CE394M: Finite Element Analysis in Geotechnical Engineering

Krishna Kumar

University of Texas at Austin

krishnak@utexas.edu

March 7, 2019

Overview

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

FE Disclaimer

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.

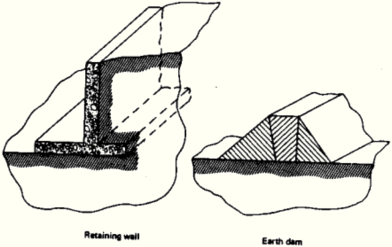
3

Consistent system of units

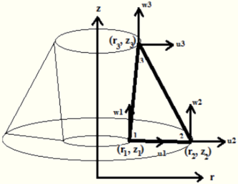
	SI			
Length	m	m	m	cm
Density	kg/m^3	$10^3\ kg/m^3$	$10^6\ kg/m^3$	$10^6\ g/cm^3$
Force	N	kN	MN	Mdynes
Stress	Pa	kPa	MPa	bar
Gravity	m/sec^2	m/sec^2	m/sec^2	cm/s^2
Stiffness*	Pa / m	kPa / m	MPa / m	bar / cm

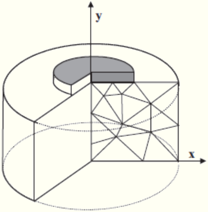
Problem definition

Plane Strain: Strain normal to x-y plane is zero $\varepsilon_z = 0$ and shear strains γ_{zy} and γ_{zx}



Axisymmetric:

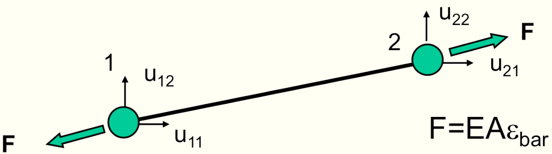




Take advantage of symmetry

1D Finite Elements: Bar element

Two node element with axial stiffness only (no flexural or shear resistance).Examples of this type of structure are cables, reinforcing bars.



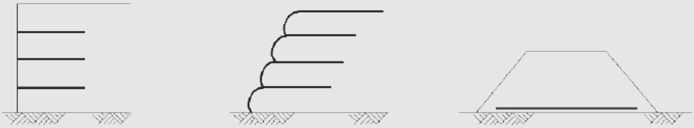
Two node element with axial stiffness only (no flexural or shear resistance).Examples of this type of structure are cables, reinforcing bars.

1. Node to Node: are springs that are used to model ties between two points.
2. It's not recommended to draw geometry line at position where node-to-node anchor is to be placed.
3. It's a 2 node elastic spring element with normal stiffness (Spring constant)
4. Element can sustain both tensile forces (anchors) as well as compressive forces (struts).
5. Fixed-End anchors: Modelling of struts or props to sheet pile walls.

1D Finite Elements: Bar element

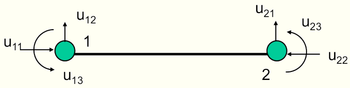
Two node element with axial stiffness only (no flexural or shear resistance).Examples of this type of structure are cables, reinforcing bars.

1. Geogrids are slender structures with a normal stiffness but no bending stifness.
2. Geogrids can only sustain **Tensile forces** and no compression!
3. Structures involving geotextiles.



1D Finite Elements: Beam element

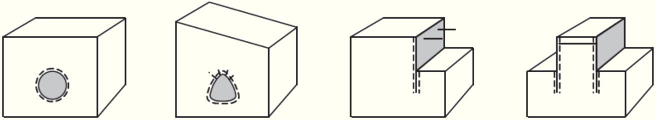
two node structure element with axial and bending stiffness (no transverse shear deformation). Three degrees of freedom for 2D beam element (1, 2 displacements and a moment). Examples are sheet pile walls, structural foundation beams, structural facing for reinforced soil walls.



