CE394M: Finite Element Analysis in Geotechnical Engineering

Krishna Kumar

University of Texas at Austin

krishnak@utexas.edu

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.

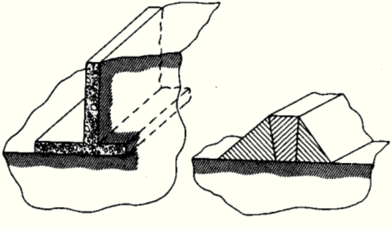
3

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

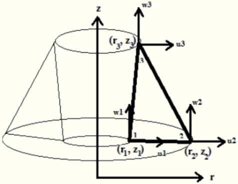
Plane Strain: Strain normal to x-y plane is zero $\varepsilon_z = 0$ and shear strains γ_{zy} and γ_{zx}

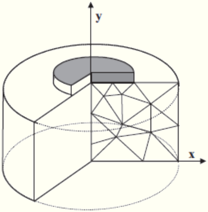


Retaining wall

Earth dam

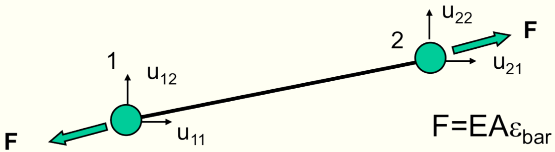
Axisymmetric:





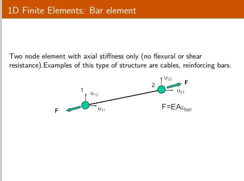
Take advantage of symmetry

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

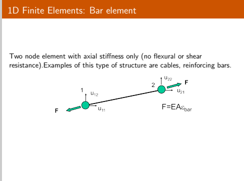


2021-03-05

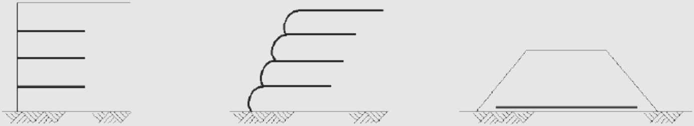
CE394M: FEM Geo
└─Geotechnical FEA
└─Element types
└─1D Finite Elements: Bar element



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.

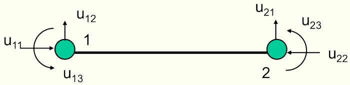


1. Geogrids are slender structures with a normal stiffness but no bending stifness.
2. Geogrids can only sustain **Tensile forces** and no compression!
3. Structures involving geotextiles.



1D Finite Elements: Beam element

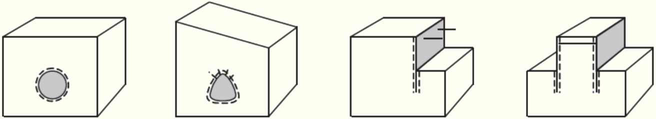
two node structure element with axial and bending stiffness (no transverse shear deformation). Three degrees of freedom for 2D beam element (1, 2 displacements and a moment). Examples are sheet pile walls, structural foundation beams, structural facing for reinforced soil walls.

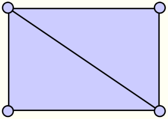


$$F_a = EA\,\varepsilon_a$$

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

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



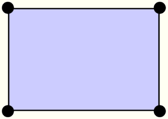


3 nodes element

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

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

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

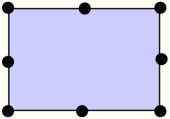


4 nodes element

linear variation of displacement in both x and y directions

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

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



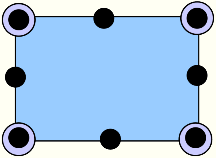
8 nodes element

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 Consolidation element

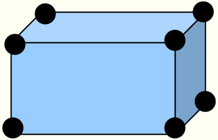


Pore pressure and displacements

Displacements

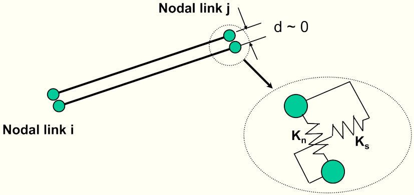
Linear variation of pore pressures and quadratic variation of displacements in x and y directions

