Nonlinear Finite Element Methods
Assignment for summer term 2020 (Examiner: Geralf Hutter)

Computational Material science
Name:Sai Karthikeya Vemuri
Immatrikulation Number: 65124

Contents

List of Figures

1 Task

The problem of creep of a thick-walled pipe under internal pressure p is considered as sketched in Figure 1. The pressure rises linearly up to its final value p_{max} and is then hold until t_f as shown in Figure 2. Plain strain σ_{zz} = 0 conditions are assumed.

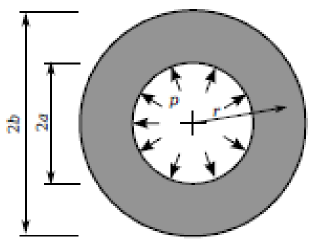

Figure 1: Thick-walled pipe

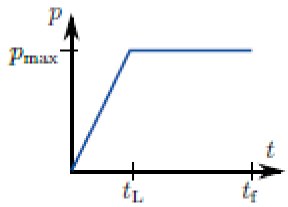

Figure 2: Load sequence

2 Overview over the implemented theory

Due to axisymmetric conditions, the only non-vanishing equilibrium condition is

$$0 = \frac{\partial \left(r \sigma_{rr} \right)}{\partial r} - \sigma_{\phi\phi} \tag{1}$$

Therein, σ_{rr} and $\sigma_{\Phi\Phi}$ refer to the stress components with respect to a polar coordinate system. The weak form of Eq.(1) reads

$$0 = \delta W = \int_a^b \underline{\delta \varepsilon}^{\mathrm{T}} \cdot \underline{\sigma} r \mathrm{d}r - \left[r \sigma_{rr} \delta u_r \right]_{r=a}^b \tag{2}$$

with stresses and strains written in Voigt notation as

$$\underline{\sigma} = \begin{bmatrix} \sigma_{rr} \\ \sigma_{\phi\phi} \end{bmatrix}, \quad \underline{\delta e} = \begin{bmatrix} \delta \varepsilon_{rr} = \frac{asu_r}{p_r} \\ \delta \varepsilon_{\phi\phi} = \frac{\delta u_r}{r} \end{bmatrix}, \quad \text{and analogously } \underline{\varepsilon} = \begin{bmatrix} \varepsilon_{rr} = \frac{(\Delta_1 x}{\partial r} \\ \varepsilon_{\phi\phi} = \frac{u_r}{r} \end{bmatrix} \tag{3}$$

Therein, the only non-vanishing displacement component is u_r(r) as the displacement in radial direction. The boundary conditions for the problem in Figure 1 are σ_{rr} = a) = -p and σ_{rr}(r = b) = 0, respectively.

2.1 Weak Form, Isoparametric Elements, Gauss Quadrature

The given element is cylindrical in shape and force is acting along the circumference from inside-out. Moreover, It has plain strain condition.Therefore the problem can be converted into a 1-D problem of a strip along radial direction, where point force is to be applied on the left-end of the strip and finite element modelling is done for this strip.
Using Isoparametric elements nodal coordinates and displacements are interpolated by using the same Ansatz function $\underline{\mathrm{N}}(\xi)$(2.4)

$$[\mathrm{N}] = \left[\frac{1}{2}(1-\xi), \frac{1}{2}(1+\xi)\right]^{\mathrm{T}} \tag{2.4}$$

The interpolated geometry and displacements are given by

$$r = N_1 r_1 + N_2 r_2 \tag{2.5}$$

$$u = N_1 u_1 + N_2 u_2 \tag{2.6}$$

One point quadrature is used in this problem, with $\xi = 0$ and weight as 2. The Jacobian and B-Matrix are given by

$$J = \frac{r_2 - r_1}{2} \tag{2.7}$$

$$B = \begin{bmatrix} -\frac{1}{r_2 - r_1} & \frac{1}{r_2 - r_1} \\ \frac{(1-\xi)}{r_1(1-\xi)+r_2(1+\xi)} & \frac{(1+\xi)}{r_1(1-\xi)+r_2(1+\xi)} \end{bmatrix} \tag{2.8}$$

