

CE394M: Finite Element Analysis in Geotechnical Engineering

Krishna Kumar

University of Texas at Austin

krishnak@utexas.edu

March 4, 2020

Overview

- 1 Geotechnical FEA
 - Element types
 - Discretization
 - Boundary conditions
 - Errors in FEA

IMPORTANT WARNING AND DISCLAIMER

PLAXIS is a finite element program for geotechnical applications in which soil models are used to simulate the soil behaviour. The PLAXIS code and its soil models have been developed with great care. Although a lot of testing and validation have been performed, it cannot be guaranteed that the PLAXIS code is free of errors.

Moreover, the simulation of geotechnical problems by means of the finite element method implicitly involves some inevitable numerical and modelling errors. The accuracy at which reality is approximated depends highly on the expertise of the user regarding the modelling of the problem, the understanding of the soil models and their limitations, the selection of model parameters, and the ability to judge the reliability of the computational results. Hence, PLAXIS may only be used by professionals that possess the aforementioned expertise.

The user must be aware of his/her responsibility when he/she uses the computational results for geotechnical design purposes. The PLAXIS organization cannot be held responsible or liable for design errors that are based on the output of PLAXIS calculations.

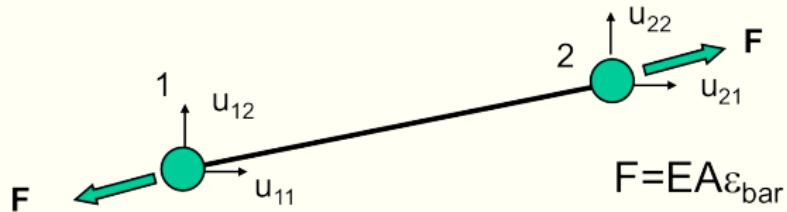
Consistent system of units

SI				
Length	m	m	m	cm
Density	kg / m^3	$10^3 \text{ kg} / \text{m}^3$	$10^6 \text{ kg} / \text{m}^3$	$10^6 \text{ g} / \text{cm}^3$
Force	N	kN	MN	Mdynes
Stress	Pa	kPa	MPa	bar
Gravity	m/sec^2	m/sec^2	m/sec^2	cm/s^2
Stiffness*	Pa/m	kPa/m	MPa/m	bar/cm

Problem definition

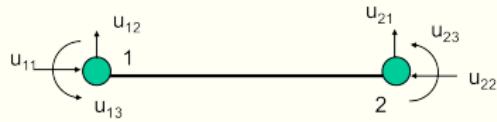
1D Finite Elements: Bar element

Two node element with axial stiffness only (no flexural or shear resistance).



1D Finite Elements: Beam element

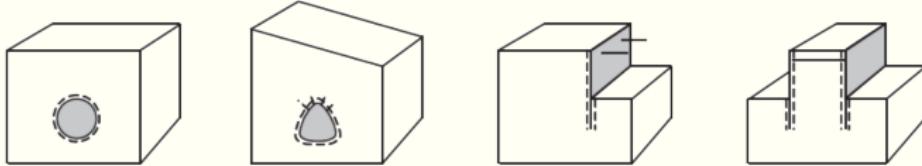
two node structure element with axial and bending stiffness (no transverse shear deformation). Three degrees of freedom for 2D beam element (1, 2 displacements and a moment).



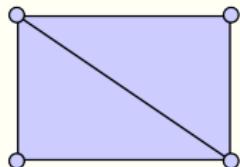
$$F_a = EA \epsilon_a$$

$$V = -EI(u_{12}-u_{21})/L^3 - 6EI(u_{13}+u_{23})/L^2$$

$$M = EI(u_{13}-u_{23})/L$$



2D plane-strain / axisymmetric elements



3 nodes element

linear variation of displacement
within the element = constant
strain in the element

$$d_1 = \alpha_1 + \alpha_2x + \alpha_3y$$

$$d_2 = \beta_1 + \beta_2x + \beta_3y$$

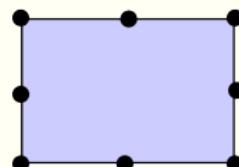


4 nodes element

linear variation of
displacement in both x and y
directions

$$d_1 = \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\xi\eta$$

$$d_2 = \beta_1 + \beta_2\xi + \beta_3\eta + \beta_4\xi\eta$$



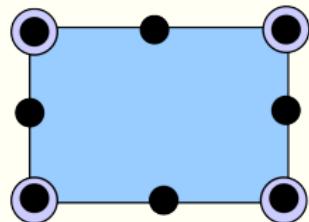
8 nodes element

quadratic variation of displacement in
both x and y directions.

