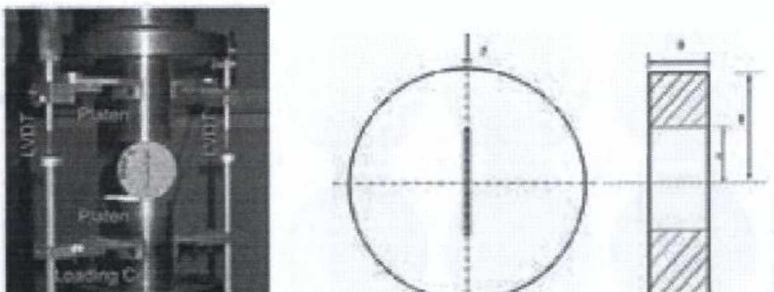
We can also simulate the behaviour of a brasilian disk with an initial crack.
In this case, the model presents a circle with a straight crack - placed with different inclinations - and a plate or a triangle, which is an unique finite element separated from the specimen and is very rigid.
So, model is made by 2 distinct elements.

In the result, we can notice that the specimen is going to break with a vertical line, so there is a deviation in propagation from the initial straight crack.

VALIDATION ON LAB TESTS

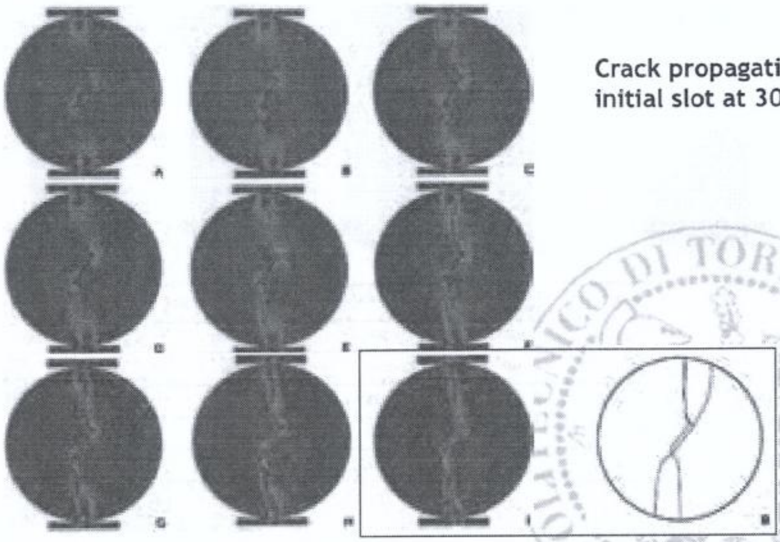


VALIDATION ON LAB TESTS



Cracked Straight Through Brazilian Disk (CSTBD)

VALIDATION ON LAB TESTS

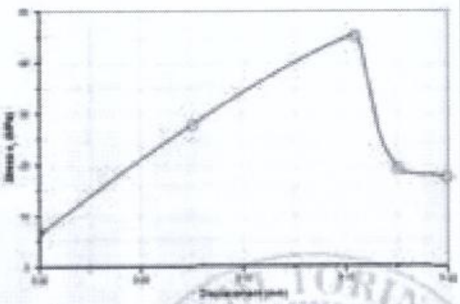


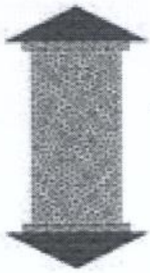
Crack propagation
initial slot at 30°

VALIDATION ON LAB TESTS

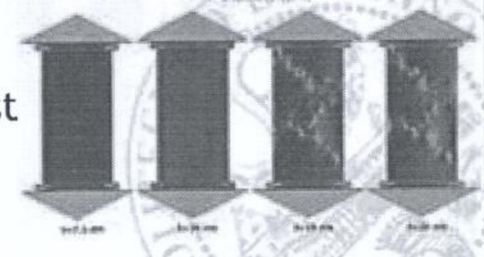
Young's modulus	E	50	GPa
Poisson's ratio	ν	0.25	-
Tensile strength	f_t	5	MPa
Fracture energy	G_f	0.05	N/mm
Cohesion	c	4.9	MPa
Friction angle	ϕ	30	°
Density	ρ	2340	kg/m^3

Material properties of the rock cylinder





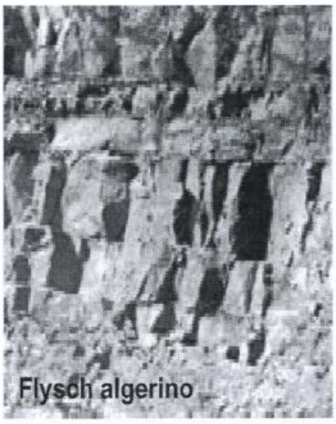
Triaxial test



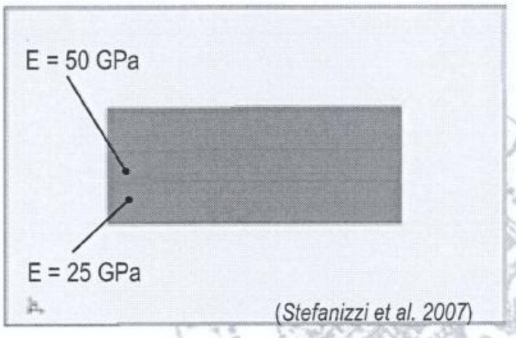
[Mahabadi, 2007]

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 53 di 242

ETEROGENEITIES (FDEM)



Fracture generated in a model which simulates a layered material where layers have different mechanical characteristics.



