Assignment on Non-Linear Finite Element Method

Abrar Hyder Mohammed - 65092

Contents

1 Working problem

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 pmax and is then hold until tf as shown in Figure 2. Plain strain in z-directions are equal to zero conditions are assumed.

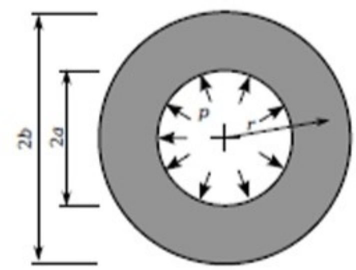

Figure 1: Thick-walled pipe

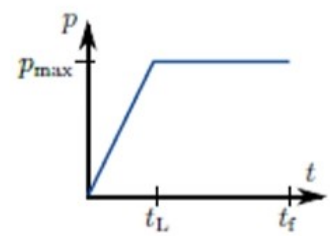

Figure 2: Load sequence

2 Thoery

The given equilibrium condition is

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

Therein, the only non-vanishing displacement component is u (r) as the displacement in radial direction. The boundary conditions for the given problem in the above figure are

$$\sigma_{rr}(r = a) = -p$$

$$\sigma_{rr}(r = b) = 0$$

respectively.

The stress and strain in voigt notation are as follows

$$\underline{\sigma} = \begin{bmatrix} \sigma_{rr} \\ \sigma_{\phi\phi} \end{bmatrix}, \quad \underline{\delta}\varepsilon = \begin{bmatrix} \delta\varepsilon_{rr} = \frac{\partial \delta u_r}{\partial r} \\ \delta\varepsilon_{\phi\phi} = \frac{\partial u_r}{r} \end{bmatrix}$$

2.1 Discretization of weak form

Because of the axisymmetric condition the quantities such as stress ,displacement,strain must be independent of circumferential variable Θ.

The linear shape function for u(r) is given by

$$[\mathrm{N}] = \left[\frac{1}{2}(1-\xi), \frac{1}{2}(1+\xi)\right]^{\mathrm{T}} \tag{2.11}$$

$$r = \begin{bmatrix} r_1 \\ r_2 \end{bmatrix} \tag{2.12}$$

The quadrature with one gauss point is used and hence

$$\xi = 0$$

$$w = 2$$

The Jacobian is determined using the formula

$$J = \frac{\delta N}{\delta \xi} \cdot \hat{r} = \begin{bmatrix} -\frac{1}{2} & \frac{1}{2} \end{bmatrix} \begin{bmatrix} r_1 \\ r_2 \end{bmatrix} \tag{2.13}$$

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

The strain displacement matrix is also computed using the relations

$$B = J^{-1} \cdot \frac{\delta N}{\delta \xi}$$

Finally we get the strain displacemnet marix which is used in weak form in order to obtain the stiffeness matrix and the internal forces.

$$B = \begin{bmatrix} -\frac{1}{r_2 - r_1} & \frac{1}{r_2 - r_1} \\ \frac{(1-\xi)}{r_1(1-\xi)+r_2(1+\xi)} & \frac{(1+\xi)}{r_1(1-\xi)+r_2(1+\xi)} \end{bmatrix} \tag{2.14}$$

The weak form of the given problem is

$$0 = \delta W = \int_a^b \underline{\delta\varepsilon}^{\mathbf{T}} \cdot \underline{\sigma} r \mathrm{d}r - [r\sigma_{rr}\delta u_r]_{r=a}^{b} \tag{2.15}$$

by discretizing the equation(2.15) we get the weak form in terms of (B,N,J) which is later solved to find the nodal displacements.

$$K_e = \int_{-1}^{1} B^T C_t B N^T \hat{r} |J| \cdot d\xi \tag{2.16}$$

$$F_{int} = \int_{-1}^{1} B^T \cdot \sigma \cdot N^T \cdot r \cdot |J| \cdot d\xi \tag{2.17}$$

$$F_{ext} = a\sigma_{rr} \tag{2.18}$$

2.2 Material Tangent and overstresses

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

$$\underline{\sigma} = \underline{\mathbf{C}} \cdot \underline{\varepsilon} + \underline{\sigma}^{\mathrm{ov}} \tag{2.21}$$

$$\underline{\dot{\sigma}}^{\mathrm{ov}} = Q\,\mathrm{dev}(\underline{\dot{\varepsilon}}) - \frac{1}{T}\underline{\sigma}^{\mathrm{ov}} \tag{2.22}$$

by discretizing the eq (2.22) we obtain overstresses and by substituing this overstress in stress equation (2.21) material tangent stiffness matrix is obtained.