$$\begin{aligned} d_1 &= \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\xi^2 \\ &\quad + \alpha_5\xi\eta + \alpha_6\eta^2 + \alpha_7\xi^2\eta + \alpha_8\xi\eta^2 \\ d_2 &= \beta_1 + \beta_2\xi + \beta_3\eta + \beta_4\xi^2 \\ &\quad + \beta_5\xi\eta + \beta_6\eta^2 + \beta_7\xi^2\eta + \beta_8\xi\eta^2 \end{aligned}$$

2D/3D Finite elements

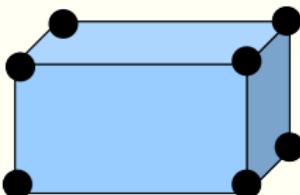
2D Consolidation element



- Pore pressure and displacements
- Displacements

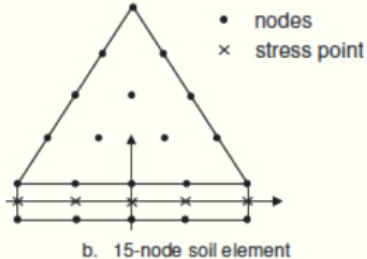
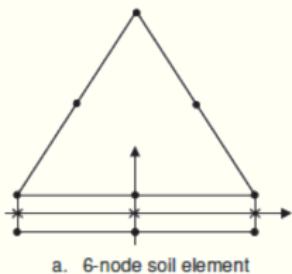
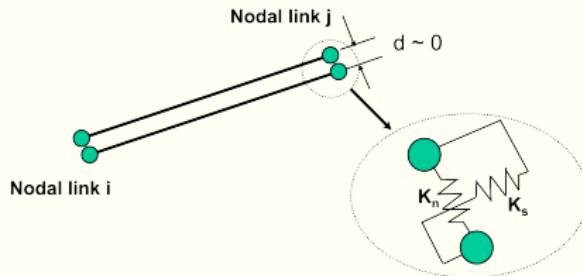
Linear variation of pore pressures and quadratic variation of displacements in x and y directions