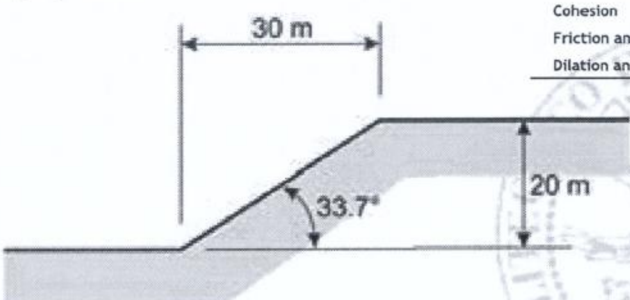
(*Stefanizzi et al. 2007*)

CIRCULAR FAILURE

Ground properties

Elastic properties	
Unit weight	19 kN/m^3
Young's modulus	50 MPa
Poisson's ratio	0.4
Strength parameters	
Tensile strength	28 kPa
Cohesion	28 kPa
Friction angle	30°
Dilation angle	0

Definition of the theoretical slope problem



[Piovano et al., 2011]

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 55 di 242

In the model, initial stress state was computed in different ways

→ from final geometry, in which gravity applies

→ erosion by removing 4 layers 5 m thick

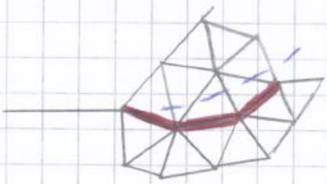
→ erosion by removing 10 layers 2 m thick

As FDEM involves an explicit solution, we may compare these 3 approaches by seeing evolution of total kinetic energy in time. Total kinetic energy goes up and down until an equilibrium in each time step and the III approach requires more time to reach complete equilibrium, whereas the I approach is the quickest.
As results are not so much different, we will adopt the I approach.

II) FRACTURE PROPAGATION AND SAFETY FACTOR

In this stage, 4 meshes were adopted in order to take into account mesh sensitivity and the safety factor is computed by using the SSR method.

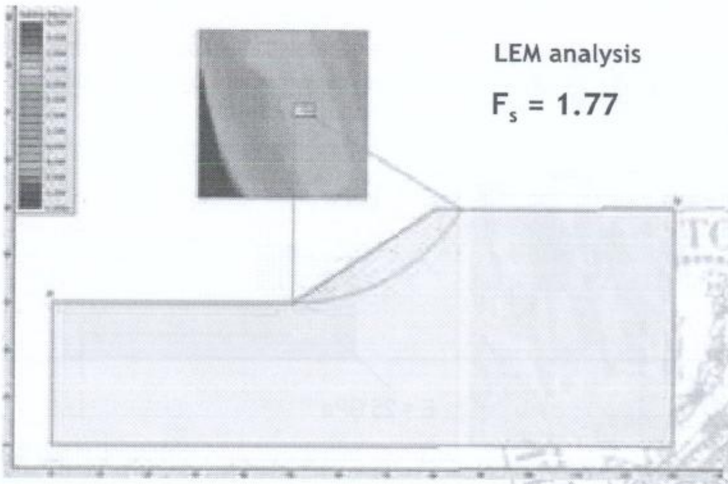
Comparing results with the ones from LEM, we can notice that FDEM is a very MESH-DEPENDENT METHOD and, if slope is discretised with a certain mesh, we will get a wrong result.



The reason is that, generally, FDEM is able to create cracks only along the elements' side and this aspect may force cracks to follow a line different from the natural one, changing results dramatically.

So, we have to build the mesh in a proper way.

CIRCULAR FAILURE



LEM analysis

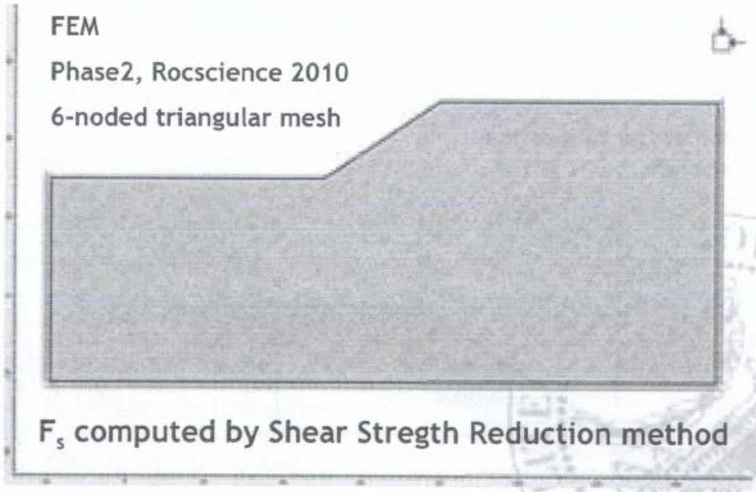
$F_s = 1.77$

CIRCULAR FAILURE

FEM

Phase2, Rocscience 2010

6-noded triangular mesh



F_s computed by Shear Stregth Reduction method

CIRCULAR FAILURE

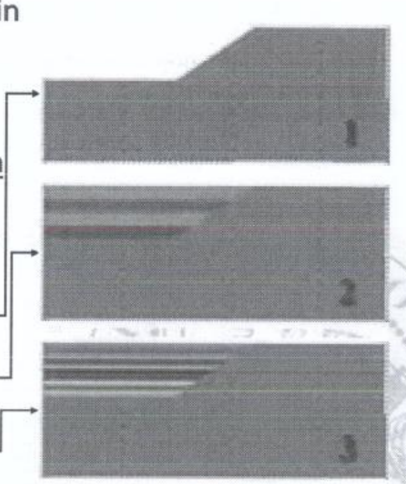
FDEM simulation is performed in two stages:

stage 1 - initial stress condition
stage 2 - fracture propagation and F_S

stage 1 - initial stress condition

To initialize the state of stress in the slope prior to triggering instability, three different methods were adopted:

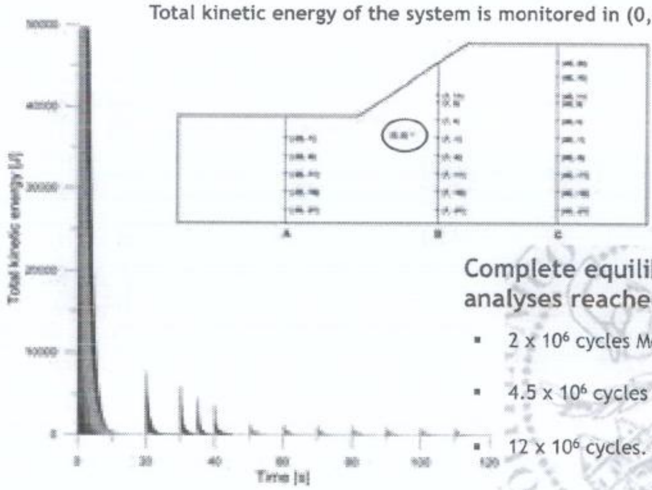
1. No excavation, simply gravity-turn on.
2. Erosion by removing 4 layers of 5 m thickness.
3. Erosion by removing 10 layers of 2 m thickness.



CIRCULAR FAILURE

stage 1 - initial stress condition

Total kinetic energy of the system is monitored in (0,0)



Complete equilibrium in all analyses reached after:

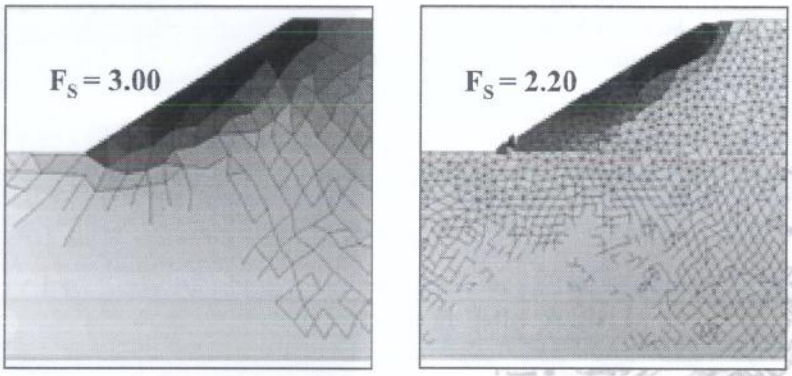
- 2 x 10^6 cycles Method 1
- 4.5 x 10^6 cycles Method 2 **(2x)**
- 12 x 10^6 cycles. Method 3 **(6x)**

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 61 di 242

CIRCULAR FAILURE

stage 2 - fracture propagation and F_s

h = 4 m   h = 2 m

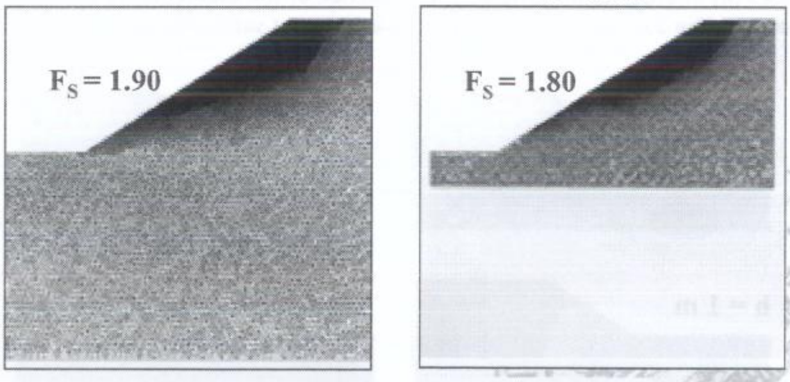


Shear Stregth Reduction method

CIRCULAR FAILURE

stage 2 - fracture propagation and F_s

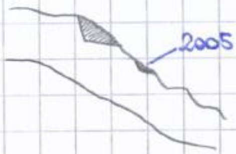
h = 1 m   h = 0.5 m



Shear Stregth Reduction method

Ex (TORGIOVANNETTO DI ASSISI)

In the locality of Torgiovannetto di Assisi, there is an abandoned open pit mine which is interested by a main wedge and an instability of a smaller wedge has been occurred in 2005



The cross section presents a wedge with a 1 m opened crash and a certain sliding surface. Thus, geometry is defined.

The phenomenon is studied with FDEM because, since it is able to simulate fracture propagation, it is useful to study not only triggering, but also the evolution of the scenario.

Firstly, we need to do GEOTECNICAL CHARACTERISATION of intact rock - compression test and brasilian test - and of discontinuities.
The combined method needs also the fracture energy, determined by using the SENB test, that simulates propagation of fractures due to opening or sliding mode.

The results of characterisation are used to validate the model.

For instance, in the compression test, a distributed load is applied by using steel plates.
The finite element model, in sake of simplicity, present at edges only one finite element which is infinitely stiff, in order to simulate loading

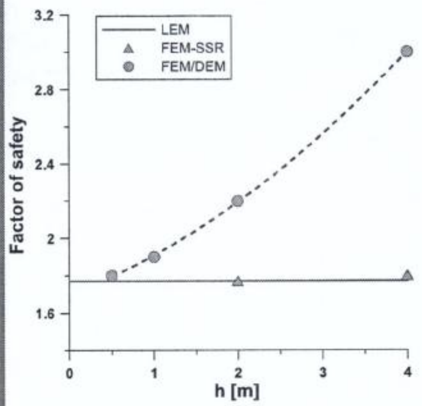
In this model, when load is increased, cracks arise and we pass to a discontinuum model.
At the end, stresses go to zero and elements get separated and slide.
↳ all energy goes into movement

The validation in situ scale is based on back analysis.

