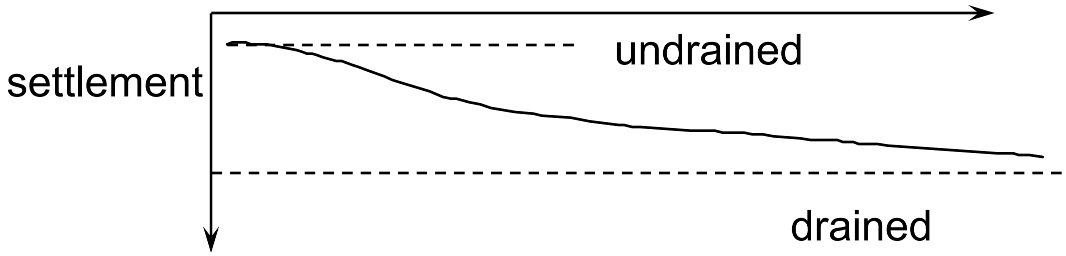
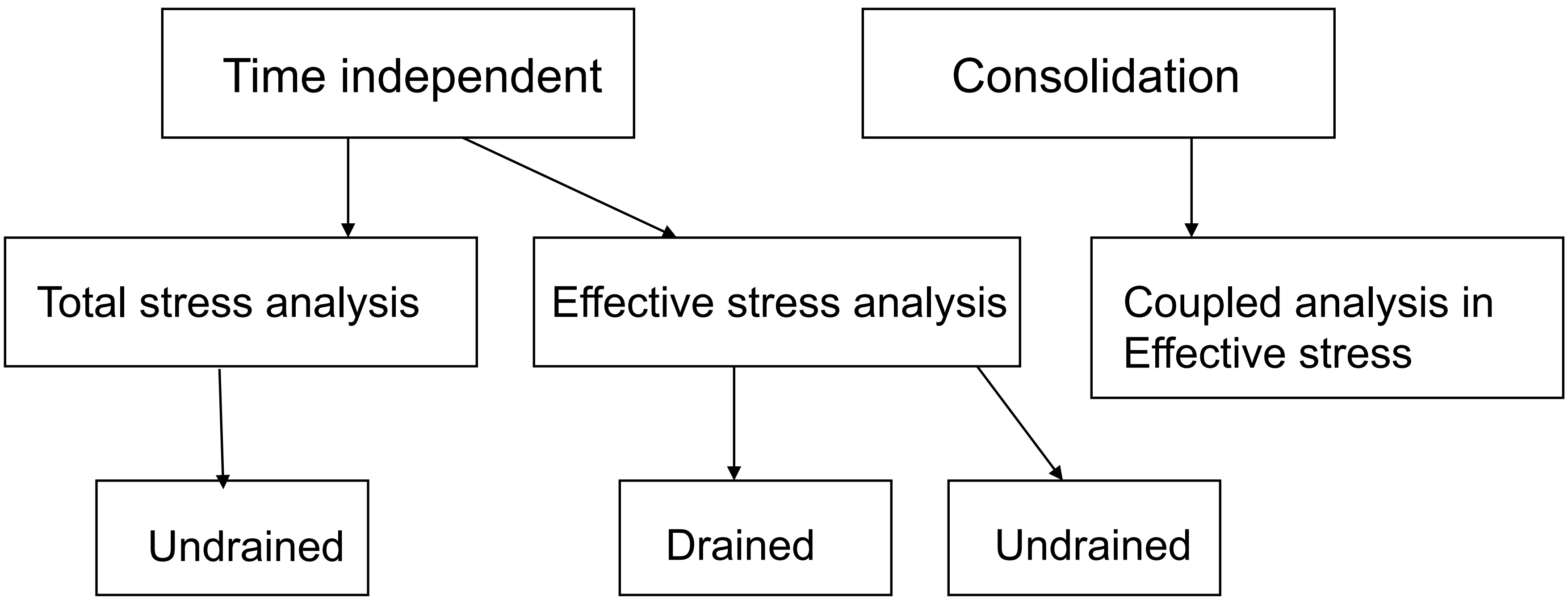
4 Steps to perform Finite Element Analysis

- **Define the problem**
  - Total stress analysis, effective stress analysis or consolidation analysis?
  - What are the unknowns?
  - Spatial dimension (plane strain, axisymmetric, 3D)
  - Element types
- **Create finite element mesh**
  - create nodes
  - connect nodes to create elements
- **Define analysis (or construction sequence) steps**
  - apply boundary condition
  - apply loading condition
- **Assign materials and element types to elements**
- **Define materials**
  - determine material properties
- **Define in-situ stress conditions**
- **Run the analysis**
- **Check!**

4.1 Types of analysis





(1) Drained analysis

- No excess pore pressure - high permeable soils
- All the loads will be transferred to the soil skeleton
- Long-term condition - mostly interested in displacements

(2) Undrained analysis - low permeable soils

- Loads will be carried by both soil skeleton and pore pressure
- No volume change
- Short-term condition - mostly interested in stresses- undrained failure of clays?

(3) Consolidation analysis

- Transition from undrained condition to drained condition.
- Check the movement of the system with time.
- Time consuming – but correct stress paths and this can be important when the soil behaves plastically (stress path dependent)
- Undrained analysis can be performed by making the time step small.

4.2 Drained analysis – Effective stress

- Need to assign initial effective stresses before the analysis.
- Can use any effective stress model.
  - Elastic model
  - Mohr-Coulomb/Drucker Prager models
  - Cam-clay models
- If plasticity models are used, need to update the effective stresses at each increment.

  σ' (new) = σ' (old) + D (soil skeleton) $d\varepsilon$
- Very common

4.3 Undrained Analysis - Total stress

- Excess pore pressure cannot be calculated. Effective stress state of the soil cannot be examined.
- Elastic model is commonly used for deformation
  – Use Undrained stiffness E_u
  – Poisson's ratio close to 0.5
  – The properties can vary with depth.
- Von-Mises model is used for modelling undrained shear strength of clays.
- Can assign different stiffness and strength at different depths explicitly by assigning different model parameters at different depths.

Effective versus Total stress simulations using elastic model in a finite element program

Effective Stress		Total Stress
E', ν'	$\longleftrightarrow$	E_u and ν_u

No volume change

$\nu u = 0.5$ ($K_u = E_u/(1-2\nu_u)/3 = E_u/0 =$ infinity)

Pore fluid cannot sustain shear stresses. Soil skeleton carries the shear stresses τ (or q)

$G' = G_u$, $G' = E'/(2(1+\nu'))$ and $G_u = E_u/(2(1+\nu_u))$

$E'/(2(1+\nu')) = E_u/(2(1+0.5))$

$E_u=1.5E'/(1+\nu')$

In finite element analysis, $\nu_u=0.5$ cannot be used. Use $\nu_u=0.49$ or 0.495. But be careful with mesh locking problem.