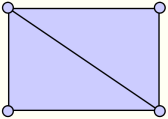
$F_a = EA\,\varepsilon_a$

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

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



2D plane-strain / axisymmetric elements

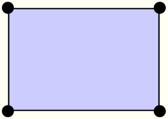


3 nodes element

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

$$d_1 = \alpha_1 + \alpha_2 x + \alpha_3 y$$

$$d_2 = \beta_1 + \beta_2 x + \beta_3 y$$

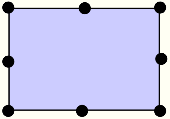


4 nodes element

linear variation of displacement in both x and y directions

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

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



8 nodes element

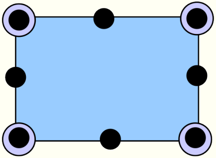
quadratic variation of displacement in both x and y directions.

$$d_1 = \alpha_1 + \alpha_2\xi + \alpha_3\eta + \alpha_4\xi^2 + \alpha_5\xi\eta + \alpha_6\eta^2 + \alpha_7\xi^2\eta + \alpha_8\xi\eta^2$$

$$d_2 = \beta_1 + \beta_2\xi + \beta_3\eta + \beta_4\xi^2 + \beta_5\xi\eta + \beta_6\eta^2 + \beta_7\xi^2\eta + \beta_8\xi\eta^2$$

2D/3D Finite elements

2D Consolidation element

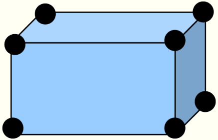


Pore pressure and displacements

Displacements

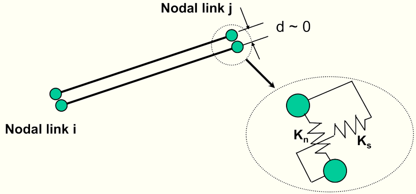
Linear variation of pore pressures and quadratic variation of displacements in x and y directions

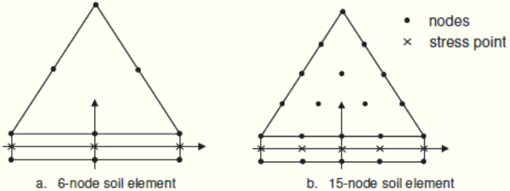
8 node 3D brick element



Linear variation of displacements in x, y and z directions

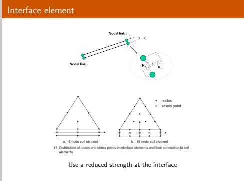
Interface element





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

Use a reduced strength at the interface



This element allows relative displacement between elements. It is capable to model soil/structure interface conditions, shear planes within a soil mass. The element is 'fictitious' four node element made up of two independent nodal links. Each link consists of two nodes connected by a normal and shear spring as shown below. The stiffness of the springs can be non-linear, modelling frictional slip behaviour. The thickness of the element is assumed to be negligibles

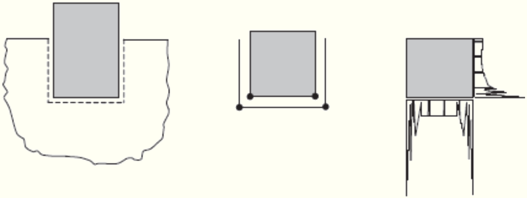


Figure 3.14 Inflexible corner point, causing poor quality stress results

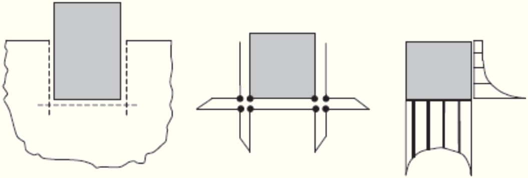
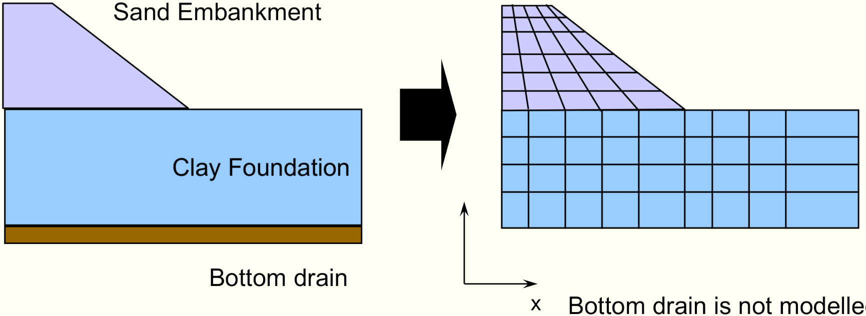