CIRCULAR FAILURE

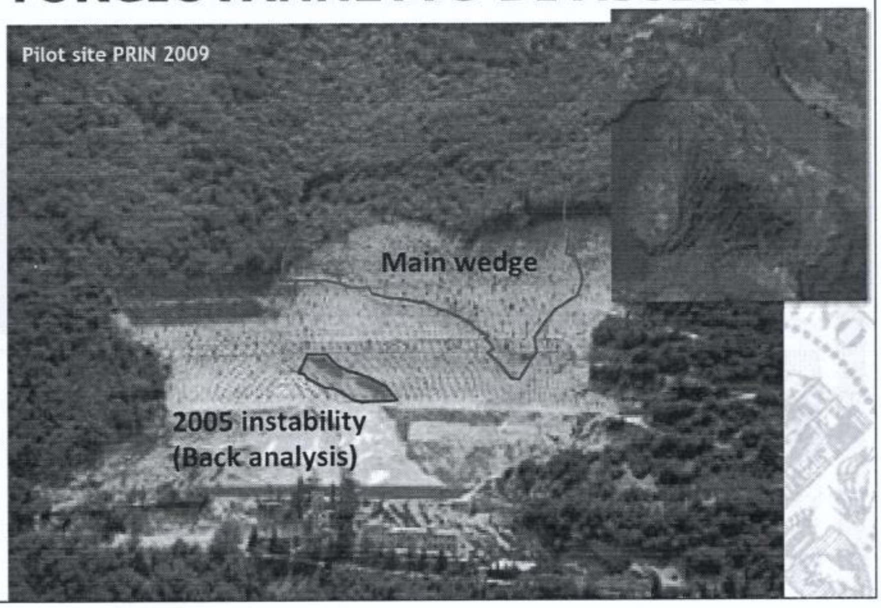
Comparison among LEM, FEM and FDEM

The numerical simulations are strongly mesh dependent.



Factor of safety			
LEM	FEM/SSR	FEM/DEM	
1.77	1.77	*4 m element size*	3.00
		2 m element size	2.20
		1 m element size	1.90
		0.5 m element size	1.80

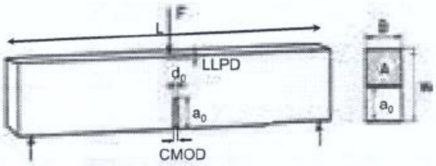
TORGIOVANNETTO DI ASSISI

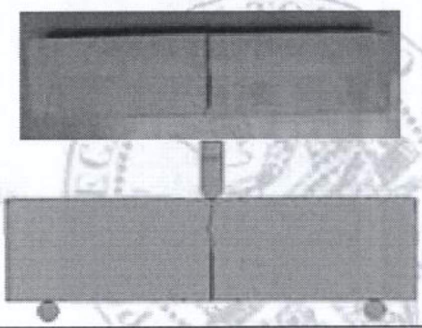


Laboratory scale

Three point bending test on Single End Notched Beam (SENB)

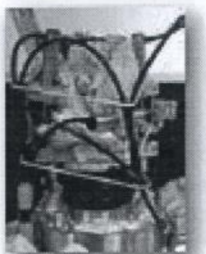
Determination of fracture energy (G_f)



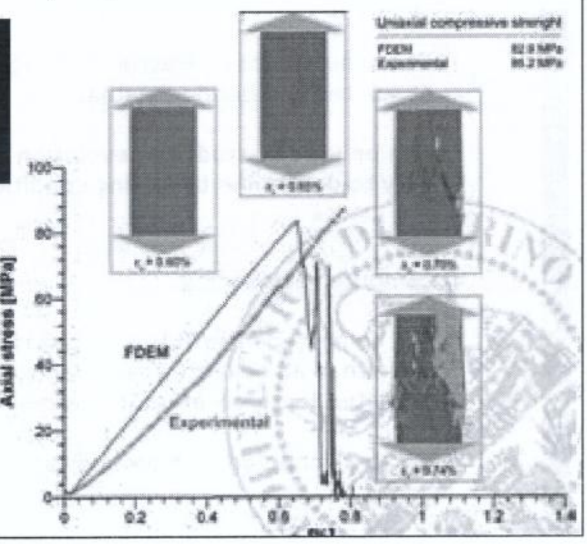


Laboratory scale

Uniaxial compression tests (UCS)



Experimental
vs FDEM



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 69 di 242

WATER FLOW ANALYSIS

From the general point of view, the governing equations for water flow are the ones used for TRANSPORTATION PROCESSES, that involve water flow, electrical flow or thermal flow.
All of these are different physical processes but, from the mathematical point of view, they are described by the same equations

→ Laplace equation, valid for STEADY STATE CONDITIONS.

$$\frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} + \frac{\partial^2 \Phi}{\partial z^2} = 0$$

Φ = generic potential function (voltage in case of electrical flow, temperature in case of thermal flow, hydraulic head in case of ground water flow)

From this equation, we are able to get the solution of the general transportation problem and we can substitute the potential related to the process in exam.

→ Poisson equation

$$\frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} + \frac{\partial^2 \Phi}{\partial z^2} = f(x; y; z; t)$$

Due to time, the equation is related to TRANSIENT PROCESSES and, again, it is valid for each type of physical process of transportation.

This expression gives the neat mass that, along x direction, is in the element.

The neat mass in the other directions is given by the same expression and we combine the contributions.

$$\left[\rho_w v_x - \left(\rho_w v_x + \frac{\partial(\rho_w v_x)}{\partial x}dx\right)\right] dy\,dz\,dt +$$
$$+\left[\rho_w v_y - \left(\rho_w v_y + \frac{\partial(\rho_w v_y)}{\partial y}dy\right)\right] dx\,dz\,dt +$$
$$+\left[\rho_w v_z - \left(\rho_w v_z + \frac{\partial(\rho_w v_z)}{\partial z}dz\right)\right] dx\,dy\,dt$$

This is the mass coming in the element from the 6 faces, i.e. the variation in the water mass due to the water flow.
This corresponds to an "internal" change of the water mass

$$\frac{\partial(\rho_w n S_r)}{\partial t}\,dx\,dy\,dz\,dt$$

n = porosity, i.e. volume of voids with respect to total volume

S_r = degree of saturation, as the volume of soil is occupied by solid grains, water and air.
The degree of saturation gives the percentage of voids filled with water, so the actual quantity of water is $n s_r$ multiplied by the volume. Multiplying it by the density ρ_w, we get the water mass.