4.4 Undrained analysis – Effective Stress

- Need to assign initial effective stresses before the analysis.
- Can use any effective stress model, so the stiffness and strength variation with depth can be modelled implicitly with the one set of model paramters.
- The applied load is carried by the soil skeleton and pore water.
- The contribution of the bulk modulus of water needs to be added.

  $D = D_{(\text{soil skeleton})} + (1/n)\, D_{(\text{water})}$, where n is the porosity
- Effective stress increment can be computed by

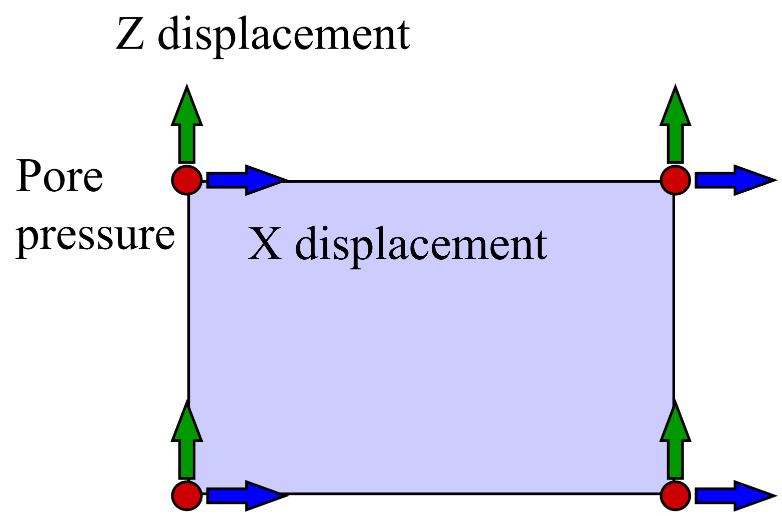
  $d\sigma' = D_{(\text{soil skeleton})}\, d\varepsilon$.
- Need to updated the effective stresses at each time step.

4.5 Consolidation Analysis – Effective Stress

- Uses Biot's three dimensional consolidation theory
- Pore pressure and displacement are computed at each time step.
- Need to use effective stress model
- Need permeability
- Lots of computating time.
- Most realistic. Undrained, partially drained, drained depending on the loading condition, drainage condition, permeability of soils
- Stress path followed is correct, which should provide good strain estimate when plasticity models are used.

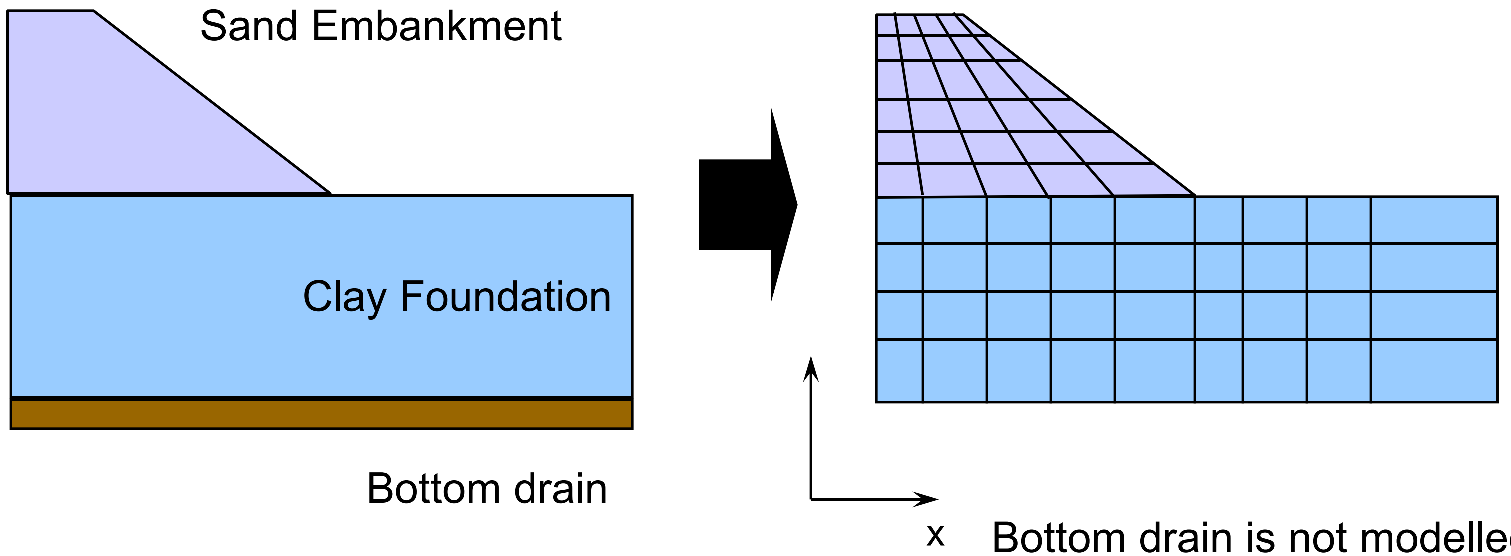
4.6 Problem Definition

- **Spatial dimension**
  - plane strain (x, y)
  - axisymmetric (r, z)
  - 3D (x,y,z)
- **Degrees of freedom**
  - x displacement
  - y displacement
  - z displacement
  - pore pressure
  - temperature
  - contaminant concentration