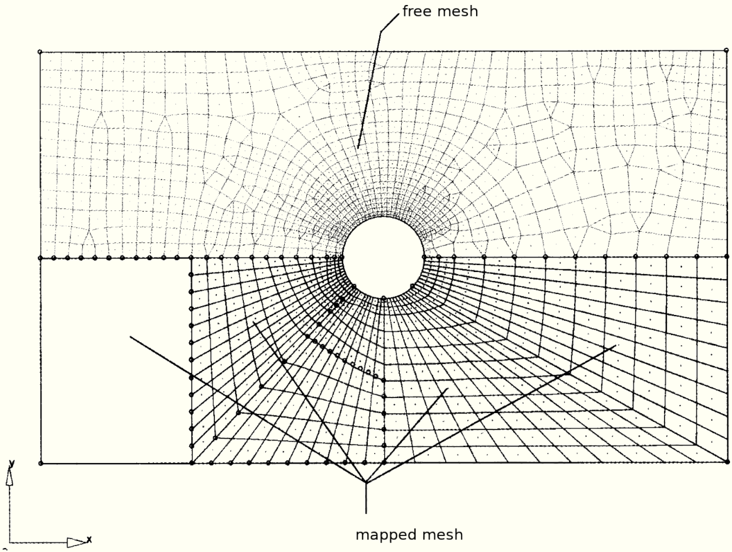
Figure 3.15 Flexible corner point with improved stress results

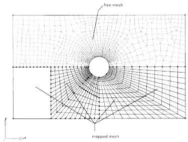
FE discretization



Bottom drain is not modelled because it is assumed to behave as a rigid material.

FE discretization

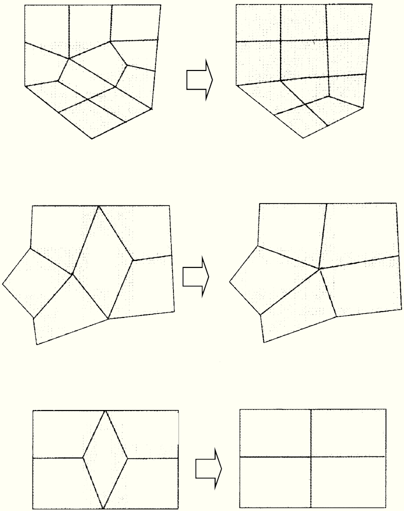




There are many algorithms to generate mesh for a given object. The top part of the geometry is a mesh generated by "*free meshing*", in which elements are generated automatically with triangles and quadrilaterals within the whole domain. The bottom part of the geometry is a mesh generated by "*mapped meshing*", in which quadrilateral elements are placed in subzones with a more regular manner. The mesh density is also more controlled.

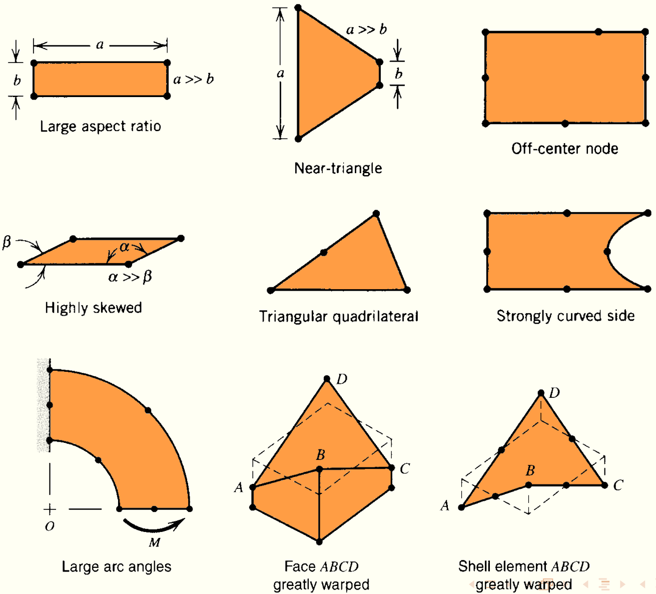
Some elements generated in the previous figure show that the interior nodes are shared by more than one element. The number of elements that share a node is called valence. The ideal valence of interior nodes for quadrilaterals is four to achieve the optimum interior angle of 90 degrees. For triangles, it is six. After a mesh is generated, it is ideal to clean up the mesh.

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

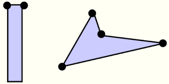


FE discretization

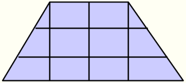
(1) Use big graph paper / draw in boundaries

(2) Keep elements as “Square” as possible.