The balance equation will be the following one.

$$\left[\rho_w v_x - \left(\rho_w v_x + \frac{\partial(\rho_w v_x)}{\partial x}dx\right)\right] dy\,dz\,dt +$$
$$+\left[\rho_w v_y - \left(\rho_w v_y + \frac{\partial(\rho_w v_y)}{\partial y}dy\right)\right] dx\,dz\,dt +$$
$$+\left[\rho_w v_z - \left(\rho_w v_z + \frac{\partial(\rho_w v_z)}{\partial z}dz\right)\right] dx\,dy\,dt = \frac{\partial(\rho_w n S_r)}{\partial t}\,dx\,dy\,dz\,dt$$

$$\Rightarrow -\left(\frac{\partial(\rho_w v_x)}{\partial x} + \frac{\partial(\rho_w v_y)}{\partial y} + \frac{\partial(\rho_w v_z)}{\partial z}\right) = \frac{\partial(\rho_w n S_r)}{\partial t}$$

$$\frac{\partial^2 h}{\partial x^2} + \frac{\partial^2 h}{\partial y^2} + \frac{\partial^2 h}{\partial z^2} = \frac{\partial n}{\partial t}$$ Poisson equation for the water flow

3 Steady state conditions

In steady state conditions, nothing is depending on time and, especially, porosity is not changing.

$$\frac{\partial n}{\partial t} = 0$$

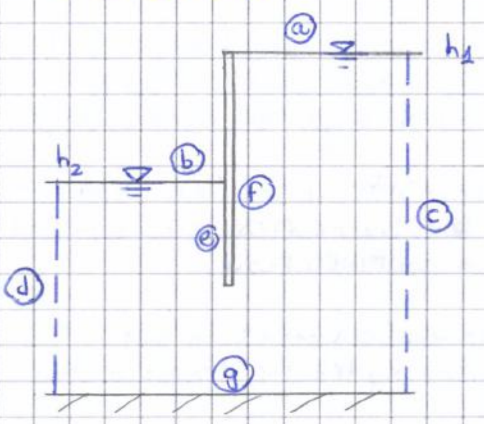
By consequence, the volume does not change in time and also effective stress don't change during the hydraulic process.
In other words, mechanical problem is independent from the hydraulic problem and we call it uncoupled problem.
In this situation, mechanical problem (stresses) and hydraulic problem (pore pressure) can be solved separately and the solutions can be put together.

In steady state condition, Poisson equation is simplified.

$$\frac{\partial^2 h}{\partial x^2} + \frac{\partial^2 h}{\partial y^2} + \frac{\partial^2 h}{\partial z^2} = 0$$ Laplace equation

STEADY STATE GEOTECHNICAL PROBLEMS

→ sheet wall



A typical problem is the ones of a sheet wall built in a ground with a certain fault level and on an impermeable rock.

The boundary conditions imposed to this problem are the ones used for water flow problems

→ sides (a) and (c), in which total head is known

$$h = h_1$$

→ sides (b) and (d)

$$h = h_2$$

→ in the wall and at the impermeable rock, we don't know the hydraulic head but there is no water flow across these elements. By consequence, given the normal direction n to sides (e), (f) and (g),

→ within the embankment, there is a phreatic line and its position is not known.

From the analytical point of view, the phreatic line position is an unknown that has to be solved.

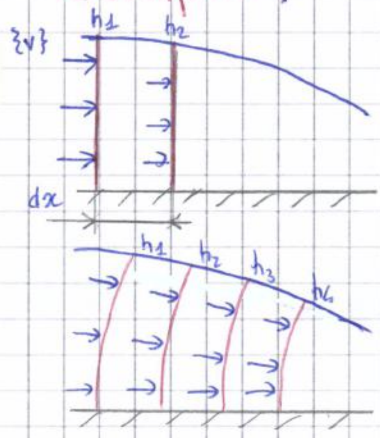
By definition, the phreatic line is a line in which water pressure is null

$$u = 0$$

and hydraulic head is equal to the geometric height

$$h = z$$

If we want to find analytically the phreatic line, we rely on Dupuit's assumption.



Given the flow on an impermeable base, e.g. rock, the flow is horizontal, according to Dupuit. By consequence, the equipotential lines - lines where the hydraulic head is constant - are vertical.

Actually, the equipotential lines are curved and flow has a certain direction.

Dupuit's assumption is a simplification of reality but, under this, we can write that discharge Q ("portata") is given by the water in the datched area.

$$Q = qA = vA$$

q = specific discharge, equal to the seepage velocity

A = area of the section

Seepage velocity is given by Darcy's law, taking into account that the loss of hydraulic head occurs only in x direction.

$$\{v\} = -k \nabla h \rightarrow v = -k \frac{dh}{dx}$$

The section is a plane section, given by 1 m x hydraulic head.

$$A = 1 \cdot h = h$$

4 Transient conditions

This is a more general condition, in which variables change in time.