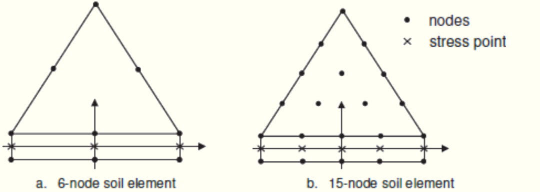
8 node 3D brick element



Linear variation of displacements in x, y and z directions

Interface element

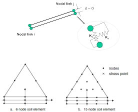




13 Distribution of nodes and stress points in interface elements and their connection to soil elements [13]

Use a reduced strength at the interface

Interface element



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

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

Interface elements for Soil Structure Interactions

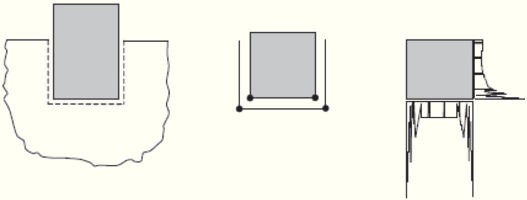


Figure 3.14 Inflexible corner point, causing poor quality stress results

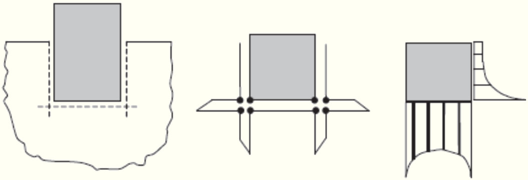
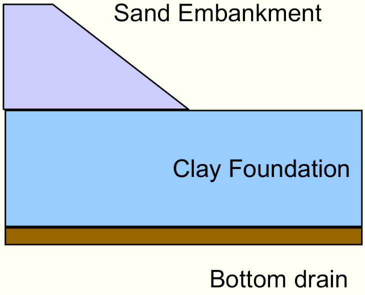
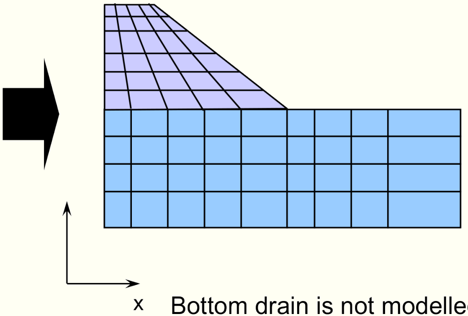


Figure 3.15 Flexible corner point with improved stress results

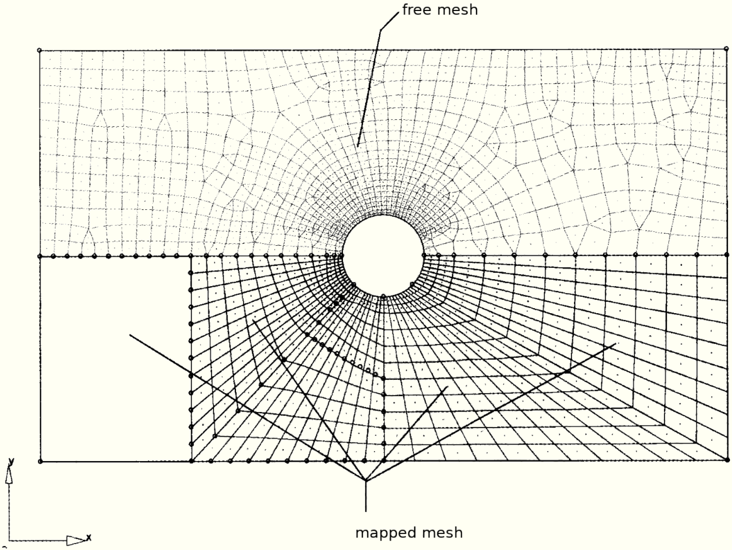
FE discretization

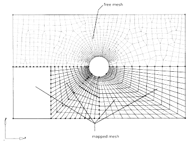




Bottom drain is not modelled because it is assumed to behave as a rigid material.