The Discretized weak form is now given by

$$\delta W = \delta u_r \left[\int_{-1}^{1} B^\top \cdot C_t \cdot B \cdot N^\top \cdot \hat{r} \cdot |J| \; d\xi \cdot U - [a \cdot \sigma_{rr}] \right] = 0 \tag{2.9}$$

with elemental stiffness and nodal forces calculated as

$$K_e = \int_{-1}^{1} B^T C_t B N^T \hat{r} |J| \cdot d\xi \tag{2.10}$$

$$F_{int} = \int_{-1}^{1} B^T \cdot \sigma \cdot N^\top \cdot r \cdot |J| \cdot d\xi \tag{2.11}$$

$$F_{ext} = a\sigma_{rr} \tag{2.12}$$

where C_t is the material tangent stiffness

2.2 Visco-Elastic behaviour of Material

The linear visco-elastic behavior of the material is described by the equations:

$$\underline{\sigma} = \underline{\mathbf{C}} \cdot \underline{\varepsilon} + \underline{\sigma}^{ov} \tag{2.13}$$

$$\underline{\dot{\sigma}}^{\text{ov}} = Q \, \text{dev}(\underline{\dot{\varepsilon}}) - \frac{1}{T} \underline{\sigma}^{\text{ov}} \tag{2.14}$$

wherein C is the isotropic (long-term) elastic stiffness matrix, expressed by Young's modulus E and Poisson ration ν. The evolution of the overstress σ_{ov} (as internal state variable) is governed by the modulus Q and a characteristic time scale T.
The overstress is calculated using the previous overstress forming an Initial Value problem. Hence Euler Modified method is used for it's discretization.

$$\sigma_{m+1}^{ov} = \frac{1}{1 + \frac{\Delta t}{2T}} \left[\left(1 - \frac{\Delta t}{2T} \right) \sigma_m^{ov} + Q \left[\Delta\varepsilon - \frac{trace(\Delta\varepsilon)}{3} \right] \right] \tag{2.15}$$

The Material Tangent is given by

$$C_t = \underline{C} + \frac{Q}{1 + \frac{\Delta t}{2T}} \begin{bmatrix} 2/3 & -1/3 \\ -1/3 & 2/3 \end{bmatrix} \tag{2.16}$$

$$C = \tfrac{E}{(1+v)(1-2v)} \begin{bmatrix} 1-v & v \\ v & 1-v \end{bmatrix}$$

2.3 Global Stiffness Matrix,Global Internal Forces,Newton-Raphson Scheme

The elemental stiffness matrices and internal forces are assembled to get global stiffness matrix and global internal forces .Together with external forces , the system of non-linear equations is solved using Newton-Raphson Method and this is done fo every time step to calculate the displacements, stresses.

$$K(\hat{u} + \Delta\hat{u}) \approx K(\hat{u}) + \frac{\partial K}{\partial \hat{u}}\Delta\hat{u} = 0 \tag{2.17}$$

$$K_t \Delta\hat{u} = F_{ext} - F_{int} \tag{2.18}$$

3 Structure of Program

The program has a main script named *MainFEM.m.* It has four other function scripts

- elementroutine.m
- materialroutine.m
- analyticalsolution.m
- meshrefinement.m

3.1 MainFEM.m

The main program is where execution of the program starts. Input material and geometrical parameters are given in the main program. The vectors and matrices are initialized to zero.
Newton -Raphson Scheme for every time step is implemented. Element routine is called for every element and individual stiffness matrices and internal force vectors are assembled to get Global Stiffness matrix and Global Internal forces.Time Scaling is employed upto loading time and external global force vector is assembled accordingly.
The system of equations is solved and displacements are updated until the convergence criteria is met. The iteration at which convergence takes place is printed for every time step.
The required results are then plotted.

3.2 elementroutine.m

Element routine is a function called from main program . It implements the shape functions, gauss quadrature scheme and calculates BMatrix . It calculates elemental stiffness matrix and internal for given parameters and nodal values. For element stiffness matrix, material tangent stiffness matrix is required and material routine is called to get overstress values , Material Tangent and Stress.

3.3 materialroutine.m

Material routine is a function called from the element routine. It calculates the material tangent stiffness and stress from overstress values implemented using the Euler Modified method.

3.4 analyticalsolution.m

Theoretical values for the linear elastic case for the given parameters is calculated.