To describe this, we can start from the mass balance equation

$$k\frac{\partial^2 h}{\partial x^2} + k\frac{\partial^2 h}{\partial y^2} + k\frac{\partial^2 h}{\partial z^2} = \frac{\partial n}{\partial t}$$

We can rewrite it in terms of the pore pressure, by applying the definition of hydraulic head.

$$h = z + \frac{u}{\gamma_w}$$

$$\Rightarrow \quad k\left[\frac{\partial^2}{\partial x^2}\left(z+\frac{u}{\gamma_w}\right) + \frac{\partial^2}{\partial y^2}\left(z+\frac{u}{\gamma_w}\right) + \frac{\partial^2}{\partial z^2}\left(z+\frac{u}{\gamma_w}\right)\right] = \frac{\partial n}{\partial t}$$

$$\frac{k}{\gamma_w}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = \frac{\partial n}{\partial t}$$

Assuming that the solid phase is uncompressible, changes in time of porosity are only related to the volume strains.

$$\frac{\partial n}{\partial t} = -\frac{\partial \varepsilon_p}{\partial t}$$

$$\Rightarrow \quad \frac{k}{\gamma_w}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = -\frac{\partial \varepsilon_p}{\partial t}$$

$$\frac{k}{\gamma_w}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = -\frac{\partial \varepsilon_p}{\partial t}$$

For sake of simplicity, we can introduce volume strain as elastic.

$$\delta\varepsilon_p = \frac{\delta p'}{K} = \qquad K = \frac{E}{3(1-2\nu)}$$

$$= \frac{3(1-2\nu)}{E}\cdot\left(\delta p - \delta u\right)$$

The equation will become

$$\frac{k}{\gamma_w}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) = \frac{3(1-2\nu)}{E}\left(\frac{\partial u}{\partial t} - \frac{\partial p}{\partial t}\right)$$

In this situation, there is NO CHANGE IN PORE PRESSURE

$$\Delta u = 0$$

By consequence, changes in effective and total stress are the same and we can neglect the mass balance equation and solve only the mechanical part.

→ Fully undrained problems

This corresponds to the condition where the load is suddenly applied and overpressures are generated.
As overpressures are difficult to be evaluated, fully undrained problems don't calculate them and do a trick, by running an ANALYSIS IN TERMS OF TOTAL STRESSES.
Due to this, we can't use the actual stiffness matrix of the material because we are not describing the real behaviour of the soil - it depends on effective stresses - and we have to define a new constitutive law, in terms of total stresses (basically, a trick).

Generally, we need to solve the coupled problem and the analysis is run in two different ways.

→ Fully coupled scheme (very computation demanding):

all 16 equations are solved in a single block at each time-step.

→ Iterative coupled scheme

For each time-step, the problem is solved by assuming that

$$\delta \varepsilon_p = - C_p \delta u$$

C_p = approximate compressibility of the solid skeleton ⇝ initial assumption

Thus, we are neglecting the contribution due to the variation in p', as we start from the equation

$$\delta \varepsilon_p = \frac{\delta p - \delta u}{K}$$

In this way, we can solve the mass balance equation, that becomes

$$-\frac{\partial \varepsilon_p}{\partial t} = C_p \frac{\partial u}{\partial t} = \frac{k}{\gamma_w}\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$

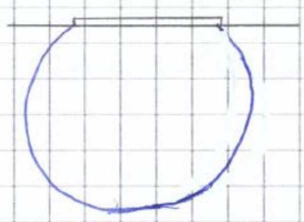
relevant for the problem.

ANALYSIS OF LONG TERM SETTLEMENTS (elastic solution)

We evaluate settlements at point A (center of the foundation) and point B (edge of the foundation).

Firstly, we have to determine the Young modulus E.

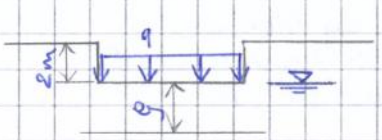
As behaviour is not linear, we have to define a fitting parameter that holds for a reasonable ample influence surface.



So, we define the area of the iso-incremental vertical stress, that corresponds to the area of interest.
As a first attempt, the depth of this area from the bottom of the foundation ~~is~~ may be evaluated as

$$g = \frac{2B}{\pi} = 1,27 \text{ m}$$

In this depth, we evaluate the average Young modulus.



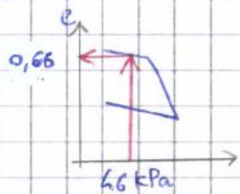
At the depth g, we evaluate the effective stress

$$\sigma'_{v0} = (2 + 1,27) \cdot 18 - 1,27 \cdot 10 = 46 \text{ kPa}$$

From the oedometric test, we get the oedometer modulus.

$$E_{oed} = \frac{(1+e)\sigma'_{v0}}{k} =$$

$$= \frac{(1+0,66) \cdot 46}{0,012} = 6350 \text{ kPa}$$



Assuming $\nu = 0,2$, the Young modulus will be

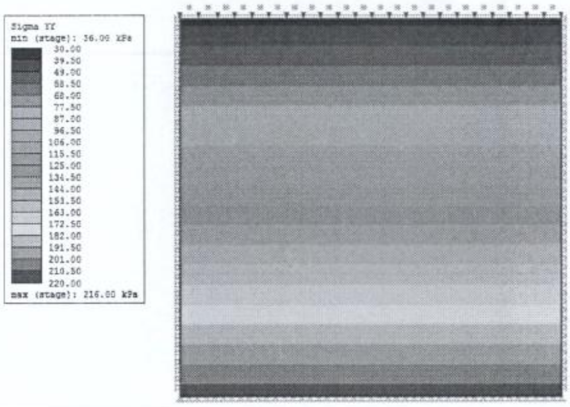
$$E = \frac{(1-2\nu)(1+\nu)}{(1-\nu)} E_{oed} =$$

$$= \frac{(1-2\cdot 0,2)(1+0,2)}{(1-0,2)} \cdot 6350 = 5715 \text{ kPa}$$

Example problem

Infinite strip loading on a fine graded soil:
long term settlements (elastic)

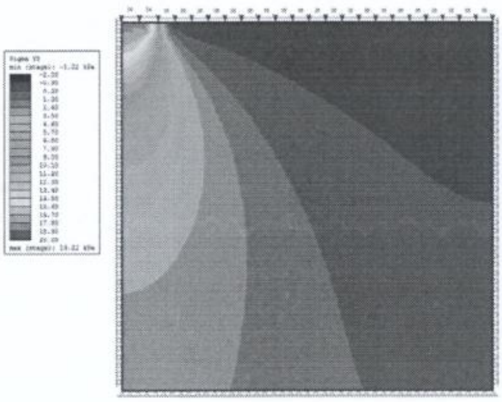
- Results of simulation: initial condition (geostatic) – vertical stress



