DEPARTMENT OF CIVIL & ENVIRONMENTAL ENGINEERING

FEM with Material Nonlinearity

Yu Zhijie

20454611

1 NONLINEAR PROBLEMS

In the previous study of finite element method, linear assumption is usually used to simplify problems.However, linearity is only an approximation. In reality, most of the problems involve different kinds of nonlinearity which can not be ignored during analysis.
In general, there are four common kinds of nonlinearity. Geometric nonlinearity involves nonlinear strain-displacement relation. Material nonlinearity involves nonlinear constitutive relation. Kinematic nonlinearity involves non-constant displacement boundary conditions or contacts.And force nonlinearity involves follow-up loads.
In practical problems usually more than one nonlinearity can exist at the same time. In this project we mainly focus on material nonlinearity, which is especially important in geotechnical engineering.
In order to describe the highly nonlinear characteristics of geotechnical materials like soil or rock, many kinds of constitutive models are proposed. When solving geomechanical problems with FEM, the key point is to involve these nonlinear constitutive models in the FEM framework which will be discussed in the following sections.

2 EQUATIONS FOR MATERIAL NONLINEARITY FEM

Weak form of the governing equation

$$\iiint_{\Omega} \boldsymbol{\varepsilon}(\boldsymbol{u})^T \boldsymbol{\sigma} d\Omega = \iint_{\Gamma_s} \boldsymbol{u}^T \boldsymbol{t} d\Gamma + \iiint_{\Omega} \boldsymbol{u}^T \boldsymbol{f}^b d\Omega \tag{2.1}$$

where $\boldsymbol{u}$ is the displacement tensor at any point,$\boldsymbol{\varepsilon}(\boldsymbol{u})$ is the corresponding strain tensor, $\boldsymbol{\sigma}$ is the stress tensor at any point, Γ_s is the natural boundary and Ω is the whole domain. Thus

$\iint_{\Gamma_s} \boldsymbol{u}^T \boldsymbol{t} d\Gamma$ represents boundary force and $\iiint_{\Omega} \boldsymbol{u}^T \boldsymbol{f}^b d\Omega$ represents body force. This governing equation is derived from principle of virtual work and is essentially an equilibrium equation as in the linear FEM.
Based on shape functions, the above terms can be expressed with nodal displacements $\boldsymbol{d}$.

$$\boldsymbol{u} = \boldsymbol{N}\boldsymbol{d} \tag{2.2}$$

$$\boldsymbol{\varepsilon} = \boldsymbol{B}\boldsymbol{d} \tag{2.3}$$

where $\boldsymbol{N}$ is the shape function matrix and $\boldsymbol{B}$ is the gradient of the shape function matrix.
Then the discretization form is as follows.

$$\iiint_{\Omega} \boldsymbol{B}^T \boldsymbol{\sigma} d\Omega = \iint_{\Gamma_s} \boldsymbol{N}^T \boldsymbol{t} d\Gamma + \iiint_{\Omega} \boldsymbol{N}^T \boldsymbol{f}^b d\Omega \tag{2.4}$$

For linear FEM, $\boldsymbol{\sigma} = \boldsymbol{D}\boldsymbol{\varepsilon} = \boldsymbol{D}\boldsymbol{B}\boldsymbol{d}$ and this equation is a linear system which can be solved directly with boundary conditions. However for a material nonlinear FEM, σ cannot be explicitly expressed and Newton-Raphson method needs to be adopted to solve this equation. Newton-Raphson(NR) method is a kind of numerical method to find roots of a real-valued function such as $P(d) = F$.The basic idea is shown in 2.1.
The iteration process is as follows.