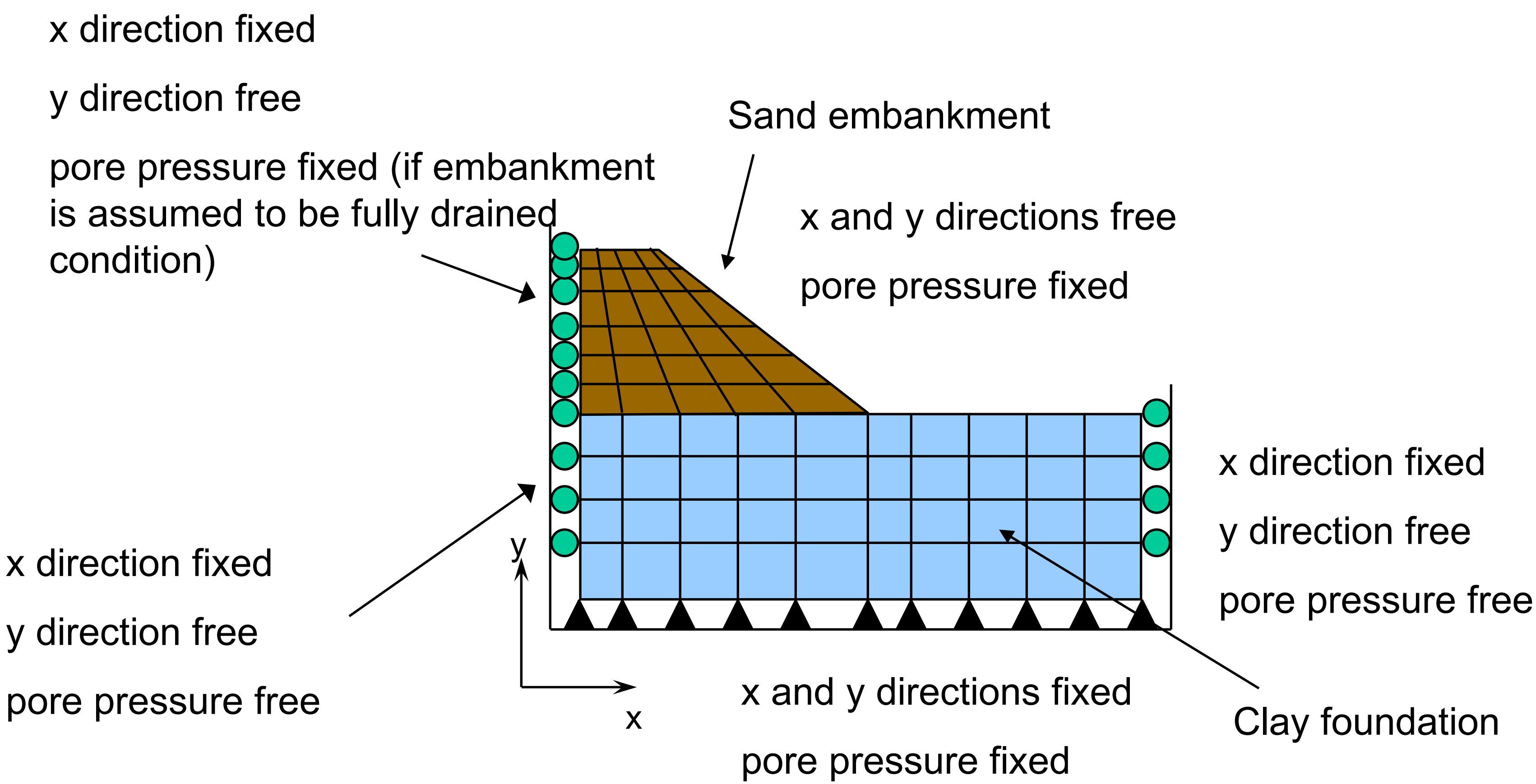
Descretization

- Create nodes
- Connect nodes to create elements



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

4.7 Boundary condition



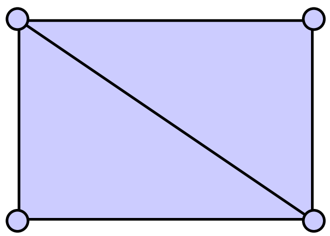
Pore pressure free = no water flow perpendicular to the boundary

4.8 Finite element types 1

- **Assign element types to elements**

2D plane strain/axisymmetric finite elements

Isoparametric Elements

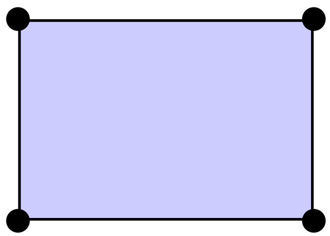


3 nodes element

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

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

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

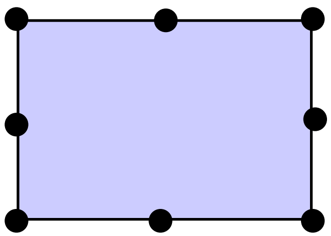


4 nodes element

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

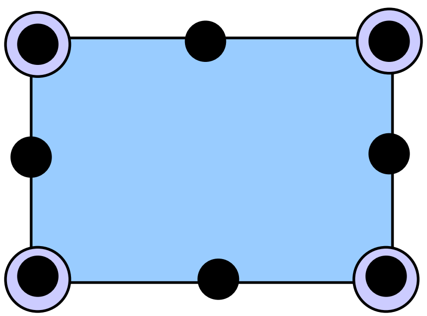
quadratic variation of displacement in both x and y directions.

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

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

Finite element types 2

2D Consolidation element

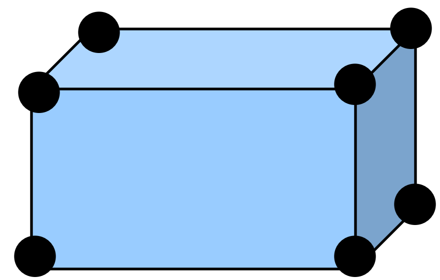


Pore pressure and displacements

Displacements

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

8 node 3D brick element



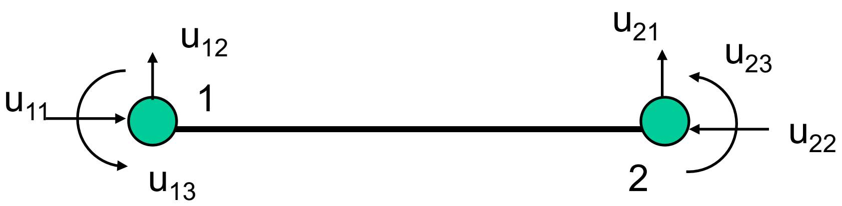
Linear variation of displacements in x, y and z directions

Finite element types 3

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

$F=EA\varepsilon_{bar}$

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. The axial force F_a, the shear force V and the moment at the centre of the beam can be calculated as follows.