FE discretization

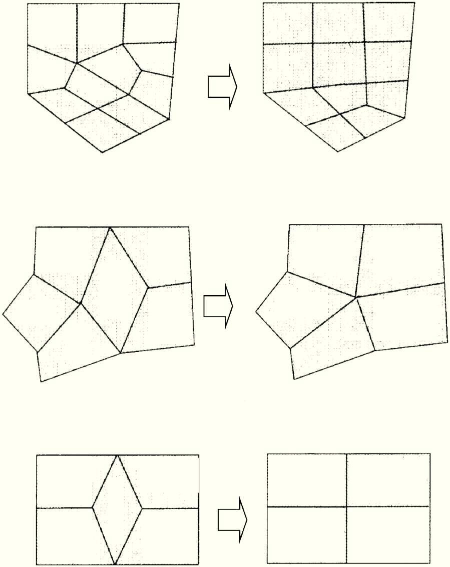




There are many algorithms to generate mesh for a given object. The top part of the geometry is a mesh generated by "*free meshing*", in which elements are generated automatically with triangles and quadrilaterals within the whole domain. The bottom part of the geometry is a mesh generated by "*mapped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

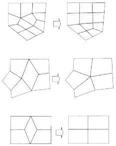
Some elements generated in the previous figure show that the interior nodes are shared by more than one element. The number of elements that share a node is called valence. The ideal valence of interior nodes for quadrilaterals is four to achieve the optimum interior angle of 90 degrees. For triangles,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

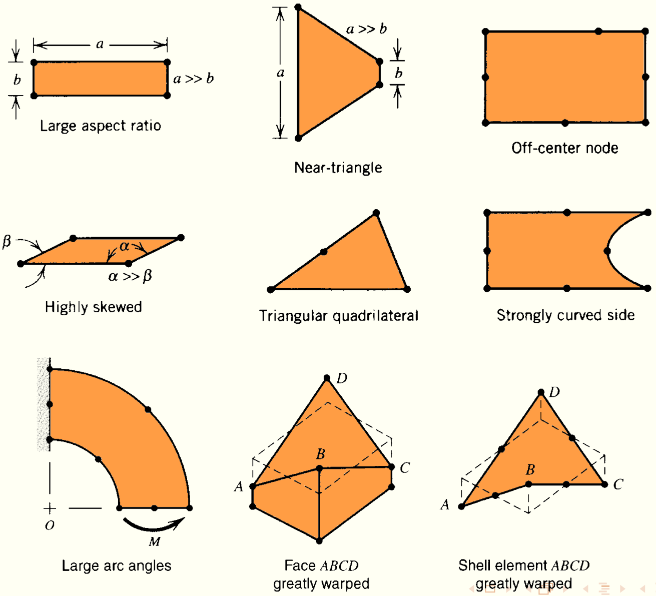
FE discretization



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

- A square element is ideal.
- Minimize "*skew*": $||\text{angle} - 90\,\text{deg}||$
- Minimize aspect ratio: $\frac{||\text{longestedge}||}{||\text{shortestedge}||}$
- Keep bad elements away from the boundary

FE discretization

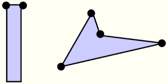


FE discretization

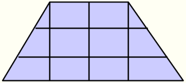
(1) Use big graph paper / draw in boundaries

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

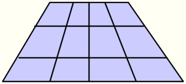
Do not want :



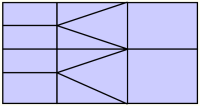
(3) For 4 node elements, try to avoid triangular