$$\sigma_{m+1}^{ov} = \sigma_m^{ov} + Q\,\mathrm{dev}(\Delta\varepsilon) - \frac{\Delta t}{T}\left[(1-v)\sigma_m^{ov} + v\sigma_{ov}^{m+1}\right]$$

$$Dev(\varepsilon) = \Delta\varepsilon - \left[\frac{1}{3}\,\mathrm{trace}(\Delta\varepsilon)\right]$$

For the EM case v=1/2 .Hence the resultant over stress will be

$$\sigma_{ov}^{m+1} = \frac{1}{1+\frac{\Delta t}{2T}}\left[\left(1-\frac{\Delta t}{2T}\right)\sigma_{ov}^{m} + Q\left[\Delta\varepsilon - \frac{1}{3}\,\mathrm{tr}\,\mathrm{ace}((\Delta\varepsilon)\right]\right. \tag{2.23}$$

Material tangent stiffness is defined as

$$C_t = \frac{\delta\sigma_{m+1}}{\delta\varepsilon_{m+1}} = \frac{\delta\sigma_{m+1}}{\delta\varepsilon_{m+1}} + \frac{\delta\sigma_{m+1}}{\delta\sigma^{o1}_{m+1}} \cdot \frac{\delta\sigma^{ov}_{m+1}}{\delta\varepsilon_{m+1}}$$

By substituting eq(2.23) in in eq(2.21) and discritizing the eq(2.21) we obtain the C_t.

$$C_t = C + \frac{1}{1+\frac{\Delta t}{2T}} \times Q \begin{bmatrix} 2/3 & -1/3 \\ -1/3 & 2/3 \end{bmatrix} \tag{2.24}$$

2.3 Newton Raphson method and Assembly

In this section Global stiffness matrix and global internal forces are calculated,because of non linearity direct solving of nodal displacement is not possible.Hence NR method is used .

$$K(\hat{u}+\Delta\hat{u}) \approx K(\hat{u}) + \tfrac{\partial K}{\partial \hat{u}}\Delta\hat{u} = 0$$

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

3 overview of program implementation

The programm consists of 5 files i.e. 1) main file 2)Elementroutine file 3) Materialroutine file 4) Analytical solution file 5) Meshgenerator file

The mail file contains all the input parameters required for the given problem.By calling the elementroutine file in the main file the global stiffness matrix and global internal forces are calculated.Load scaling and Newton Raphson method is performed in this file. Finally all the plots are generated here.

The element routine file contains strain displacement matrix(B) and the Jacobian(J) . It gives the two output values element stiffness matrix(k) and the internal force(F).The materialroutine file is called by this file to get the inputs of overstress and previous strains.Stresses for linear and non linear case is calculated here.

The material routine file gives the two output values i.e. material tangent stiffness and overstresses.

Mesh generator file is used to get the nodal positions of all the nodes based on the number of elements given by user.

The analytical file is used to calculate the analytical solution using the required the input parameters given in the main file.

4 user manual

The program starts from main.m file. When the user runs the program a prompt command pops out wherein user needs to enter [0]:Linear case and [1]: Non linear case.

INput: The user can access all the input parameters from the main.m file itself.

OUtput: The program generates the following outputs Displacement,stresses and strains . All the values are stored in the main.m file.

5 verification

5.1 convergence and results

The visco elastic material must converge to analytical solution when Q=0 i.e.(Elastic case):