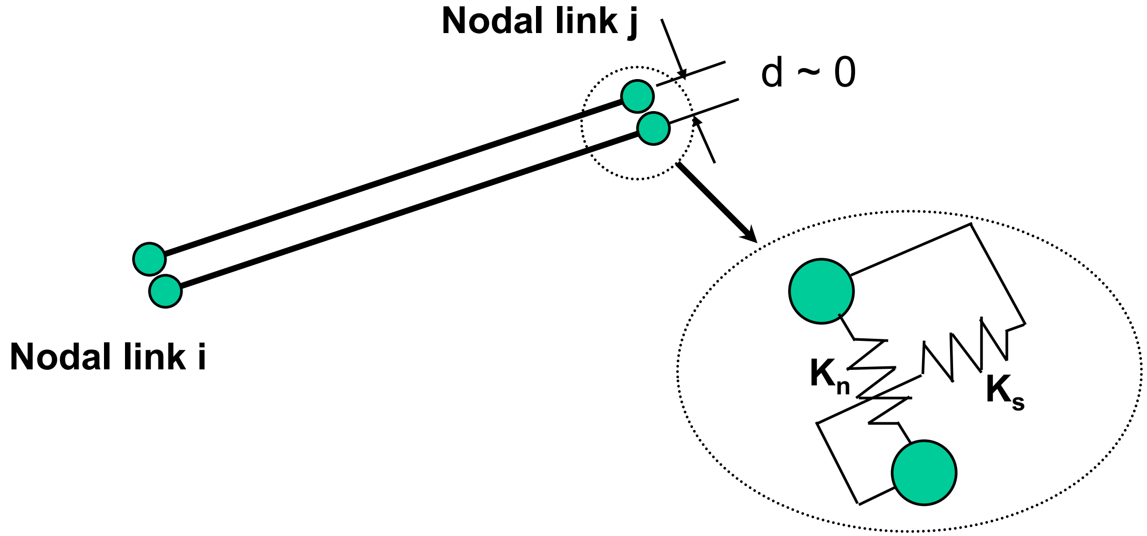
$F_a = EA\,\varepsilon_a$

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

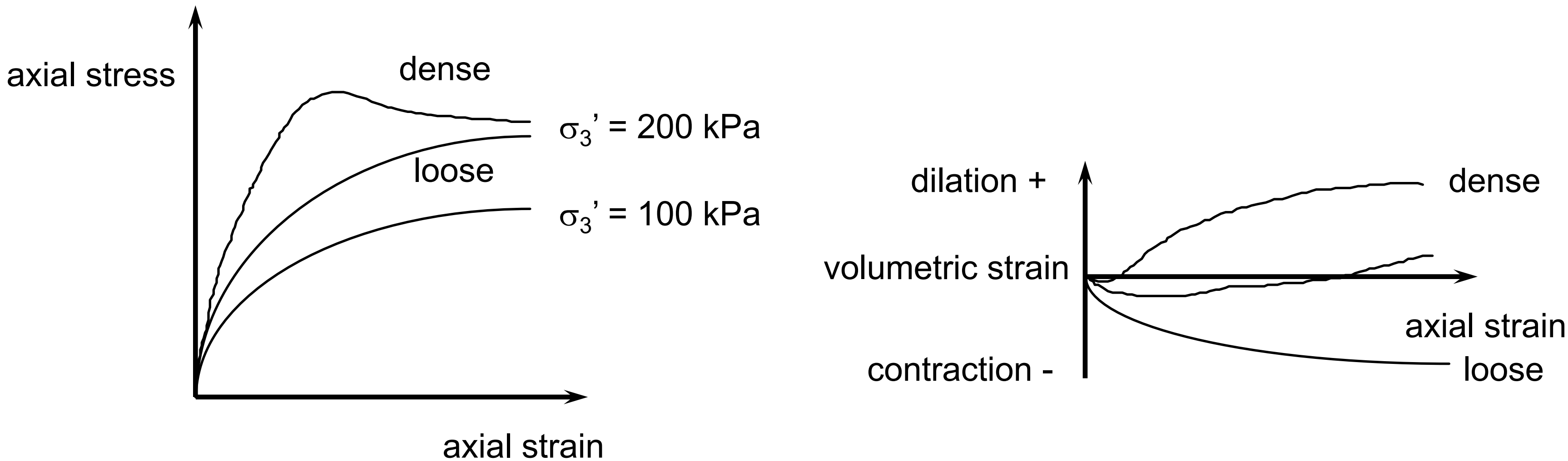
$M = EI(u_{13}-u_{23})/L$

Finite element types 4

Interface element This element allows relative displacement between elements. It is capable to model soil/structure interface conditions, shear planes with in 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 negligible.



4.9 Soil Models



(STEP 1) Decide soil models and input model parameters

Sand - Linear elastic model, Non-liner elastic model, Drucker-Prager elasto-plastic model, Mohr-coulomb elasto-plastic model or Advanced models ?

Clay - Linear elastic model, Non-linear elastic model, Cam-clay model, or Advanced models ?

(STEP 2) Assign soil models to elements

Embankment elements - sand model

Foundation elements - clay model

Verification of Design Parameters (after Atkinson, 1995)

- Are the soil models and analyses being used appropriate for the soils and for the structure?
- Are the design parameters being measured appropriate for the soil models being used?
- Are the tests being carried out the appropriate ones to determine the required design parameters?
- 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.
- 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 2

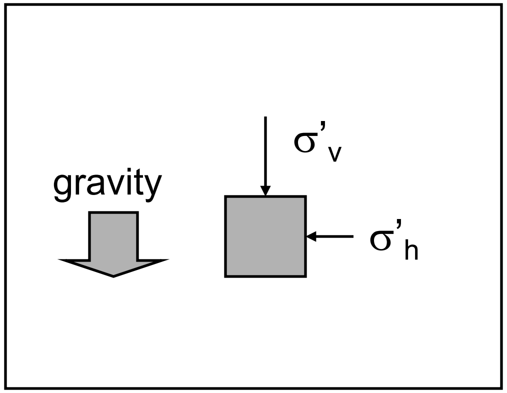
- Have any tests been done which investigate whether the soils reasonably follow the critical state theories or the elasto-plastic theories being used in the analyses?
- Have tests being carried out at stress levels corresponding to the range of stresses in the ground? Is it necessary to follow special stress paths representing the previous stress history and the current loading?
- Have linear parameters been fitted to non-linear test results over appropriate ranges?
- 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?

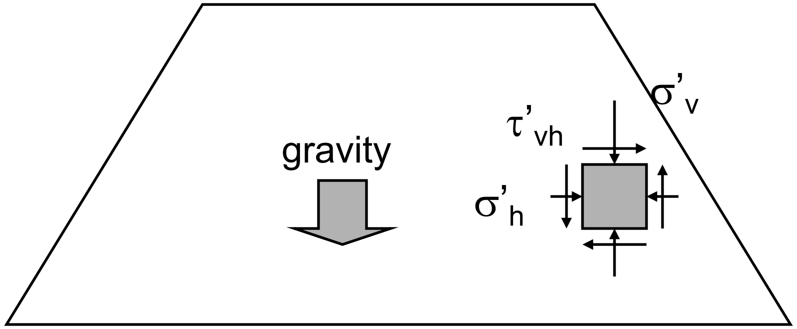
4.10 Geostatic stresses 1

Before conducting your analysis, you need to make sure that the stresses in the ground are the correct values. - Because soil behaviour 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. (like CRISP)

Horizontally levelled ground





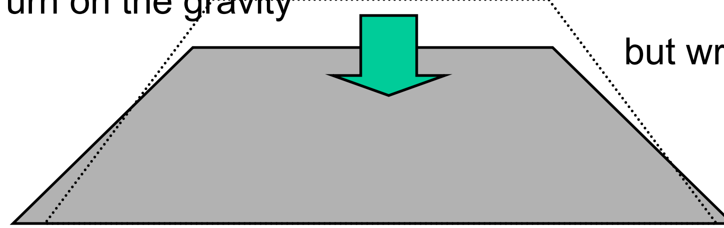
- Vertical effective stress σ'_v from the unit weight of the soil
- Horizontal effective stress $\sigma'_h = K_0\, \sigma'_v$

- Estimation of in-situ stresses becomes difficult - depends on soil model used.