8 node 3D brick element



Linear variation of displacements in x, y and z directions

Interface element



13 Distribution of nodes and stress points in interface elements and their connection to soil elements

13

Use a reduced strength at the interface

Interface elements for Soil Structure Interactions

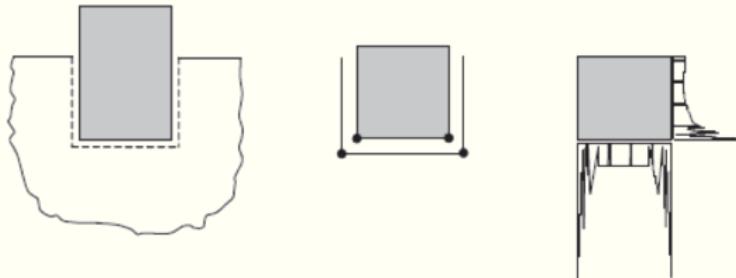


Figure 3.14 Inflexible corner point, causing poor quality stress results

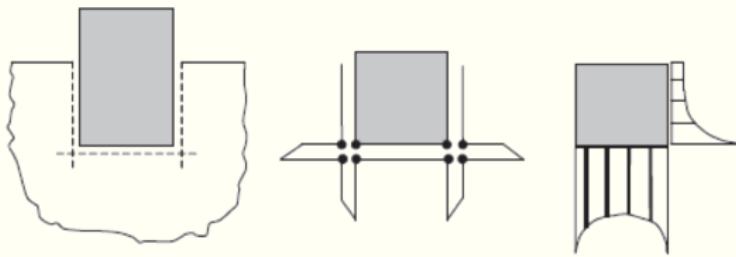
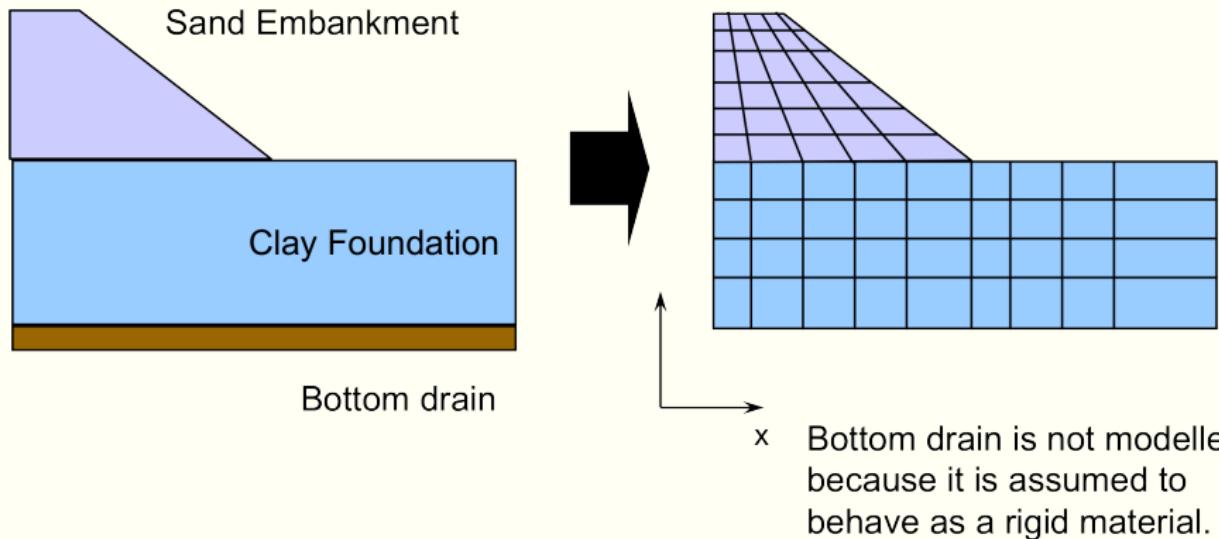
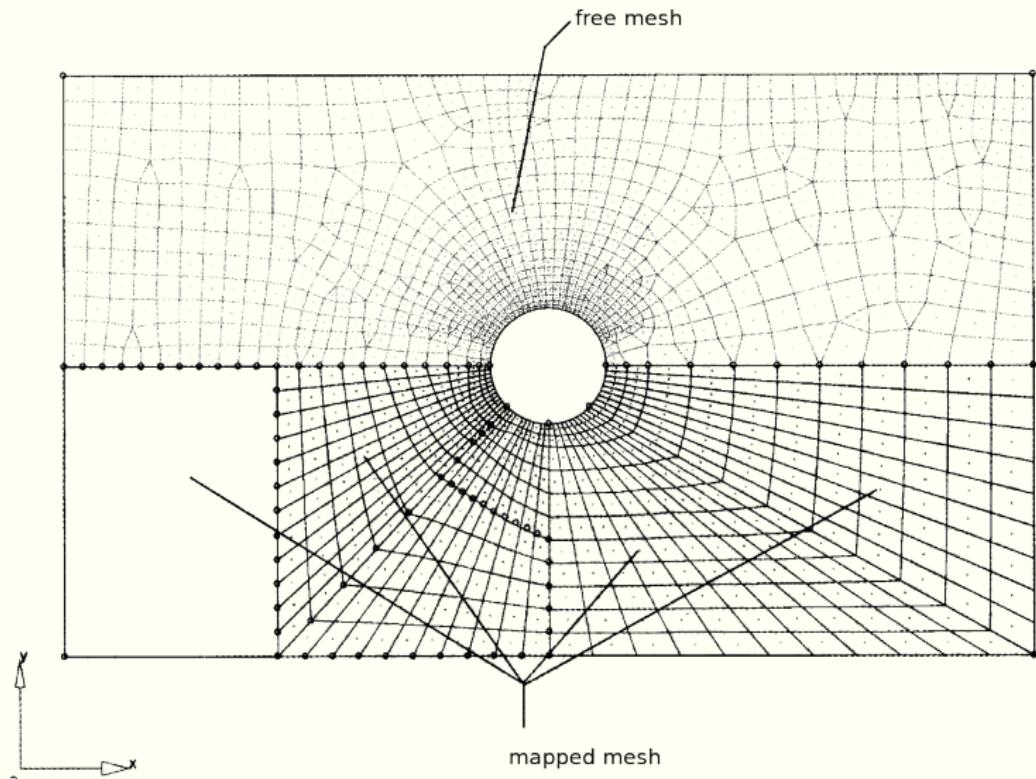