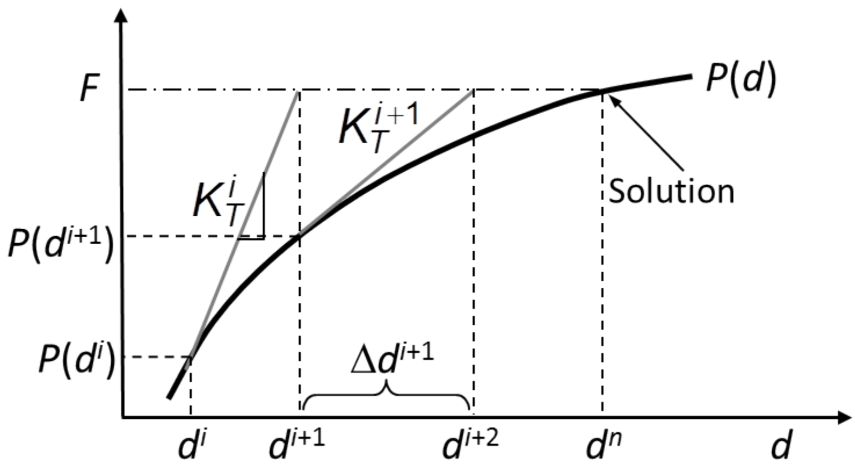

Figure 2.1: Newton-Raphson method sketch

- We assume $\boldsymbol{d}^i$ at i-th iteration is known.
- Solve $\boldsymbol{K}_T^i \Delta \boldsymbol{d}^i = \boldsymbol{R}^{\boldsymbol{i}} = \boldsymbol{F} - \boldsymbol{P}(\boldsymbol{d}^i)$ to find the increment $\Delta \boldsymbol{d}^i$.
- Update solution $\boldsymbol{d}^{i+1} = \boldsymbol{d}^i + \Delta \boldsymbol{d}^i$.

When $\Delta \boldsymbol{d}^i$ is smaller than the given threshold, this process converges and we can get the approximate solution.

For the above discretized equation, the residual $\boldsymbol{R}$ can be expressed as $\iint_{\Gamma_s} \boldsymbol{N}^T \boldsymbol{t} d\Gamma + \iiint_{\Omega} \boldsymbol{N}^T \boldsymbol{f}^b d\Omega - \iiint_{\Omega} \boldsymbol{B}^T \boldsymbol{\sigma} d\Omega$. Thus we have

$$\boldsymbol{K}_T \Delta \boldsymbol{d} = \boldsymbol{R} = \iint_{\Gamma_s} \boldsymbol{N}^T \boldsymbol{t} d\Gamma + \iiint_{\Omega} \boldsymbol{N}^T \boldsymbol{f}^b d\Omega - \iiint_{\Omega} \boldsymbol{B}^T \boldsymbol{\sigma} d\Omega \tag{2.5}$$

$$\boldsymbol{K}_T = \frac{\partial \iiint_{\Omega} \boldsymbol{B}^T \boldsymbol{\sigma} d\Omega}{\partial \boldsymbol{d}} \tag{2.6}$$

$\boldsymbol{K}_T$ is called tangent stiffness matrix in FEM. By solving this equation, we can get the nodal displacement and thus all the other useful values.

3 Application

3.1 1D isotropic hardening model

In this section we will use this nonlinear method to solve a 1D problem with isotropic hardening model.
Isotropic hardening model is a kind of elastoplastic model. The 1D sketch of this model is shown in 3.1.
This isotropic hardening model has following characteristics.

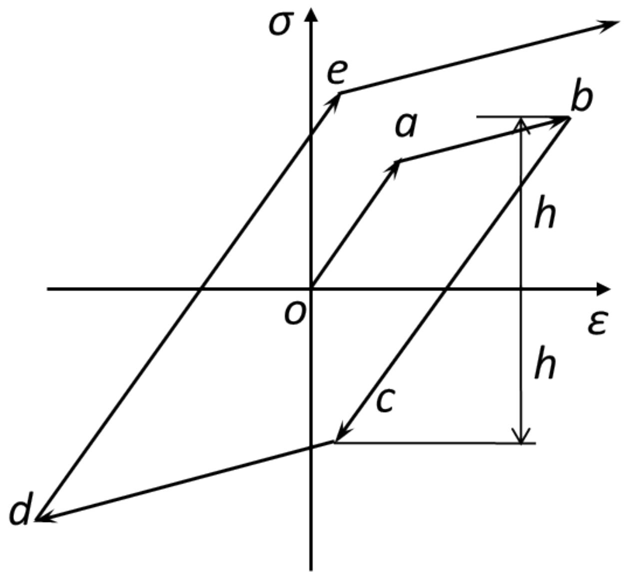

Figure 3.1: Isotropic hardening model sketch

- Initial elastic behavior with slope $\boldsymbol{E}$ (elastic modulus) until yield stress $\boldsymbol{\sigma}_y$ (line o-a).
- After yielding, the plastic phase with slope $\boldsymbol{E}_t$ (tangent modulus) (line a-b).
- Upon removing load, elastic unloading with slope $\boldsymbol{E}$ (line b-c).

- Loading in the opposite direction, the material will eventually yield in that direction (point c).
- Elastic range (yield stress) increases proportional to plastic strain.
- The yield stress for the reversed loading is equal to the previous yield stress.
- Plastic strain can be used as an evolution variable to depict the material state.
- The basic material parameters for this model involve elastic modulus $\boldsymbol{E}$, plastic modulus $\boldsymbol{H}$ and initial yield stress $\boldsymbol{\sigma}_y^0$.