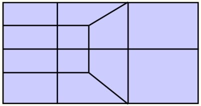
Not recommended



Better

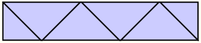


Not recommended



Better

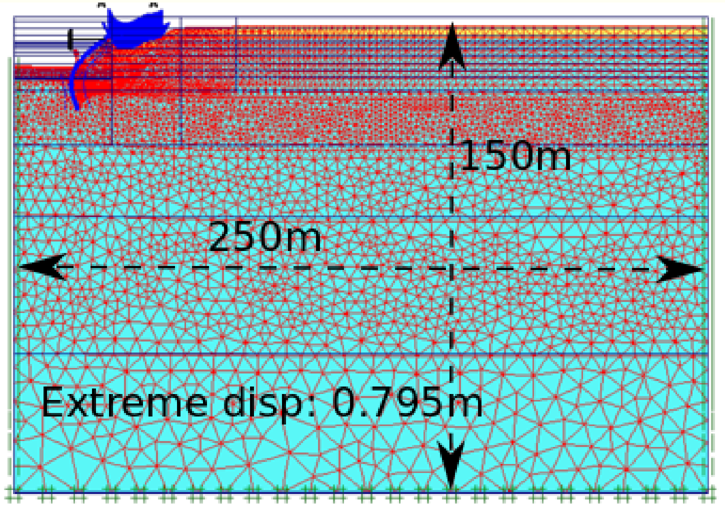
(4) Use regular mesh



Not recommended

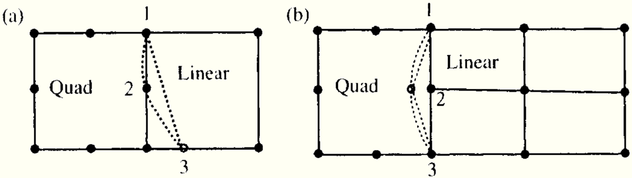
Better

FE discretization: Refining

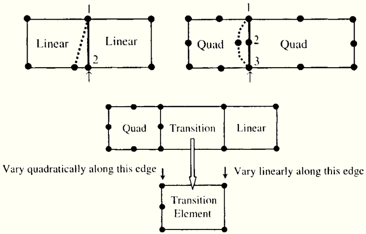


Avoid large jumps in element size: size jump should be < 3

FE mesh compatibility



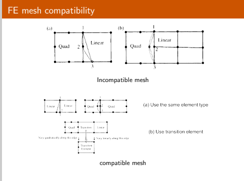
Incompatible mesh



(a) Use the same element type

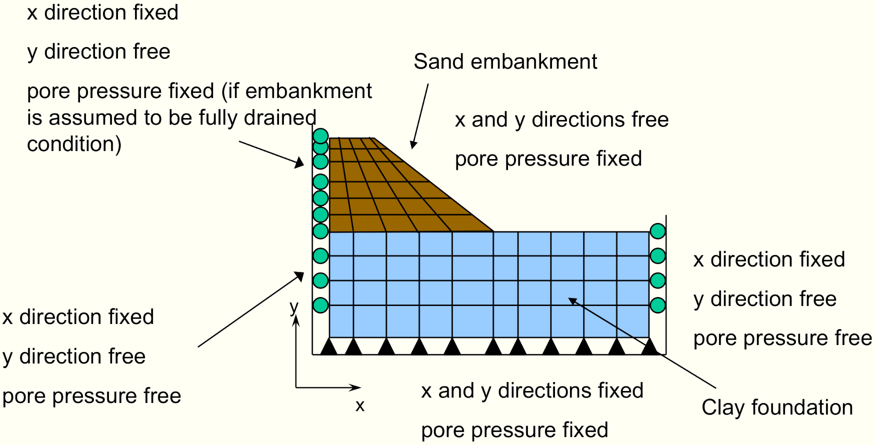
(b) Use transition element

compatible mesh



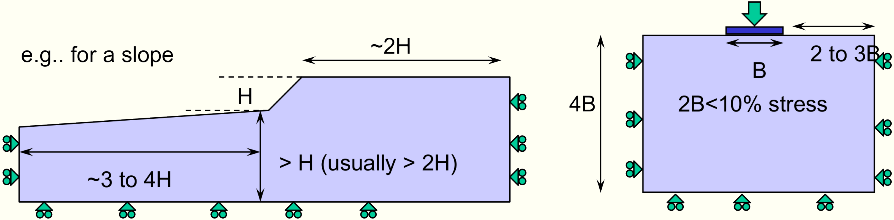
A mesh is said to be 'compatible' if the unknown variables are continuous along all edges between all the elements in the mesh. The