Figure 3.15 Flexible corner point with improved stress results

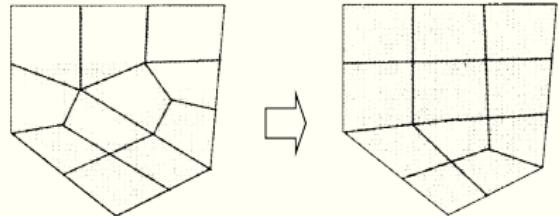
FE discretization



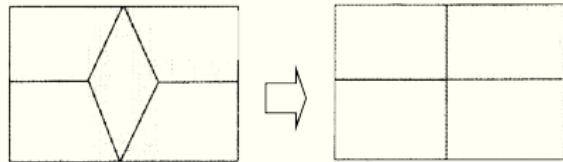
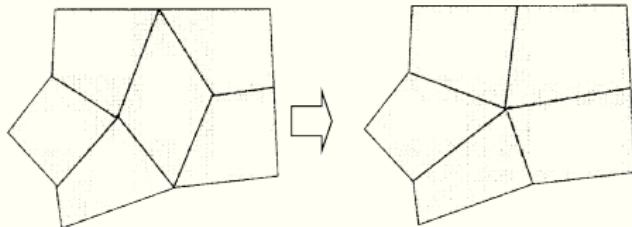
FE discretization



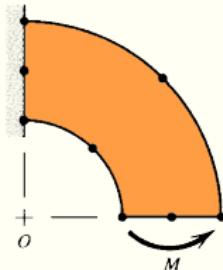
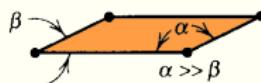
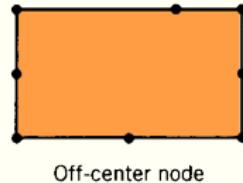
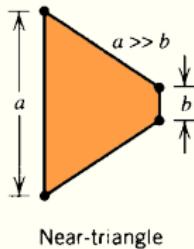
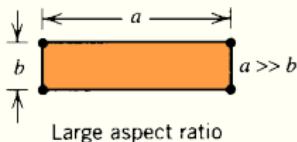
FE discretization



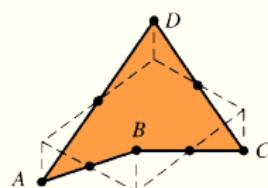
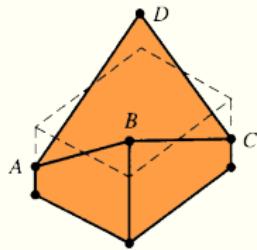
- ① Node valence less or equal to two and greater than or equal to six should be eliminated.
- ② The number of nodes with valence of three or five should be minimized.
- ③ Angles greater than 160 degrees should be eliminated.
- ④ The aspect ratio should be less than 3 for stress analysis and 10 for displacement analysis.