To solve a 1D problem with isotropic hardening model like in 3.2, the key points lie in the determination of the current state (whether elastic or plastic) and the decomposition of strain increment (elastic strain increment and plastic strain increment).
The solution procedure can be described as follows. For each load step, $\boldsymbol{E}$, $\boldsymbol{H}$, $\boldsymbol{\sigma_y^0}$, $\boldsymbol{\sigma}^n$(current

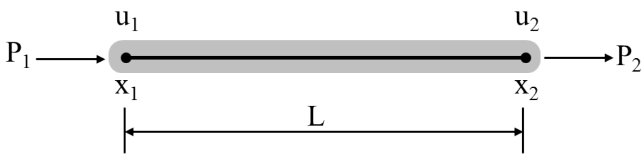

Figure 3.2: 1D bar element

yield stress), $\boldsymbol{\varepsilon}_p^n$(current plastic strain) and $\Delta\boldsymbol{\varepsilon}$(strain increment of this step) are given.

- Trial state(assume purely elastic).

$$\boldsymbol{\sigma}^{tr} = \boldsymbol{\sigma}^n + \boldsymbol{E}\Delta\boldsymbol{\varepsilon}$$
$$\boldsymbol{\sigma_y^n} = \boldsymbol{\sigma}_y^0 + \boldsymbol{H}\boldsymbol{\varepsilon}_p^n$$
$$\boldsymbol{f}^{tr} = |\boldsymbol{\sigma}^{tr}| - \boldsymbol{\sigma}_y^n$$

- If $\boldsymbol{f}^{tr} \leq 0$, this step is purely elastic.Thus we have $\boldsymbol{\sigma}^{n+1} = \boldsymbol{\sigma}^{tr}, \boldsymbol{\varepsilon}_p^{n+1} = \boldsymbol{\varepsilon}_p^n$. Then this step is finished.
- If $\boldsymbol{f}^{tr} > 0$, this step involves plastic part.

$$\Delta\boldsymbol{\varepsilon}_p = \frac{\boldsymbol{f}^{tr}}{\boldsymbol{E}+\boldsymbol{H}}$$
$$\boldsymbol{\sigma}^{n+1} = \boldsymbol{\sigma}^{tr} - sgn(\boldsymbol{\sigma}^{tr})\boldsymbol{E}\Delta\boldsymbol{\varepsilon}_p$$
$$\boldsymbol{\varepsilon}_p^{n+1} = \boldsymbol{\varepsilon}_p^n + \Delta\boldsymbol{\varepsilon}_p$$

For the 1D bar element problem, $\boldsymbol{E} = 100GPa$, $\boldsymbol{H} = 25GPa$, $\boldsymbol{\sigma_y^0} = 250MPa$. This element will undergo cyclic loading unloading and the final stress strain curve is shown in 3.3.
This 1D problem doesn't involve N-R iteration as this problem is too simple. Tangent stiffness is constant and many relations can be derived based on geometric features. The more complicated 3D problem is discussed in the next section.

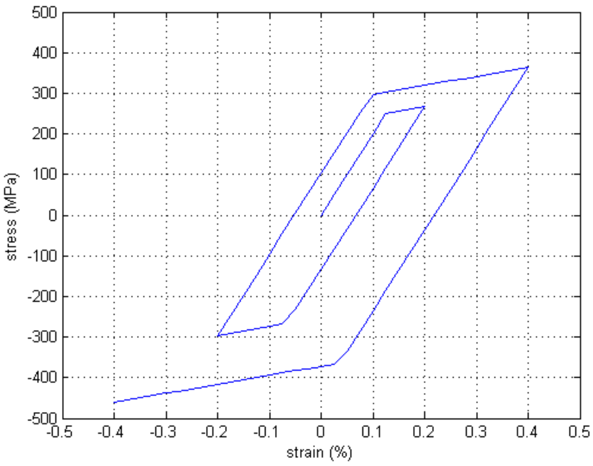

Figure 3.3: Stress-strain curve

3.2 3D ISOTROPIC HARDENING MODEL

For 3D problem with isotropic hardening model, more material parameters are needed. λ and μ are needed to assemble tangent operator rather than merely elastic modulus E. Plastic modulus H and initial yield stress σ_y^0 are also needed.

The whole framework is shown in 3.4. This illustrates one iteration of N-R method for a single