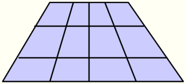
Do not want :



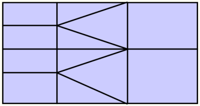
(3) For 4 node elements, try to avoid triangular



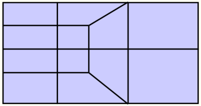
Not recommended



Better

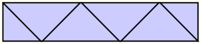


Not recommended

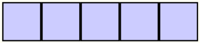


Better

(4) Use regular mesh

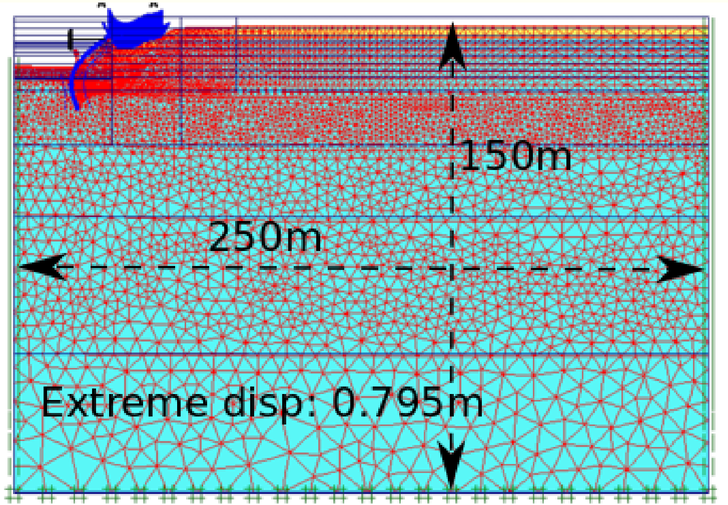


Not recommended



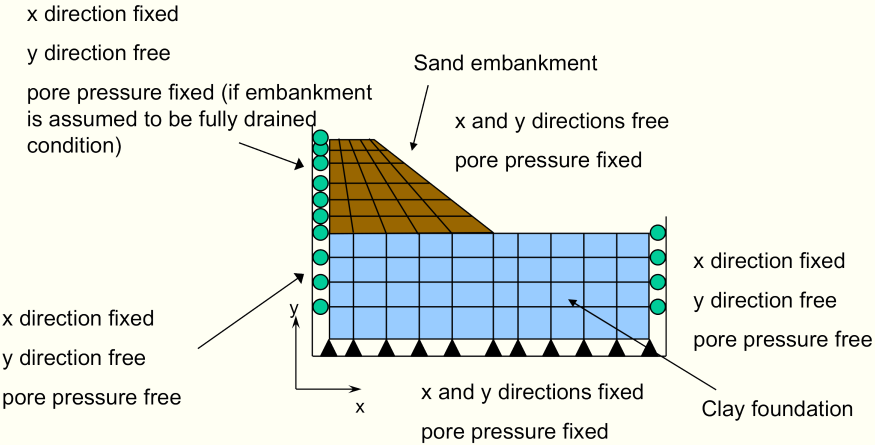
Better

FE discretization: Refining



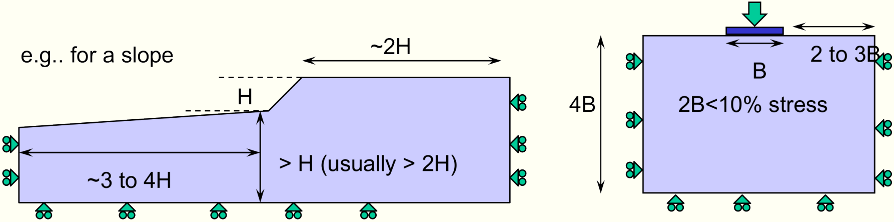
Avoid large jumps in element size: size jump should be < 3

FE boundary conditions



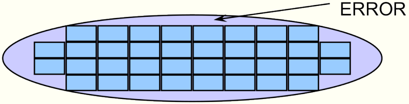
Pore pressure free = no water flow perpendicular to the boundary

FE boundary conditions



FE errors

(1) Creating elements



(2) Numerical errors - e.g. finite element approximation (variation within the elements), numerical integration and time integration, mesh locking

(3) Constitutive model - dominant error for us.

Liner elastic model?

Non-linear elastic model?

Cam-clay model?