use of different types of elements in the same mesh or improper connection of elements can result in an incompatible mesh. The examples of this problem are given in Fig. 1. Solutions of this problem are (i) use the same type of elements throughout the entire domain (Fig. 2(a)) or (ii) use transitional elements whose shape functions have different orders on different edges (Fig. 2(b)).

FE boundary conditions



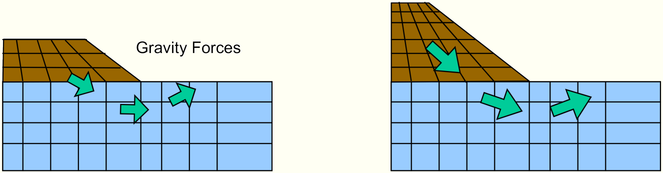
Pore pressure free = no water flow perpendicular to the boundary

FE boundary conditions



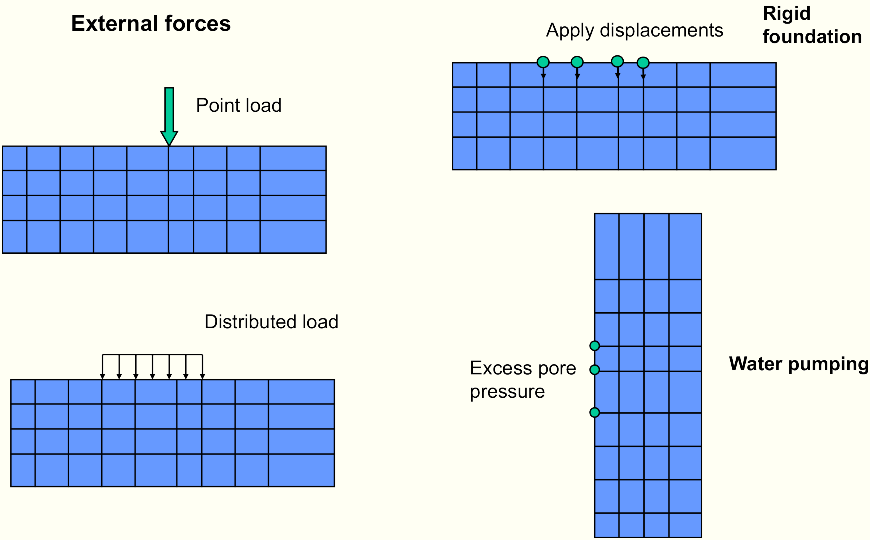
FE boundary conditions

1. **Body force**: apply unit weight
2. **Force**: node is free
3. **Displacement**: node is fixed
4. **Pore pressure**: node fixed to apply external pore pressures