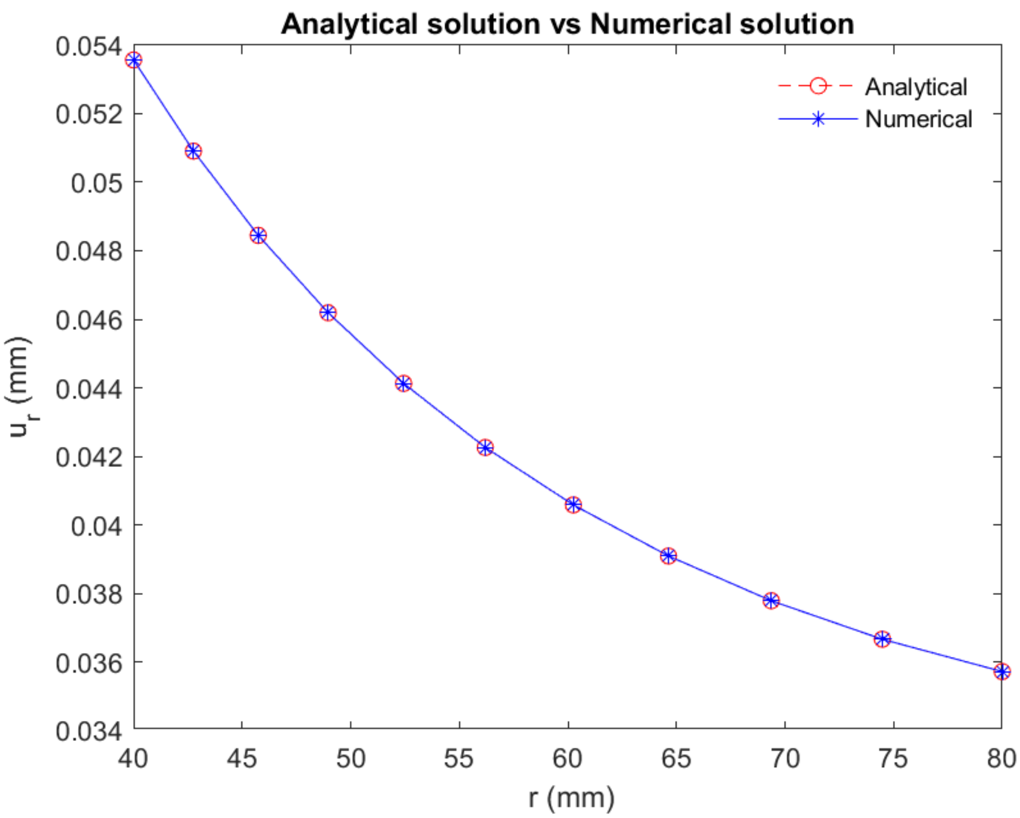

Figure 1: Analytical Solution vs Numerical Solution

The visco elastic material with respect to analytical solution when Q=50000 i.e.(Viscoelastic case):

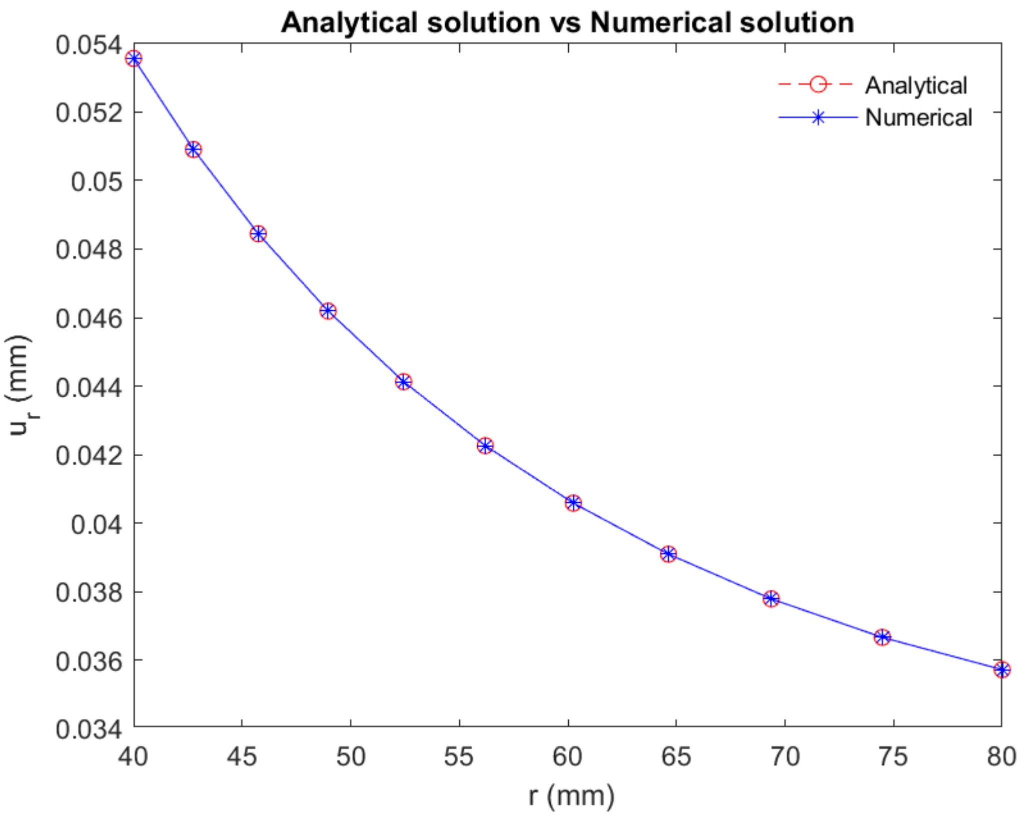

Figure 2: Analytical Solution vs Numerical Sloution

Both elastic and non elastic solution converges with the analytical solution.