Example problem

Infinite strip loading on a fine graded soil:
long term settlements (elastic)

- Results of simulation: increase of vertical stress for q= 54 kPa

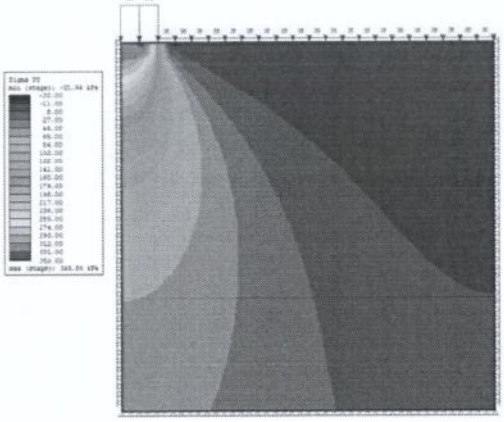


Only the increase in vertical stress with respect to the initial condition is given

Example problem

Infinite strip loading on a fine graded soil:
long term settlements (elastic)

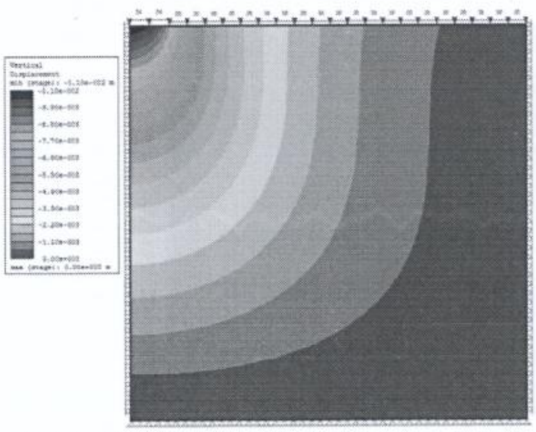
- Results of simulation: increase of vertical stress for q= 360 kPa



Example problem

Infinite strip loading on a fine graded soil:
long term settlements (elastic)

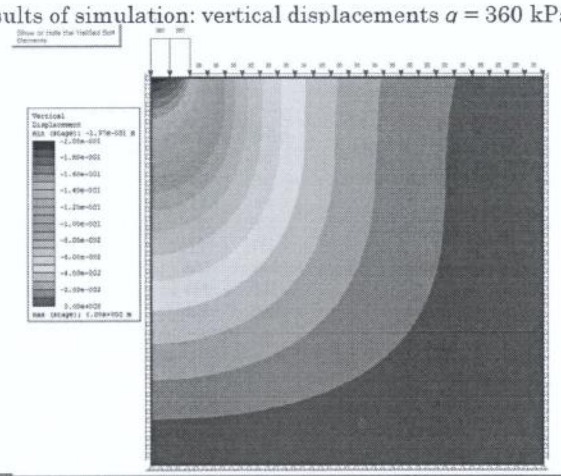
- Results of simulation: vertical displacements q = 54 kPa



Example problem

Infinite strip loading on a fine graded soil:
long term settlements (elastic)

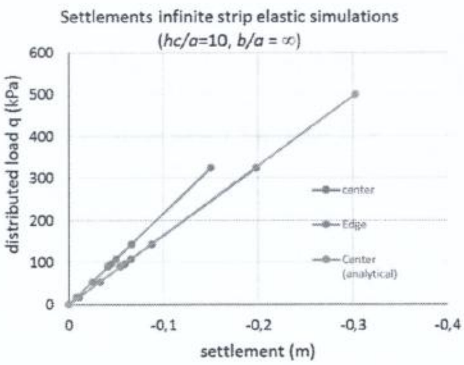
- Results of simulation: vertical displacements q = 360 kPa



Example problem

Infinite strip loading on a fine graded soil:
long term settlements (elastic)

- Results of simulation: load – vertical displacements curve

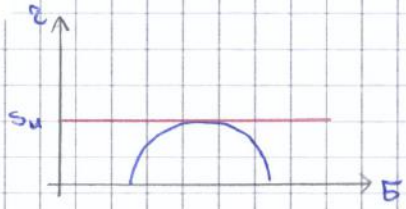


Very small difference between the analytical solution and the simulations –

Numerical simulations, infinite strip:
$I_F = 1{,}781$

Analytical, b/a= 10:
$I_F = 1{,}77$

UNDRAINED ANALYSIS OF BEARING CAPACITY



In the evaluation of bearing capacity, we adopt the elasto perfectly plastic Tresca criterion, characterised by the undrained shear strength s_u, as analysis is run in undrained conditions.

We also need to define undrained elastic parameters E_u and ν_u.

→ UNDRAINED ELASTIC PARAMETERS

In undrained conditions, we can assume that there is no volume change.

$$\varepsilon_p = 0$$

As we are using an elastic model, we can say that, for each increase of total mean stress

$$\varepsilon_p = \frac{\Delta p}{k_u} = 0$$

The undrained bulk modulus k_u follows the same definition of the drained one.

$$k_u = \frac{E_u}{3(1-2\nu_u)}$$

In order to have nil volume strain for each increase of total mean stress, we need to have

$$k_u \to \infty$$

So, we have

$$\nu_u = 0,5$$

$\nu_u = 0,5$ Undrained Poisson's coefficient

This is only a fitting parameter, not a real one.
Moreover, in the model we can't set this coefficient as equal to 0,5, otherwise we would have numerical problems. So, we set a similar value, e.g. 0,49999.