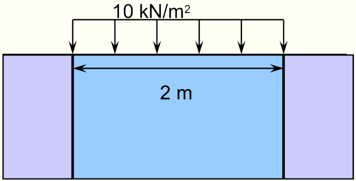


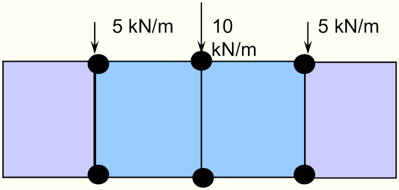
Construction sequence

FE loading conditions

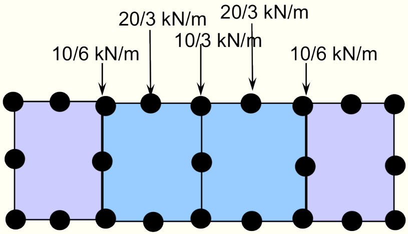


FE loading conditions



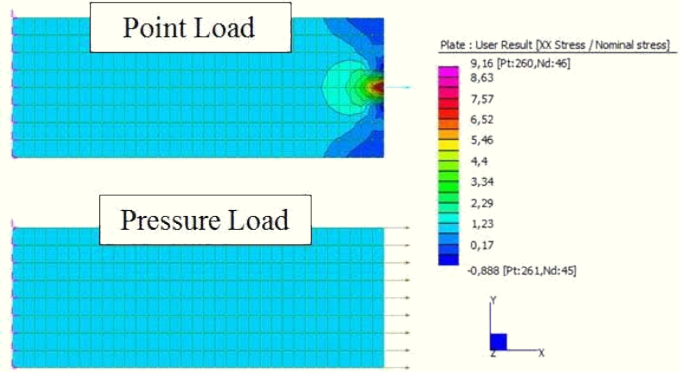


(1) 4 nodes elements



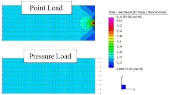
(2) 8 nodes element

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



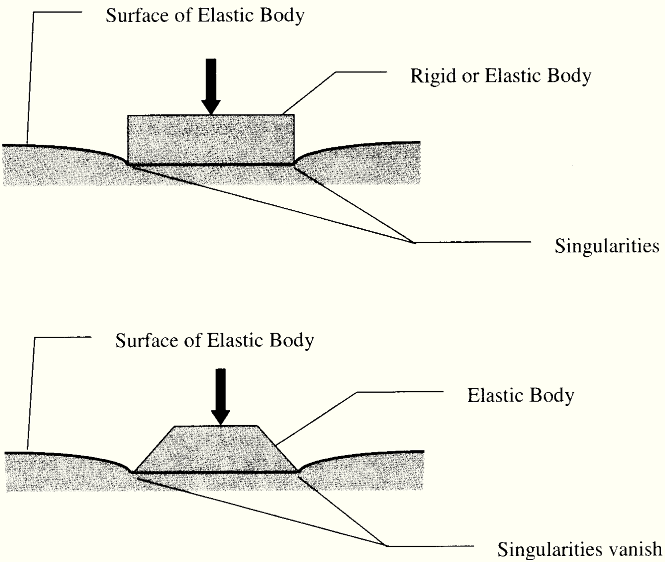
2D uniaxial bar model. A single point load leads to stress singularities whereas an edge pressure results in a uniform stress field. (Gonzalez., 2015)

A stress singularity is a point of the mesh where the stress does not convergence towards a specific value. As we keep refinement the mesh, the stress at this point keeps increasing, and increasing, and increasing... Theoretically, the stress at the singularity is infinite.

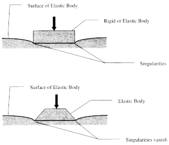
Although stress at these singularities is infinite, this does not mean that the model results are incorrect overall. First of all, the displacements are correct even at the singularity point. On the contrary, the stress at the singularity will pollute the stress results near the singularity, however some distance away from the singularity the stress results will be fine!

As you can see, far away from the point load singularity the stress field is uniform despite of the local singularity. Note that pressure load do not cause point singularities, this is because the pressure attribute is converted by the solver into a set of point loads along the nodes of the edge in a consistent fashion that do not cause a singularity.

FE stress singularity



FE stress singularity



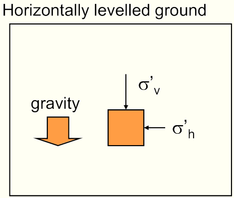
If a concentrated force is applied on a continuum, it results in unbounded stresses, which is called stress singularities. The values of stresses become a function of mesh density. The denser the mesh is, the better the result becomes. It is often better to replace a concentrated load by a load distributed over a small region to avoid the effect of stress singularities.