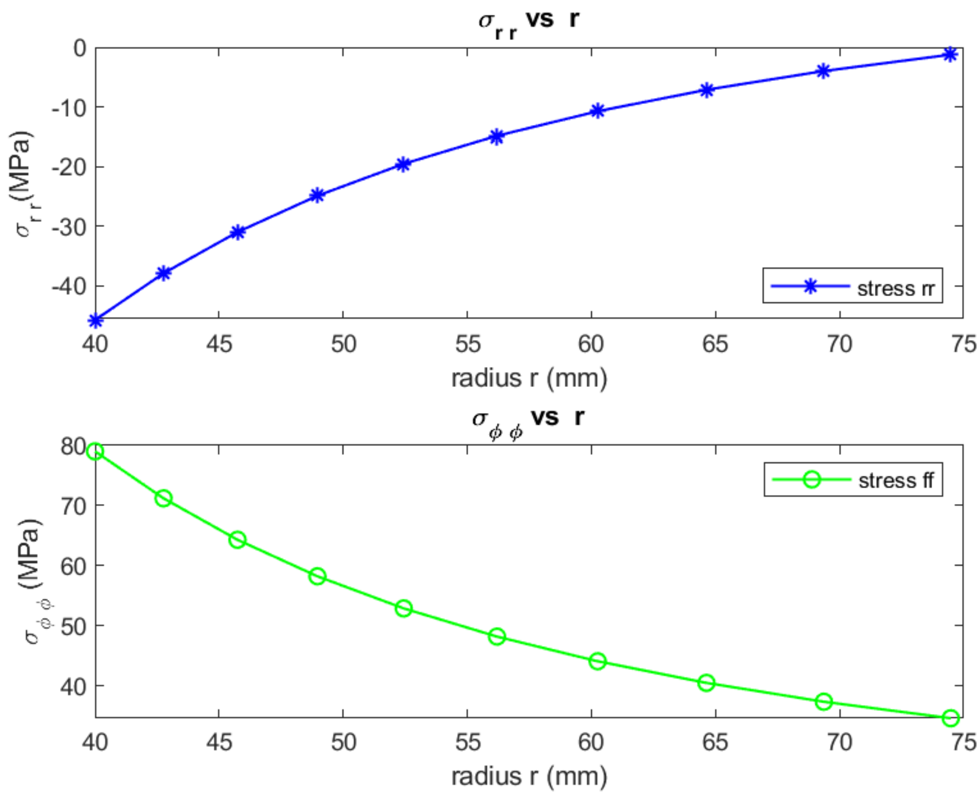

Figure 3: Stresses rr and ff

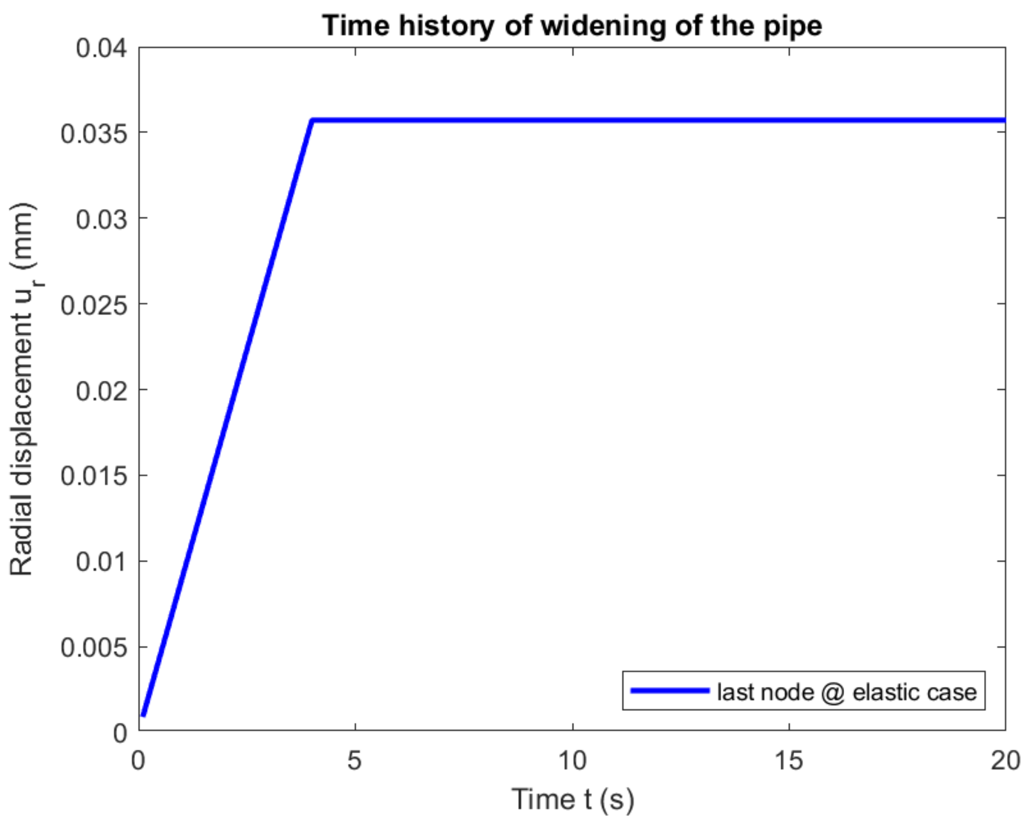

Figure 4: Time history of