To determine the undrained Young's modulus, we can notice that shear modulus doesn't change because water is not able to carry tangential stresses.

$$G = G_u$$

In undrained conditions, the undrained shear strength is directly related to the deviatoric stress.

$$s_u = \frac{q}{2} = \frac{M p_0'}{2} e^{\frac{\Gamma - N}{\lambda}}$$

In MCCM, we can write this as

$$s_u = \frac{M}{2^{\Lambda+1}} p_0' \quad , \quad \Lambda = \frac{\lambda - k}{\lambda}$$

If soil is OC, we have to add the factor OCR^{Λ}.

$$s_u = \frac{M}{2^{\Lambda+1}} p_0' \, OCR^{\Lambda}$$

$$s_u = \frac{M}{2^{\Lambda+1}} p_0' \, OCR^{\Lambda}$$ Undrained shear strength

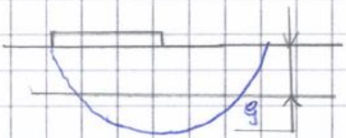
There is also the practical expression given by Koutsofas and Ladd

$$s_u = 0,22 \, \sigma_{v0}' \, OCR^{0,8}$$

These expressions are very similar, as

$$\frac{M}{2^{\Lambda+1}} \sim 0,22 \quad \text{and} \quad p_0' \sim \sigma_{v0}'$$

The parameters are evaluated along the failure surface.



Assuming that failure occurs along a cilinder, we should evaluate the shear strength along this surface. In order to have an average value, we refer to the depth defined by g

$\sigma_{v0}' = 46 \text{ kPa} \Rightarrow s_u = 50 \text{ kPa}$

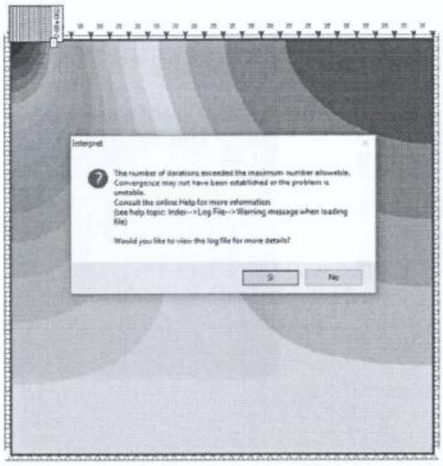
In numerical analysis, as we are in a plasticity problem, we can try to increase the load applied by the foundation.
At a certain point, there is no convergence anymore, i.e. at failure there are so many yielded points that model can't work anymore

$\Rightarrow$ we assume that NO CONVERGENCE CONDITION IS EQUAL TO FAILURE of the foundation.

Example problem

Infinite strip loading on a fine graded soil

- Undrained analysis of bearing capacity



The load was increased in steps until no convergence was found.

It is assumed that the no convergence condition is equal to failure of the foundation

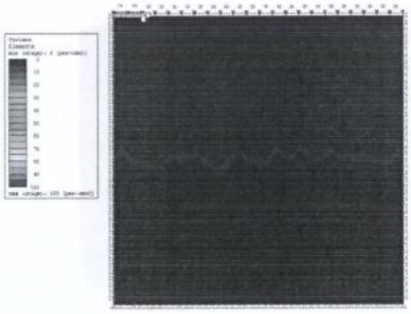
Example problem

Infinite strip loading on a fine graded soil

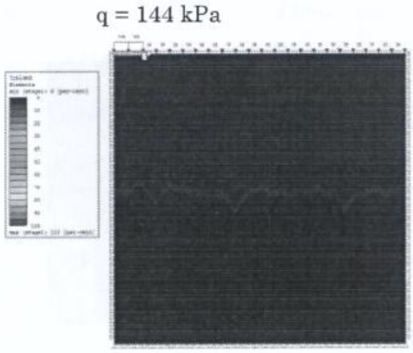
- Undrained analysis of bearing capacity: numerical results

Yielded elements

q = 54 kPa



q = 144 kPa

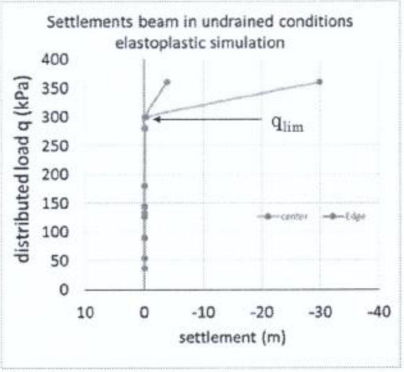


Example problem

Infinite strip loading on a fine graded soil

- Undrained analysis of bearing capacity: numerical results

Load – settlement curve



- Reference solution

The analytical reference solution for this problem is Brinch Hansen bearing capacity expression:

$$q_{lim} = (\pi+2)\ s_u + \sigma_{v0} = 293\ \text{kPa}$$

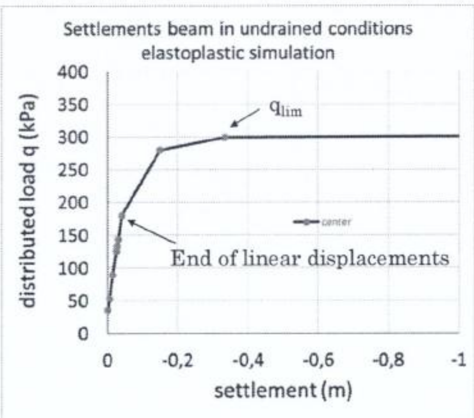
In the simulation the vertical displacements 'explode' when q > 300 kPa

Example problem

Infinite strip loading on a fine graded soil

- Undrained analysis of bearing capacity: numerical results

Load – settlement curve



Notwisthanding being an elasto perfectly plastic model the predicted settlements are non linear

(linear until q = 180 kPa)

Settlement for q = 144 kPa
s = 0,03 m
(s= 0,07 for the 'fully drained' case)