FE discretization

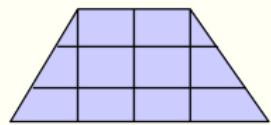


Large arc angles

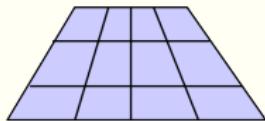


Shell element ABCD
greatly warped

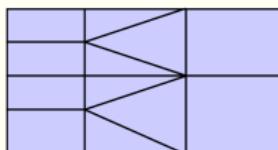
FE discretization



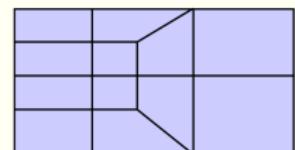
Not recommended



Better

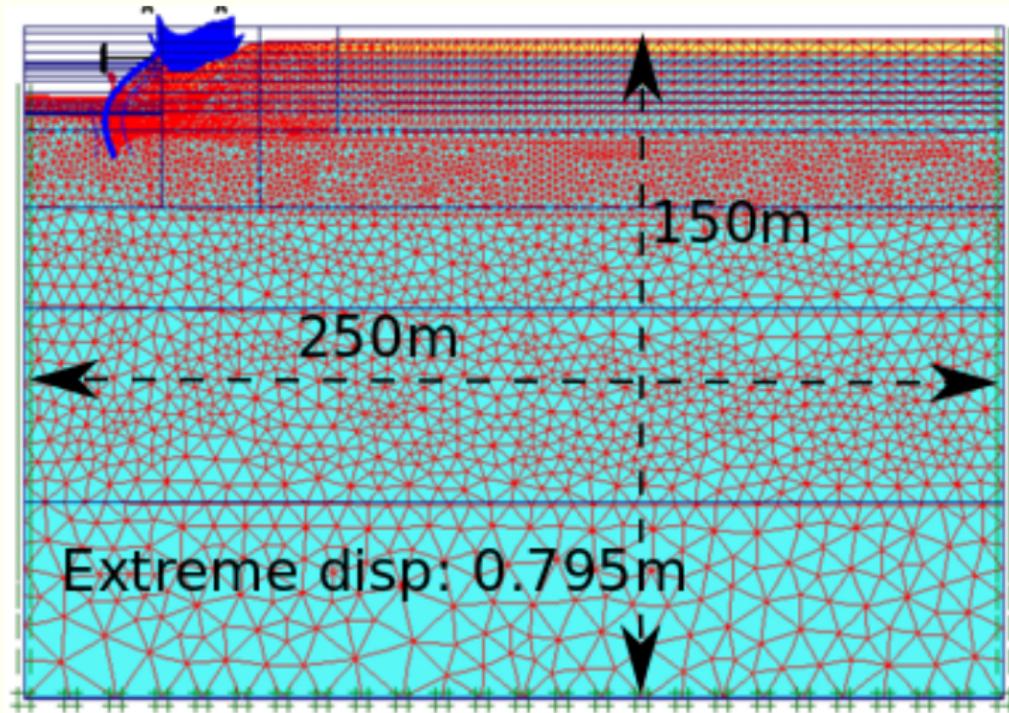


Not recommended



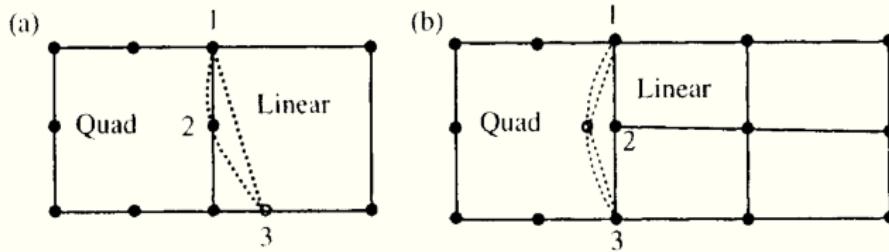
Better

FE discretization: Refining



Avoid large jumps in element size: size jump should be < 3

FE mesh compatibility



Incompatible mesh

FE boundary conditions

x direction fixed

y direction free

pore pressure fixed (if embankment
is assumed to be fully drained
condition)

Sand embankment

x and y directions free
pore pressure fixed

x direction fixed

y direction free

pore pressure free

x direction fixed

y direction free

pore pressure free

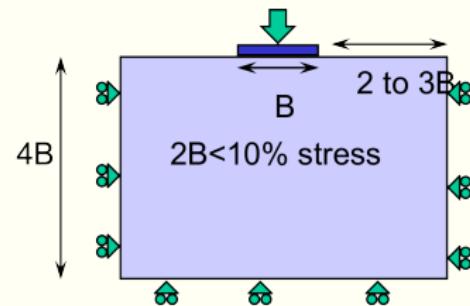
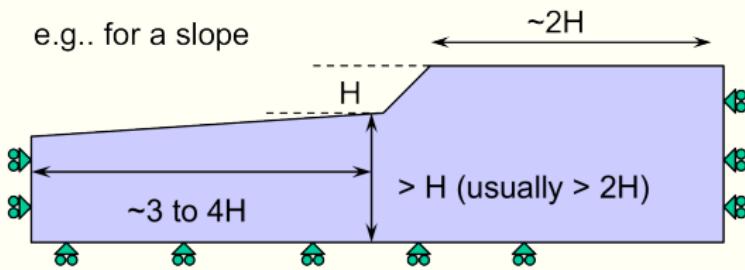
Clay foundation