Geostatic stresses 2

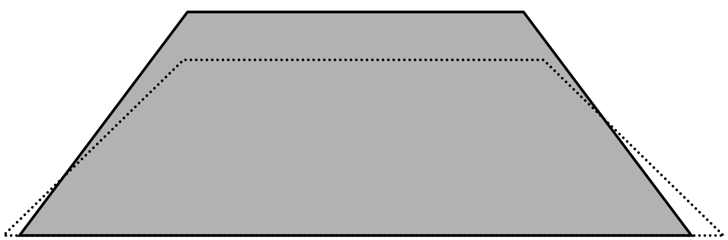
(2) **Zero displacement approach** - ask the program to compute the in-situ stresses from the equilibrium condition (very few programs allow you to do this)

(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

- Cam-clay predictions (Wood, 1990).

K_0 (Modified CC) < K_0 (Original CC)

- $K_0 = K_{nc} = 1 - \sin\phi'$ (Jaky, 1944) (ϕ' is the friction angle of the soil).

Overconsolidated soils

- Cam-clay predictions

- $K_{oc} = (1 - \sin\phi') \times (OCR)^{\sin\phi'}$ (Mayne et al., 1982) up to $K_p = (1 + \sin\phi')/(1 - \sin\phi')$

- Wroth's method (1975)

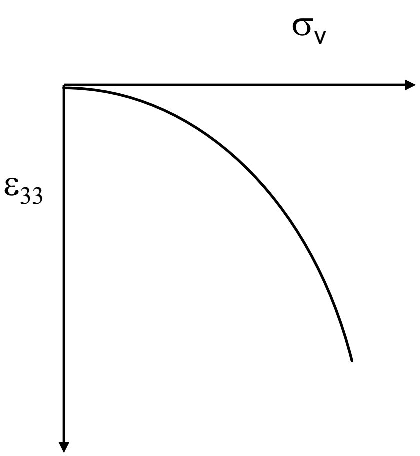
$K_{oc} = OCR\ K_{nc} - (\nu'/(1 - \nu'))(OCR - 1)$ for OCR<5

where ν' is the poisson's ratio = 0.254 - 0.371.

$$m\left[\frac{3(1-K_{nc})}{1+K_{nc}} - \frac{3(1-K_{oc})}{1+2K_{oc}}\right] = \ln\left[\frac{OCR(1+2K_{nc})}{1+2K_{oc}}\right] \quad \text{for large OCR}$$

where m = 0.022875PI + 1.22 (PI = Plasticity index)

One Dimensional Compression Test



Data (σ_v-ε_{11})

Using an isotropic elastic model, find correlation between K_0 and ν

$$\varepsilon_{11} = \varepsilon_{22} = 0$$

$$\sigma'_v = \sigma'_{33} = \frac{E(1-\nu)}{(1+\nu)(1-2\nu)}\varepsilon_{33}$$

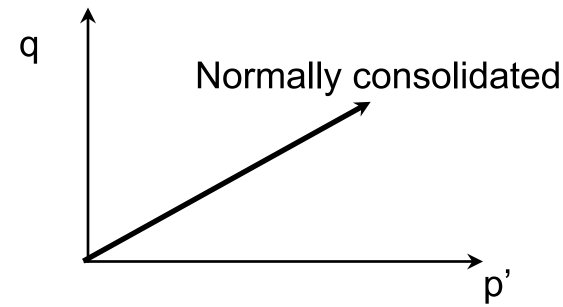
$$\sigma'_h = \sigma'_{11}(or\,\sigma'_{22}) = \frac{E\nu}{(1+\nu)(1-2\nu)}\varepsilon_{33}$$

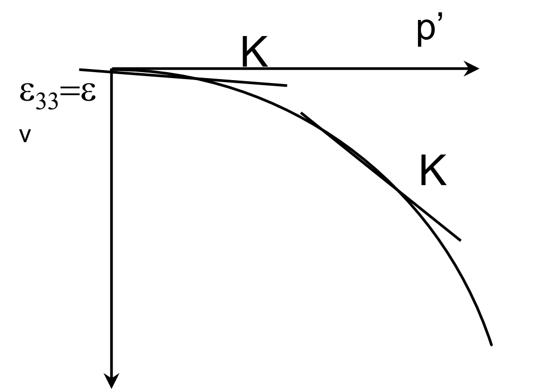
$$K_0 = \frac{\sigma'_h}{\sigma'_v} = \frac{\nu}{1-\nu}$$

Ko = 0.5 then ν = 1/3

$p' = (\sigma'_v+2\sigma'_h)/3 = (1+2K_0)\sigma'_v/3$

$q=\sigma_v-\sigma_h = (1-K_0)\sigma_v'$



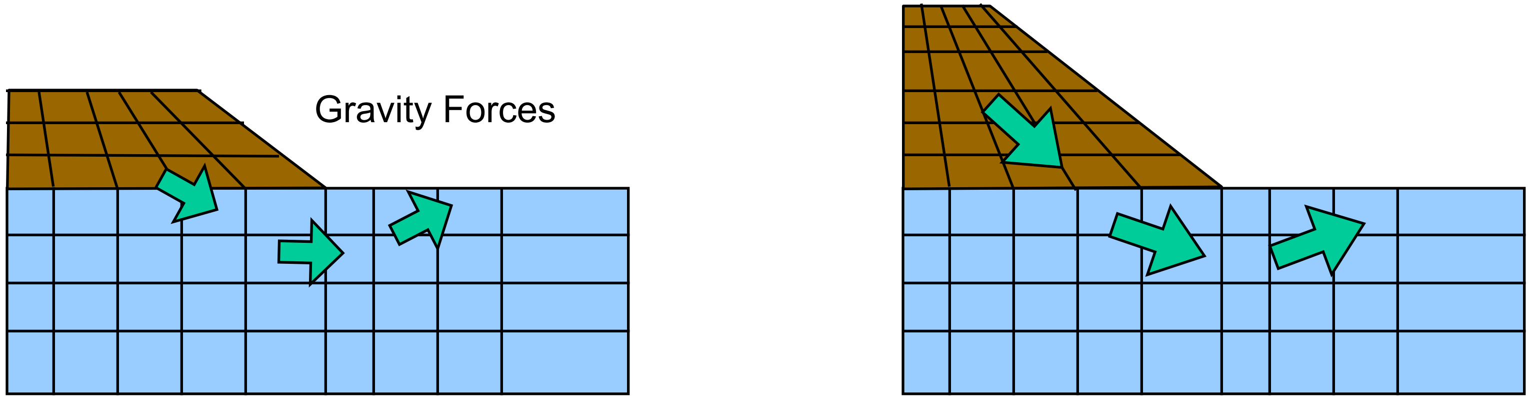


$\varepsilon_v = \varepsilon_{33} + 2\varepsilon_{11} = \varepsilon_{33}$

By assuming K_0, K and G can be determined.