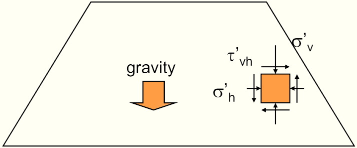
A sharp re-entrant corner as shown in the figure (a) also produces singularity (i.e. solution is mesh dependent and the zones close to the singularity is becomes polluted). The nature of the singularity depends on the elastic properties of the two bodies. It may be possible to weaken or eliminate the singularity by changing the rectangular object to a tapered one as shown in the figure (b).

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.

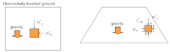




Geostatic stresses

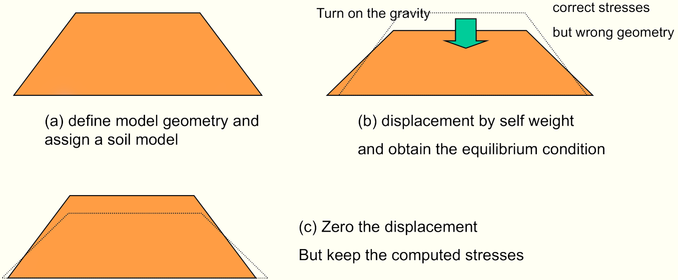
Before conducting your analysis, you need to make sure that the stresses in the ground are the correct values, as the soil behavior depends on the current in-situ stresses.

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.



1. Vertical effective stress σ'_v from the unit weight of the soil
2. Horizontal effective stress $\sigma'_h = K_0 \sigma'_v$
3. Estimation of in-situ stresses becomes difficult - depends on soil model used.
4. **Total stress analysis – need to give initial total 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)

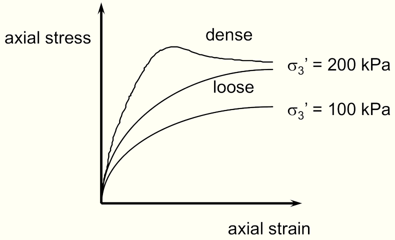


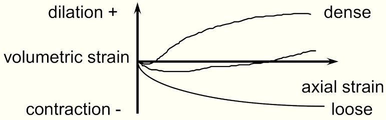
3. **Intermediate approach** - Guess the insitu stress distribution, apply gravity and perform the equilibrium check (hopefully the displacements are zero) - ABAQUS GEOSTATIC approach

The coefficient of earth pressure at rest K_0

- **Normally consolidated soils**
  - Cam-clay predictions (Wood, 1990). K_0 (Modified CC) $<$ K_0 (Original CC)
  - $K_0 = K_{\mathrm{nc}} = 1 - \sin\phi'$ (Jaky, 1944) (ϕ' is the friction angle of the soil).
- **Over consolidated soils**
  - **Cam-clay predictions** $K_{\mathrm{oc}} = (1 - \sin\phi') \times (\mathrm{OCR})$ (Mayne et al., 1982) up to $K_p = (1 + \sin\phi')/(1 - \sin\phi')$
  - **Wroth's method (1975)** $K_{\mathrm{oc}} = (\mathrm{OCR})K_{\mathrm{nc}} - \left(\frac{\nu'}{1-\nu'}\right)(\mathrm{OCR} - 1)$ for OCR < 5 and ν' is the poisson's ratio $=$ 0.254 - 0.371.

Soil models





1. **Sand** - Linear elastic model, Non-liner elastic model, Drucker-Prager elasto-plastic model, Mohr-coulomb elasto-plastic model or Advanced models ?
2. **Clay** - Linear elastic model, Non-linear elastic model, Cam-clay model, or Advanced models ?