widening of the pipe linear case

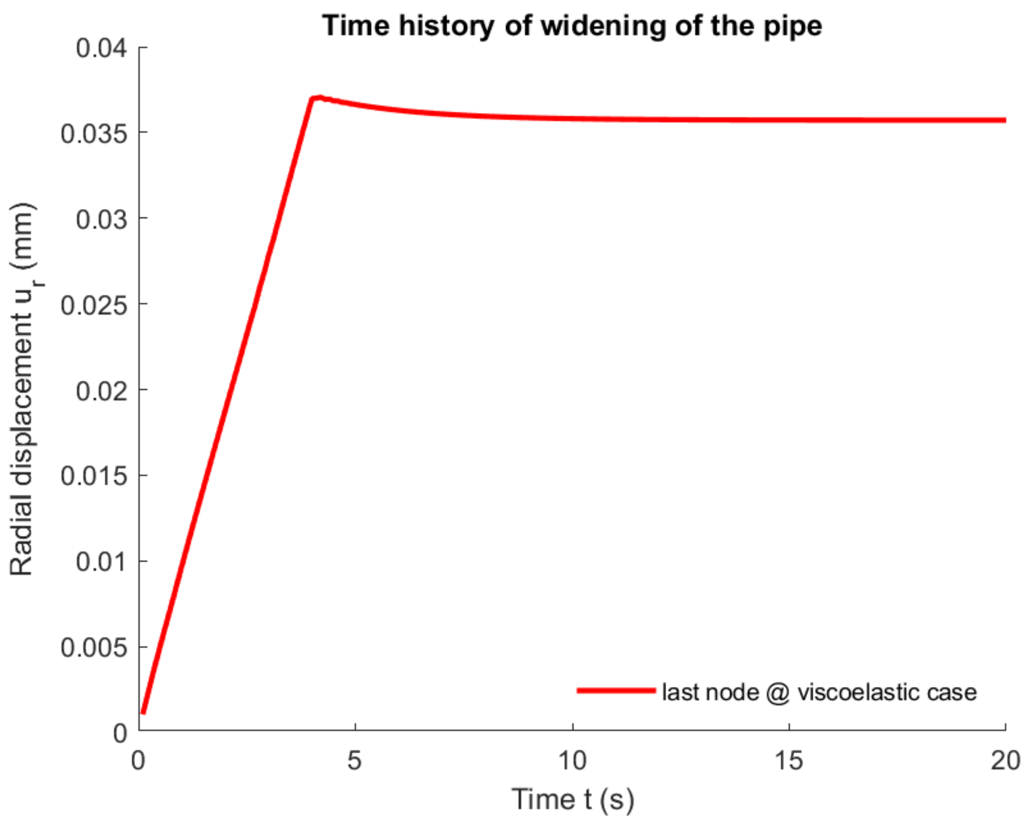

Figure 5: Time history of widening of the pipe non-linear case