4.11 Loading conditions 1

- Body force
  – apply unit weight
- Force
  – node free
- Displacement
  – node fixed
- Pore pressure
  – node fixed to apply external pore pressures

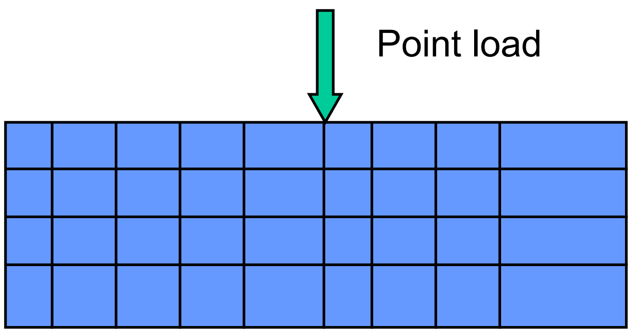


Construction sequence

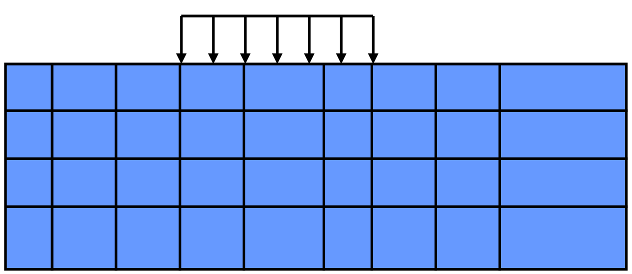
Loading conditions 2

External forces

Point load

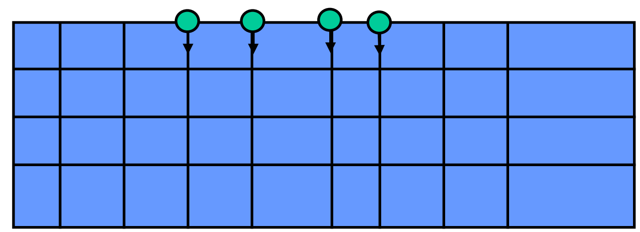


Distributed load

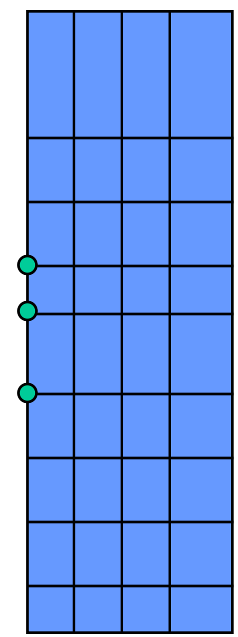


Apply displacements

Rigid foundation



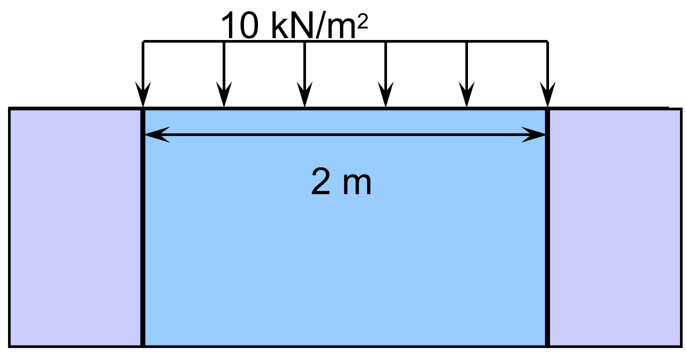
Excess pore pressure

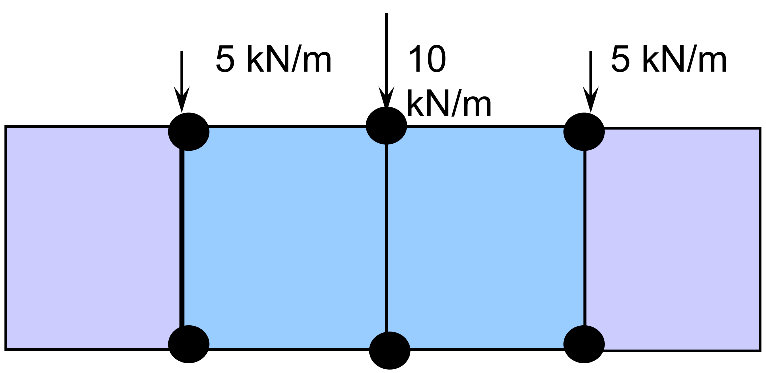


Water pumping

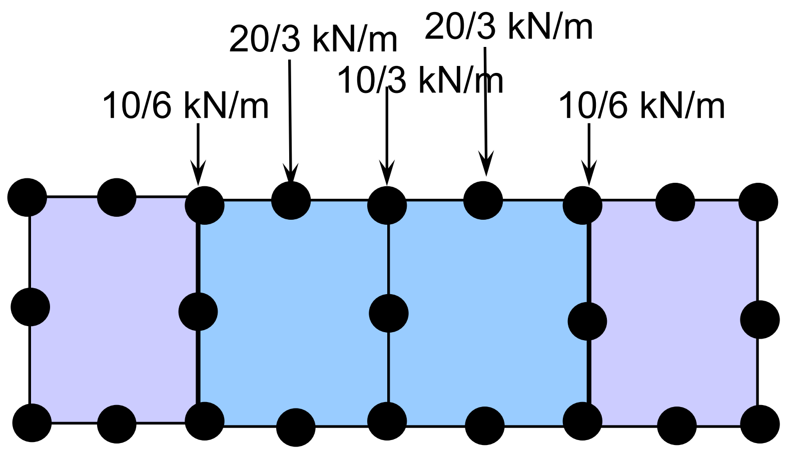
Loading conditions 3

Distributed load -> Nodal forces in plane strain condition





(1) 4 nodes elements



(2) 8 nodes element

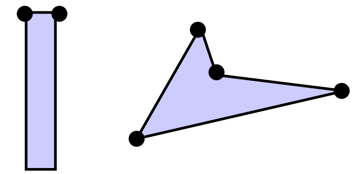
Some FE programs do this automatically

4.12 Application of FEM 1

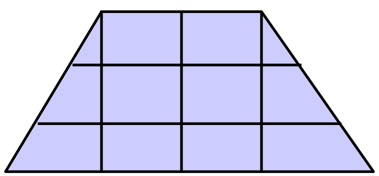
(1) Use big graph paper / draw in boundaries

(2) Keep elements as "Square" as possible.