x and y directions fixed

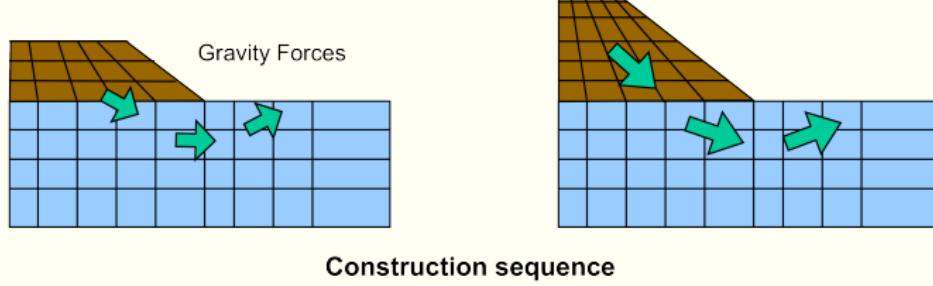
pore pressure fixed

Pore pressure free = no water flow perpendicular to the boundary

FE boundary conditions

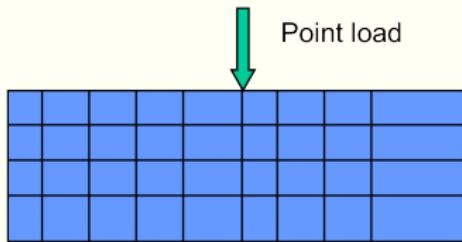


FE boundary conditions



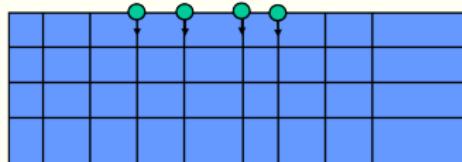
FE loading conditions

External forces



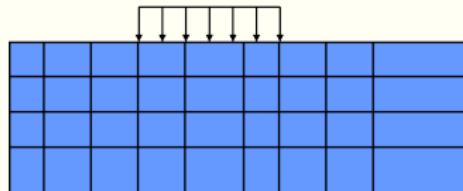
Point load

Apply displacements



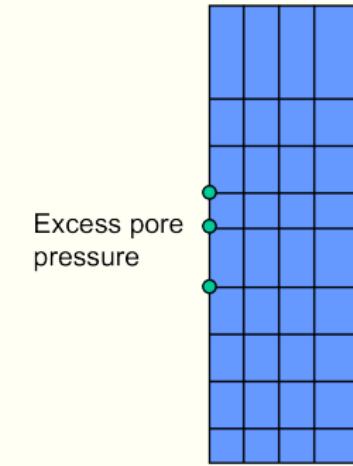
Rigid foundation

Distributed load

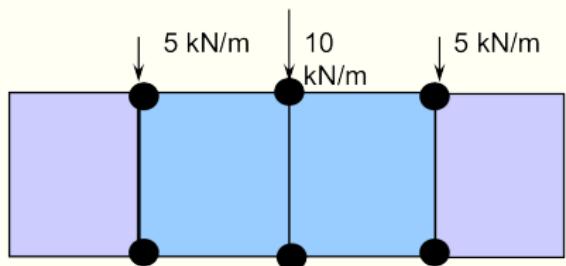
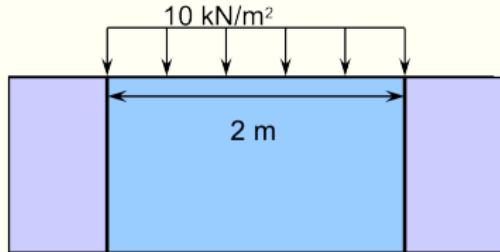