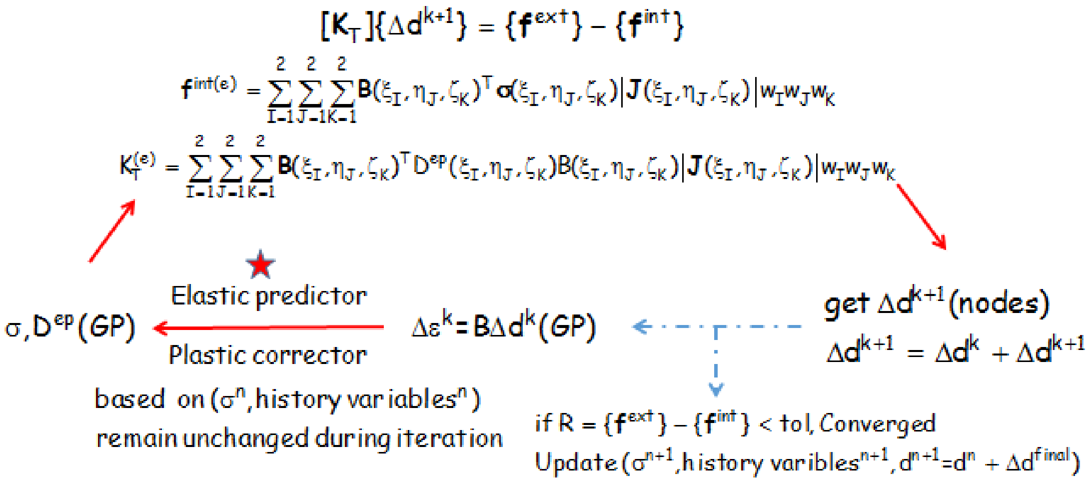

Figure 3.4: Nonlinear framework

load step. Given nodal displacement increments Δd^k as the starting point, we first calculate the strain increment at each Gauss point. By using isotropic hardening model, we can then compute σ and tangent operator D^{ep} at each Gauss point. In this process, we first assume this procedure to be elastic and then check whether the yield surface has been reached. If so, return mapping algorithm are used to correct the stress and the corresponding history variables. Based on these values, the tangent stiffness matrix and internal force vector at each Gauss point can be obtained, which can be further assembled to get global tangent stiffness matrix and residual force vector. We then solve the discretized system to get new nodal displacement increment Δ^{k+1}. If the residual force is smaller than the given threshold, this problem converges and we can move to the next load step. If not, another iteration is needed.

We use this nonlinear FEM framework to solve the following problem in Figure 3.5. The material parameters are shown in Table 3.1. The nodal force F_i at the top increases from 0 to 11 kN.

With a matlab code, this problem has been solved and the stress and displacement of node 12 at x_3 direction are shown in Figure 3.6. In this curve we can get that when stress rises to 400 MPa, this point reaches the yield stress and goes into plastic part which is consistent with σ_y^0=400 MPa.

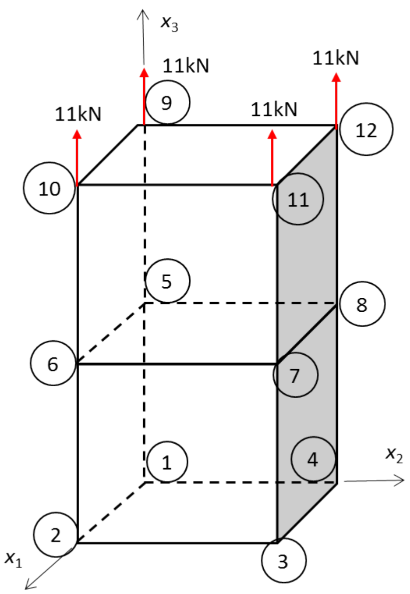

Figure 3.5: 3D hexahedron elements

Table 3.1: Material parameters for 3D isotropic hardening model

λ	μ	σ_y^0	H
110.7 GPa	80.2 GPa	400 MPa	100MPa

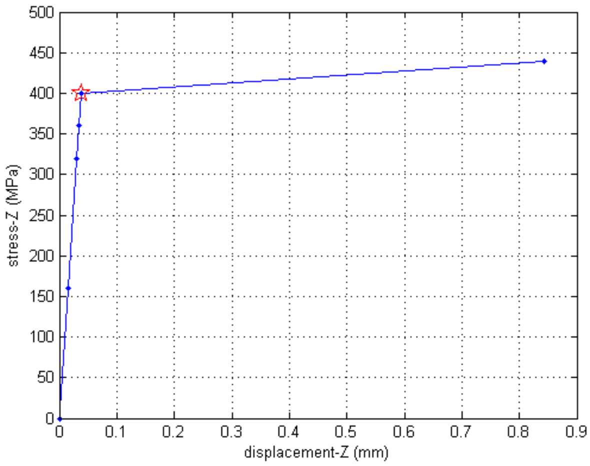

Figure 3.6: Stress-displacement cure of Node 12

4 ACKNOWLEDGEMENT

I have to admit that this course seems really a nightmare to me. Meanwhile, it really reveals to me the beauty of FEM which for me is merely a bland tool before this course. The most important ideas I learn from this course are approximation and discretization. The magic of discretizing complex systems into small domains and using approximation to solve their details really fascinates me. Thanks Prof.Ilias for your kind help and hard push.