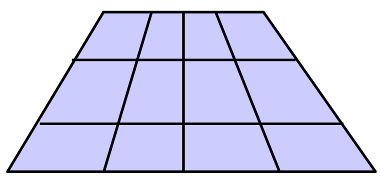
Do not want :



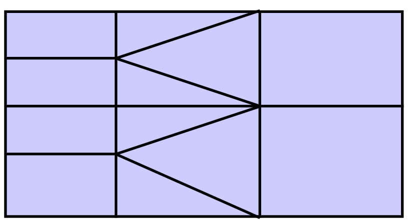
(3) For 4 node elements, try to avoid triangular



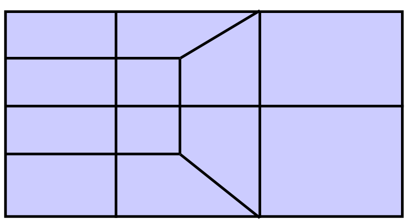
Not recommended



Better

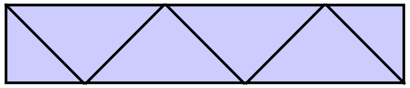


Not recommended

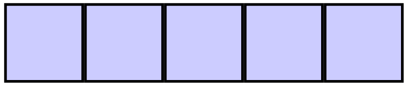


Better

(4) Use regular mesh



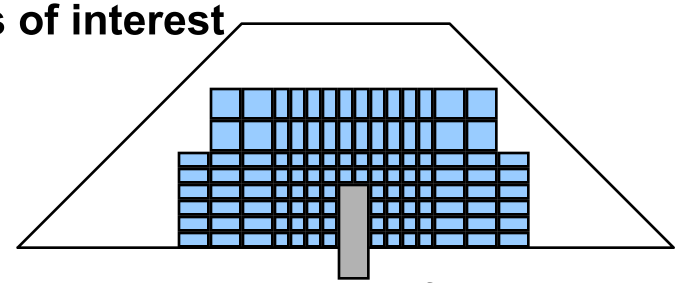
Not recommended



Better

Application of FEM 2

(5) Use refined mesh in regions of interest



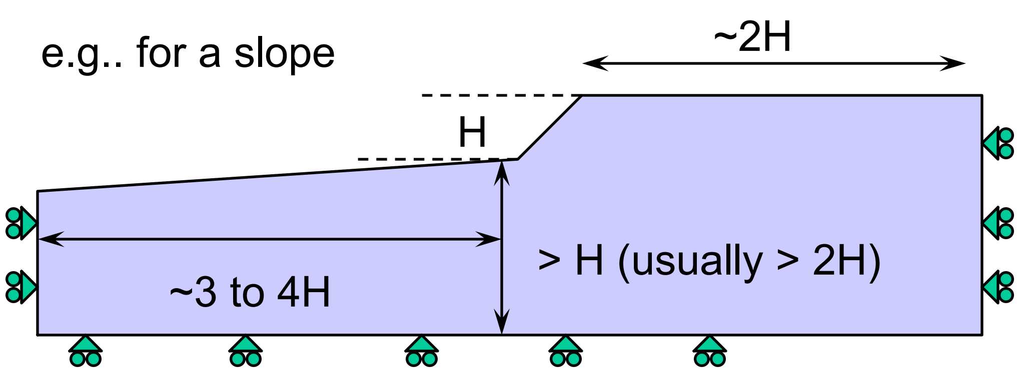
finer around cut-off wall, and coarse elsewhere

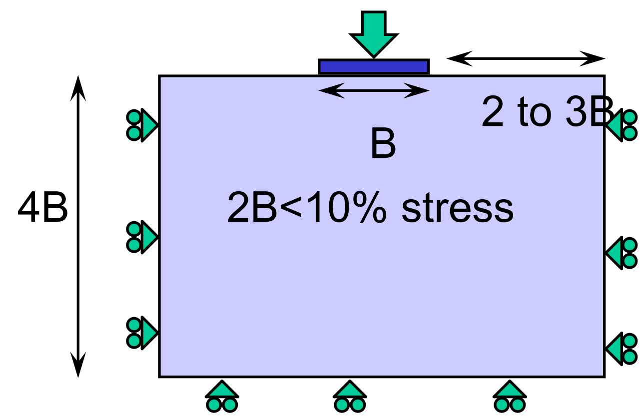
(6) Avoid large jumps in element size

size jump <3

(7) Model boundary conditions properly - Define bottom and side boundaries

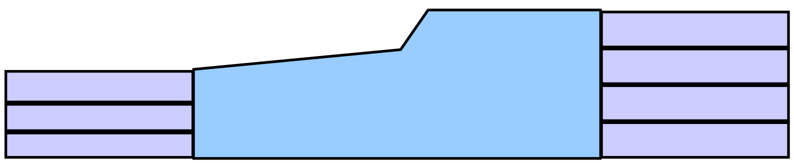
e.g.. for a slope





Application of FEM 3

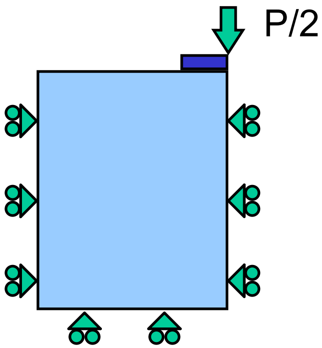
If you get too much vertical deflection at rollers, side boundaries probably too close.



Break rules, but since shear stress&strains are small, small error

better bad elements than close boundaries

(8) Can also use symmetry



(9) Avoid using equal size elements if the solution is oscillations - typical problem for undrained analysis

(10) Try to keep the number of nodes and elements down, but always check the mesh size sensitivity of the problem

non-critical structure (e.g. culvert) 150~200 4-node elements

dam without foundation 200~300 4-node elements

dam with deep foundation 300~400 4-node elements

Application of FEM 4

(11) Built mesh like it's being built in reality.

(12) Always check your answer

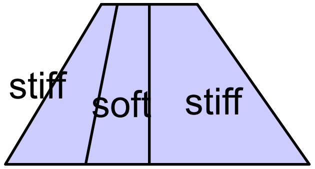
(a) Use well documented field study or model study

(b) Do hand calculations or use chart solution

(c) Check orientations of the maximum principal stress directions − tells how loads flowing through the system

(d) Check failure parameters-how close to failure line, look to see whether reasonable

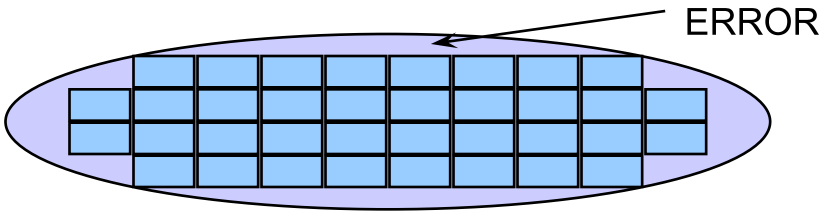
(e) check the vertical stress σv and σh



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

4.13 Errors in FEM

(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?

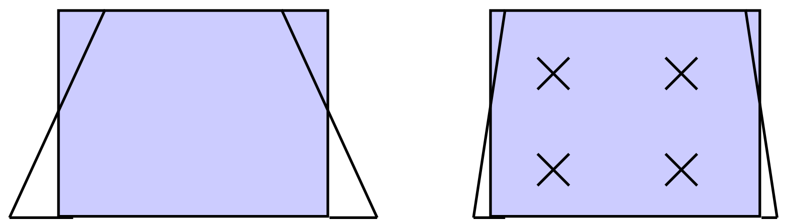
More sophisticated models?

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

Volume Mesh Locking

Conventional FE formulations produces very stiff behaviour of a finite element when modelling (nearly) incompressible materials. The volume at each integration point is fixed and this condition puts severe constraints on the kinematically admissible displacement fields at the nodes. This stiff behaviour is called "**mesh locking**". For soils, incompressible condition (no volume change) occurs when it is in undrained condition.