Excess pore pressure

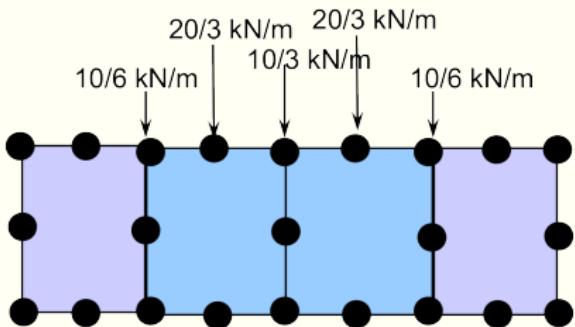
Water pumping



FE loading conditions

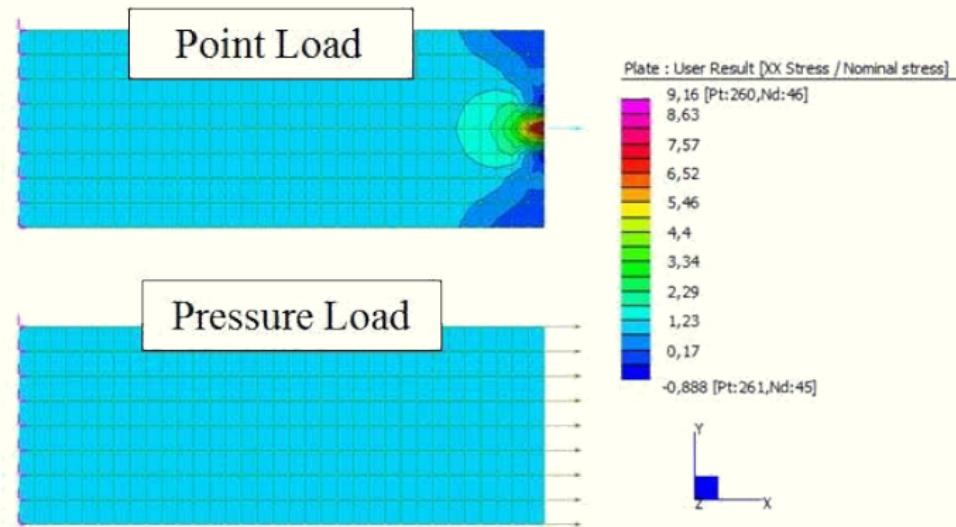


(1) 4 nodes elements



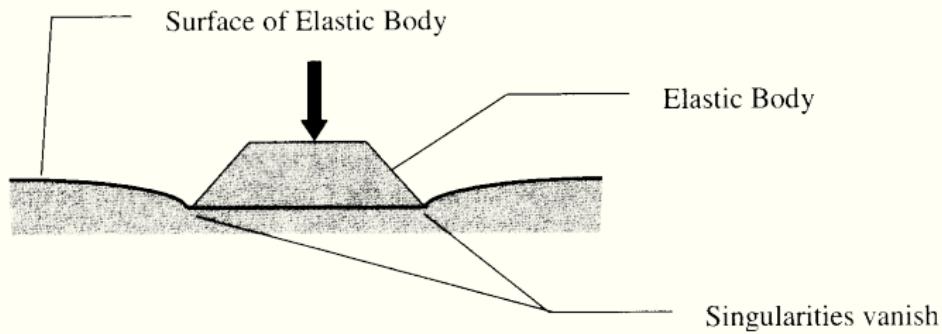
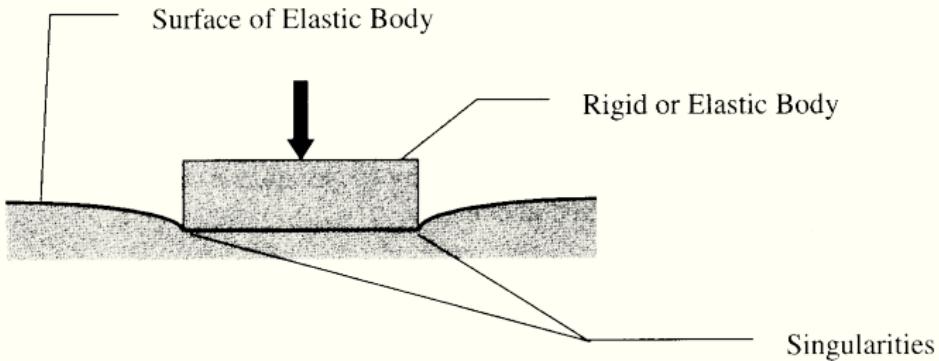
(2) 8 nodes element