4 User Manual

All the files have to be saved in a single folder and opened in GNU Octave.Required parameters are given in the array *params* in the order of [E[MPa], poisson's ratio ,Q [MPa], T[s], a[mm], b[mm], pmax[MPa], tL[s], tf[s]]
The program after running prints the number of iterations it took for each time step to converge and radial and cylindrical components of stress and over stress.

5 Verification

Verification is done for Q=0(linear elastic case) and the number of iterations it took for Newton-Raphson to converge for each time step is 1. The output of the program is:

```
converged for time: 1
converged after iteration: 1
converged for time: 2
converged after iteration: 1
```

```
converged for time: 3
converged after iteration:0
converged for time: 4
converged after iteration:0
converged for time: 5
converged after iteration:0
converged for time: 6
converged after iteration:0
converged for time: 7
converged after iteration:0
converged for time: 8
converged after iteration:0
converged for time: 9
converged after iteration:0
converged for time: 10
converged after iteration:0
```

The displacements should match with analytical solution.

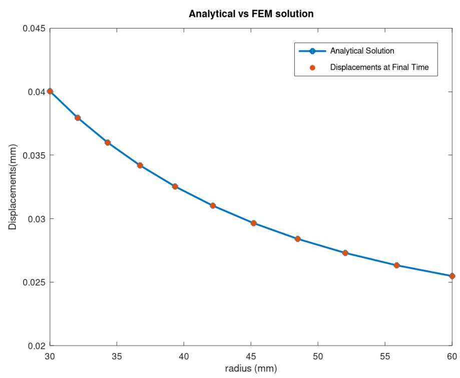

Figure 3: Analytical Solution vs FEM solution for Q=0

6 Results

The Displacements , Stresses , Widening History are shown in the below figures

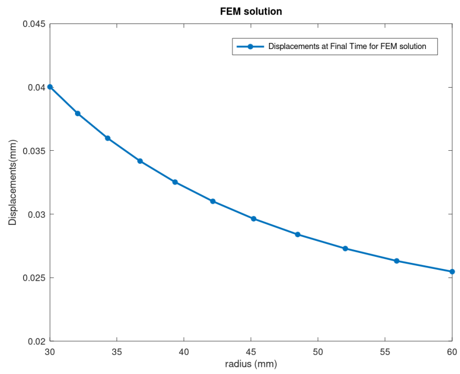

Figure 4: Displacement at Final time

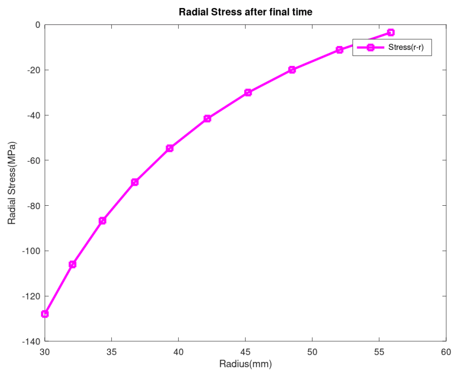

Figure 5: Stresses in Radial Direction

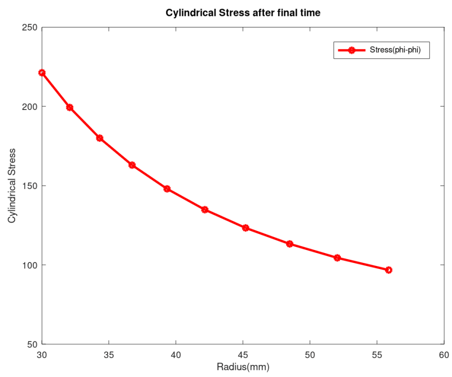

Figure 6: Stresses in Cylindrical direction

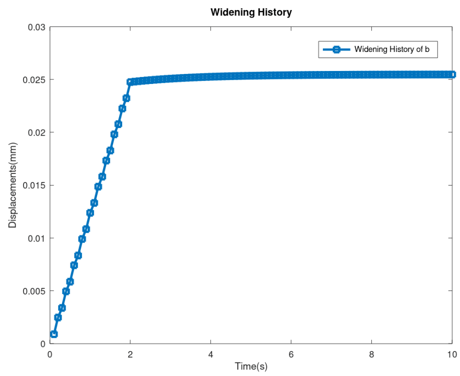

Figure 7: Widening of last node(b) w.r.t time