No volume change, only shear deformation



reality **mesh locking**

How to avoid mesh locking

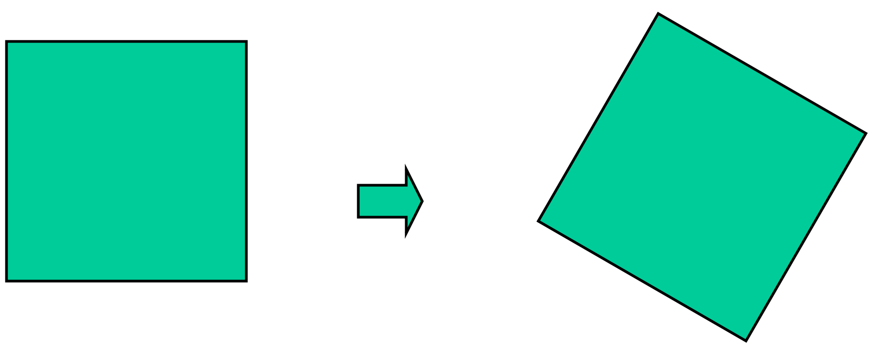
- **Use reduced integration method to "soften" the element**
  - 4 node isoparametric element uses 4 integration points, but one point in the middle in reduced integration.
  - The solution may be affected by the mesh size and mesh instability. Need to investigate there are no problems before the analysis.
  - This method may not work with second order elements (8 or 9 node 2D elements).
- **Use selectively reduced integration method**
  - Volumetric and deviatoric strains are decomposed and different integration schemes are used.
  - Some commercial programs have this capability.

2.4 Solving Non-linear problem
2.4.1 Linear and Nonlinear Problem

- Linear problem
  - The response can only be approximated as linear if its deformations/motions are small.
  - In linear analyses, the response to individual load cases can be scaled and added to the results from other linear analyses, which is the principal of superposition.
- Non-linear problem
  - Superposition is invalid.
  - The solution is an incremental/iterative process.
  - An iteration is the solution of a system of equations linearised about the current state of the nonlinear physical problem.

2.4.2 Non-linearity in Geotechnical Engineering

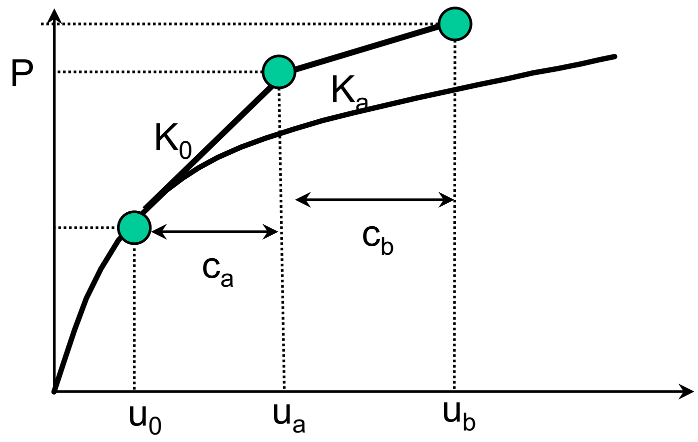
- Material non-linearity
  - plasticity
- Contact
  - discontinuous source of nonlinearity
- Large deformations and motions of a geotechnical structure
  - rotation, rigid body motion
  - often ignored, still in the research area.



No strains if linear strain-displacement relations are used.

2.4.3 Tangent stiffness method

- A piece-wise linear approximation to the non-linear problem



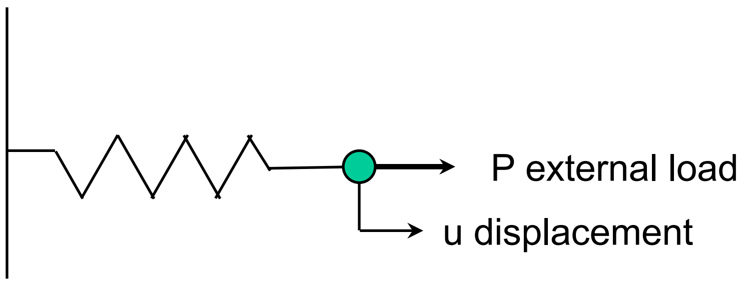
Many small increments are need to obtain accurate solution

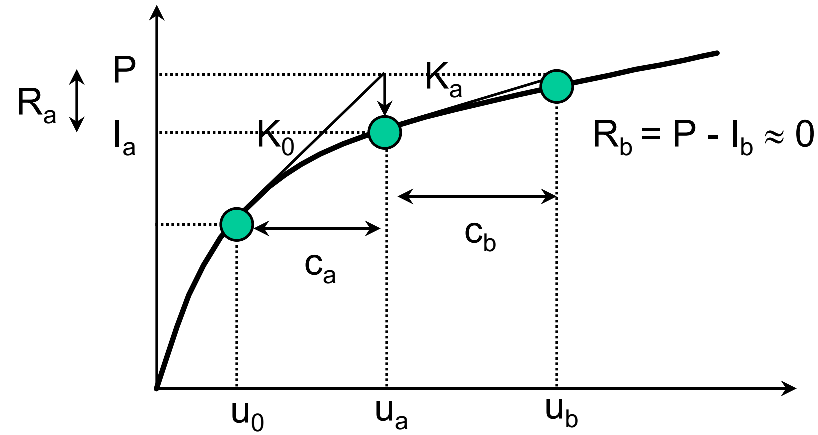
Need to perform a parametric study to find the optimum incremental

Defining increments is very important

Program - SAGE CRISP

2.4.4 Newton-Raphson technique 1





Program - ABAQUS

- Using the initial stiffness K_0, apply an increment of load DP, calculate an approximate solution c_a caused by this increment.
- The stiffness K_a is updated using the new position, and the internal force in the spring I_a is calculated.
- If the difference R_a between the total load applied to the spring, P, and I_a is smaller than the tolerance, $u_a = u_0 + c_a$ is the converged solution.

Newton-Raphson technique 2

- If R_a is not small, a new displacement correction c_b is calculated by solving $c_b = R_a/K_a$
- The new displacement u_b is updated, and the internal force I_b in the updated configuration is calculated.
- The new force residual R_b is obtained. If R_b< tolerance, the solution is converged. If not, continue the iteration.
- **Tolerance R**
  - must be small enough to ensure that the approximate solution is close to the exact mathematical solution.
  - Must be large enough so that reasonable number of iterations are performed.
- **Quadratic convergence**
  - If the tangent stiffness is calculated correctly, R should reduce quadratically from one iteration to the next.

Note : The Newton-Raphson method is the most standard method to solve nonlinear problems in FE. However, there are other methods available to solve nonlinearity, especially for softening problem. See Crisfield (vol2) for example.

2.4.5 Comparison