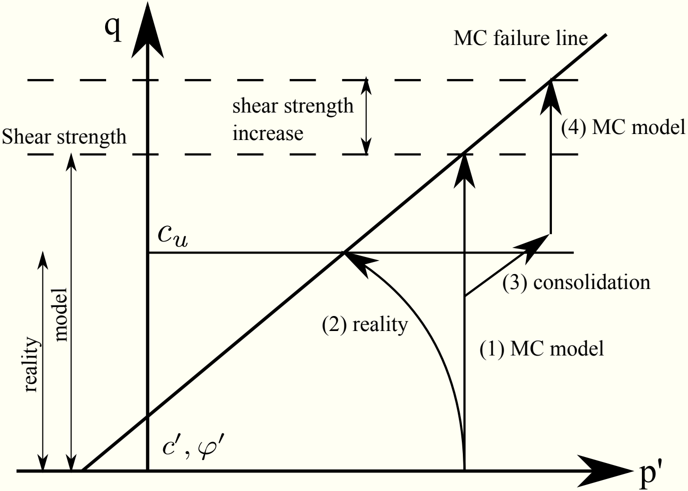
More sophisticated models?

(4) Modelling the boundary conditions - anything we do to approximate boundary conditions introduces error.

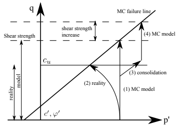
Undrained analysis

- Method A and Method B refers to 2 alternatives modeling of undrained behaviour in Plaxis.
- **Method A is an effective stress analysis** of an undrained problem it assumes an isotropic elastic behavior and a Mohr-Coulomb failure criterion.
- As a result mean effective stress p' is constant until yield.
- Method A was being applied to marine clays which were of low over-consolidation or even under-consolidated because of recent reclamation.

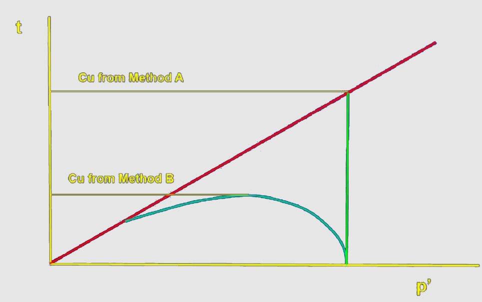
Undrained effective stress analysis



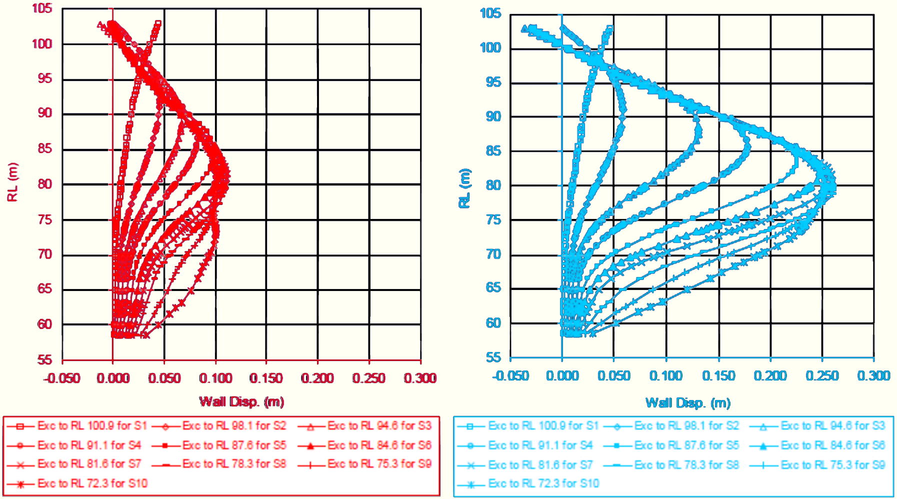
Undrained effective stress analysis



Discuss the effect of depth on the increase in shear strength

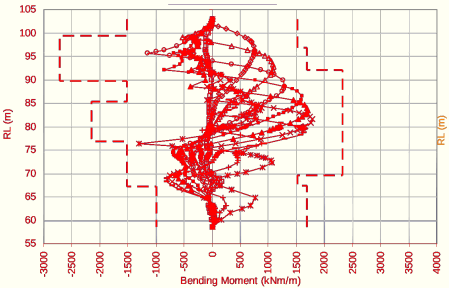


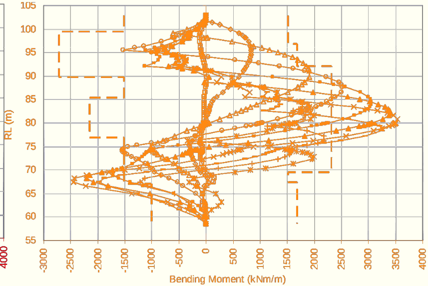
Wall displacements: Effective stress vs Undrained strength



Nicoll Highway Collapse: Method A vs Method B

Bending moments: Effective stress vs Undrained strength





Nicoll Highway Collapse: Method A vs Method B