FE stress singularity



2D uniaxial bar model. A single point load leads to stress singularities whereas an edge pressure results in a uniform stress field. (Gonzalez., 2015)

FE stress singularity

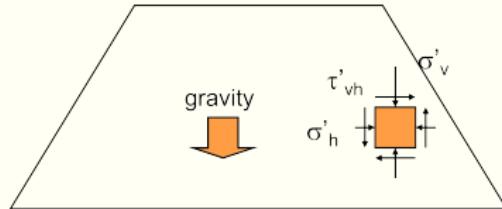
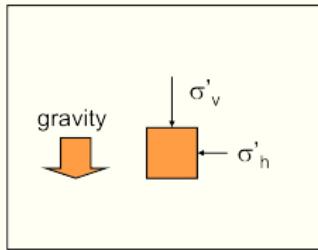


Geostatic stresses

Before conducting your analysis, you need to make sure that the stresses in the ground are the correct values, as the soil behavior depends on the current in-situ stresses.

- ① **list your estimated stresses in the input file:** hopefully the system is in equilibrium – difficult to find the in-situ stresses in the sloping ground.

Horizontally levelled ground

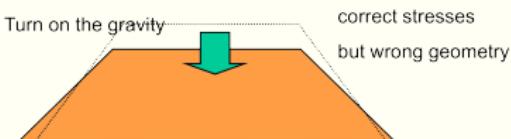


Geostatic stresses

- ② **Zero displacement approach** - ask the program to compute the in-situ stresses from the equilibrium condition (very few programs allow you to do this)



(a) define model geometry and assign a soil model



(b) displacement by self weight and obtain the equilibrium condition



(c) Zero the displacement
But keep the computed stresses

- ③ **Intermediate approach** - Guess the insitu stress distribution, apply gravity and perform the equilibrium check (hopefully the displacements are zero) - ABAQUS GEOSTATIC approach

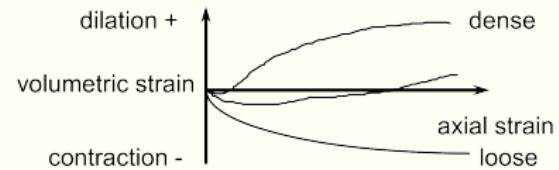
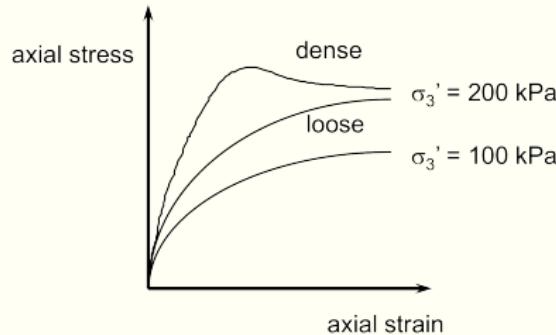
The coefficient of earth pressure at rest K_0

- **Normally consolidated soils**

- **Over consolidated soils**

- **Cam-clay predictions** $K_{oc} = (1 - \sin \phi') \times (\text{OCR})$ (Mayne et al., 1982) up to $K_p = (1 + \sin \phi') / (1 - \sin \phi')$
- **Wroth's method (1975)** $K_{oc} = (\text{OCR})K_{nc} - \left(\frac{\nu'}{1-\nu'} \right) (\text{OCR} - 1)$ for $\text{OCR} < 5$ and ν' is the poisson's ratio = 0.254 - 0.371.

Soil models

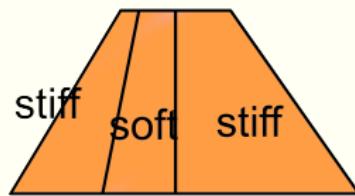


Verification of Design Parameters (Atkinson., 1995)

Verification of Design Parameters (Atkinson., 1995)

FE errors

Check! Check! Check!



Beware : σ_v here lower than γh due to stress transfer to stiff material - hanging up

Steps to perform Finite Element Analysis

Summary of FE analysis