Verification of Design Parameters (Atkinson., 1995)

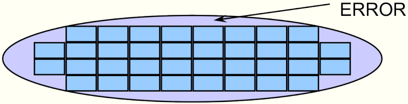
1. Are the soil models and analyses being used appropriate for the soils and for the structure?
2. Are the design parameters being measured appropriate for the soil models being used?
3. Are the tests being carried out the appropriate ones to determine the required design parameters?
4. Is the laboratory where the tests are being done capable of doing the required tests? Ideally, the engineer should inspect the laboratory and equipment, observe tests and check the procedures being used to analyse and interpret the results.
5. Are the correct samples being used? Are the appropriate methods of sample preparation being used? Do you need high quality samples or reconstituted samples?

Verification of Design Parameters (Atkinson., 1995)

1. Have any tests been done which investigate whether the soils reasonably follow the critical state theories or the elasto-plastic theories being used in the analyses?
2. Have tests being carried out at stress levels corresponding to the range of stresses in the ground? Is it necessary to follow special stress paths representing the previous stress history and the current loading?
3. Have linear parameters been fitted to non-linear test results over appropriate ranges?
4. Are the design parameters internally self consistent? Do parameters determined from triaxial or other loading tests correspond to parameters estimated form the grading and nature of the grains? How the values compare to the values obtained from empirical equations?

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?

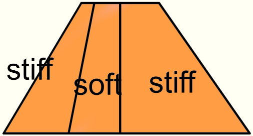
Cam-clay model?

More sophisticated models?

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

Check! Check! Check!

- Use well documented field study or model study
- Do hand calculations or use chart solution
- Check orientations of the maximum principal stress directions - tells how loads flowing through the system
- Check failure parameters-how close to failure line, look to see whether reasonable
- check the vertical stress σ_v and σ_h.

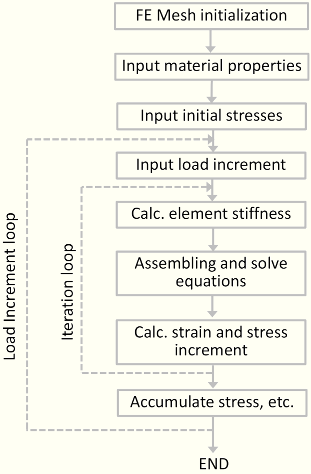


Beware : σ_v here lower than γh due to stress transfer to stiff material - hanging up

Steps to perform Finite Element Analysis

1. **Define the problem**
   - Total stress analysis, effective stress analysis or consolidation analysis?
   - What are the unknowns?
   - Spatial dimension (plane strain, axisymmetric, 3D)
   - Element types
2. **Create finite element mesh**
3. Proper discretization and avoid bad elements
4. **Define analysis (or construction sequence) steps**
   - apply loading and boundary conditions
5. **Define materials**
   - determine material properties
6. **Assign materials and element types to elements**
7. **Define in-situ stress conditions**
8. **Run the analysis**
9. **Check! Check! Check!**

Summary of FE analysis



$\boldsymbol{\sigma}$

$\Delta \boldsymbol{f}$

$$K_{ij} \approx \sum_{p}^{n_p} B_i^p D^p B_j^p W_p$$

$\Delta \boldsymbol{u} = \boldsymbol{K}^{-1} \cdot \Delta \boldsymbol{f}$ **Equilibrium eq.**

$\Delta \boldsymbol{\varepsilon} = \boldsymbol{B} \cdot \Delta \boldsymbol{u}$ **Compatibility eq.**

$\Delta \boldsymbol{\sigma} = \boldsymbol{D} \cdot \Delta \boldsymbol{\varepsilon}$ **Constitutive eq.**

$$\boldsymbol{u} = \boldsymbol{u} + \Delta \boldsymbol{u}$$
$$\boldsymbol{\varepsilon} = \boldsymbol{\varepsilon} + \Delta \boldsymbol{\varepsilon}$$
$$\boldsymbol{\sigma} = \boldsymbol{\sigma} + \Delta \boldsymbol{\sigma}$$