MAE 7750: HW #4

Hanfeng Zhai

May 23, 2023

You will consider J2 plasticity with linear isotropic hardening. The yield function is defined as:

$$f(\bar{\mathbf{s}}, \hat{\alpha}) = \|\bar{\mathbf{s}}\| - \sqrt{\frac{2}{3}}\left[Y_0 + \hat{H}\right] \leq 0 \tag{1}$$

and you will consider the following material properties:

- *Young's modulus: 200GPa*
- *Poisson's ratio: 0.3*
- *Coefficient of linear hardening* $\hat{H} = 1930MPa$
- *Coefficient of linear kinematic hardening* $H = 1GPa$
- *Initial yield stress* $Y_0 = 268MPa$

Pseudocode *Design a detailed pseudocode for a small strain elastoplasticity problem in 2D. The pseudocode should refer to which functions and subroutines will be called for each step. Discuss what happens in each step and also mention the relevant variables. (the file points to the consists of the "hyper" folder to find other subroutines and read input files. Specific emphasis should be placed on the predictor-corrector solution scheme. What are the convergence criteria? Build on the pseudocode from HW3 (this should be the first step before you start coding).*

Here, the main file is not majorly modified, whereas the plasticity implementations are mainly controlled by `elasticity` & `fem`, representing the definition of the constitutive models and the FEM iteration updates. The two related algorithms' pseudocode (Algorithms 2 & 3) are also attached after the pseudocodes for the main file (Algorithm 1).

In my implementation, I define two different boundary conditions. For the first boundary condition, I apply a radial displacement of 5×10^{-5} on the interior boundary. For the second boundary condition, I apply a radial displacement of -1×10^{-4} on the exterior boundary.

The convergence is majorly governed by the combination of three factors: **tolerance**, machine precision, and maximum iterations. Also, the incremental loading loop steps and the general meshing structures, the geometry also affect the convergence. In our implementation,

Algorithm 1 Nonlinear Finite Element Analysis for Elastoplasticity

function `cut`($\mathbf{M}$, bcDofs, freeDofs)

    Cut the degree of freedom of the elasticity tensor $\mathbf{M}$ into the boundary and free degree of freedom parts.

function `verify`($\mathbf{U}$, $\mathbf{U}_{ref}$, bcDofs, tolerance)

    Checks if the values of $\mathbf{U}$ at the indices specified in bcDofs are the same as the corresponding values in $\mathbf{U}_{ref}$ up to the specified tolerance. If they are not, it throws an error or warning.

Define the material properties: Set material parameters for steel: E, ν, $\hat{H}$, H, Y_0. Recall the constitutive model of Elastoplasticity (both the Neo-Hookean & St. Venant models)

Define the iteration parameters: Specify the number of steps in the loop, the size of each step, and the magnitude of the radial displacement applied to the interior boundary of the cylinder (I further change it to the exterior cylinder).

Define the solver parameters: Set the precision, tolerance, and the maximum number of iterations.

Specify the file and folders to save the output files. Create something like "`/dirout/`".

Generate mesh based on geometry and recall `gmsh` *python module.* One can either load pre-generated mesh in a specific directory or generate using the existing module.

Set boundary conditions and force vectors.

Assembly the global stiffness matrix. Solve based on $\mathbf{M} \cdot \mathbf{U} = \mathbf{F}$ (internal + external).

Apply the loading and update the information. Apply the iteration using the Newton-Raphson iteration solution scheme. Initialize the Residual, Stiffness, etc.

while Residual > Precision **do**

    Compute the internal stiffness matrix $\mathbf{K}$ from displacements.

    Compute the stiffness matrix $\mathbf{M}$ from $\mathbf{K}$.

    Solve the unknown displacement $\mathbf{U}$ from $\mathbf{M}$ and Residual $\mathbf{R}$.

    Compute Force $\mathbf{F}$ from $\mathbf{U}$; Compute external force $\mathbf{F}_{\text{ext}} = \mathbf{M}_{11}\mathbf{U}_1 + \mathbf{M}_{12}\mathbf{U}_2$.

    Update Residual $\mathbf{R} = \mathbf{F}_{\text{ext}} - \mathbf{F}_{\text{int}}$; Cut $\mathbf{R}$ into $\mathbf{R}_1$ & $\mathbf{R}_2$.

    Verify the boundary conditions satisfy preset tolerance.

    if normalized test function < precision **then**

        Update iteration.

    end if

    for Local nodes in Displacement nodes **do**

        Compute the local displacement $\mathbf{u}$, Obtain the reference configuration $\mathbf{X}$; $\rightarrow$ Get current configuration $\mathbf{x} = \mathbf{X} + \mathbf{u}$; $\rightarrow$ Obtain radial strain ϵ_r.

        Update nodal force and internal displacement from ϵ_r.

    end for

    for Local element in Overall elements **do**

        Compute different kinds of Stresses (i.e., Cauchy, First Piola-Kirchhoff, ...), and Cauchy Green strains $\mathbf{E}$.

    end for

end while

Plot the figures and postprocessing.

the tolerance is 1×10^{-7}, the machine precision is 1×10^{-15}, and the maximum iteration is 1000. I discretized my applied radial displacements into 10 loading steps.

Algorithm 2 Constitutive Model Definition

```
class Plasticity(E,ν,H,Ĥ,Y_0)
    Initialize and obtain the corresponding materials' properties, i.e., G, K, λ, μ, ...
    Obtain the stress and stiffness matrices from the corresponding constitutive laws.
    Define and obtain the properties related to both elasticity & plasticity.
class Neo-Hookean Elasto-Plasticity(Plasticity)
    Obtain the potential, stress, & stiffness from the hyperelasticity theories.
    Define the yield check and compute the plastic stress based on the passed plasticity
    parameters. Then compute the materials tangent from the plastic stress.
class St-Venant Elasto-Plasticity(Plasticity)
    Obtain the potential, stress, & stiffness from the hyperelasticity theories.
    Define the yield check and compute the plastic stress based on the passed plasticity
    parameters. Then compute the materials tangent from the plastic stress.
```

Algorithm 3 Finite Element Iteration Update

```
class FiniteElement
    Initialize all the related variables to be calculated, i.e., F, E, σ, S, K, ...
    Define operators to calculate shapes, obtain dimensions, & the integration points, ...
    Define the update operator:
        Looping over the integration points: update the related stresses and strains
        required in the calculations.
    Check whether the materials reach plasticity:
        First calculate the yield criteria f — if the material yields, update the new
        plastic stresses and strains.
    Compute forces, stiffness, etc.
```

Base BVP *Run the code and plot the deformed mesh for the St. Venant and Neo-Hookean constitutive laws. Plot the evolution of the equivalent plastic strain at select locations. Adjust the loading accordingly to make sure plastic loading occurs while staying within the small deformation limit.*

The deformed mesh (red) and the original mesh (blue) comparison for the Neo-Hookean model is shown in Figure 1. The cumulative plastic strains for the ten different loading steps are shown in Figure 2. The deformed mesh (red) and the original mesh (blue) comparison for the St. Venant model is shown in Figure 3. The cumulative plastic strains for the ten different loading steps are shown in Figure 4.

Alternate BVP *Change the boundary conditions and run an alternate BVP of your choice (consistent with HW3) making sure the response goes in the plastic regime.*

The deformed mesh (red) and the original mesh (blue) comparison for the Neo-Hookean

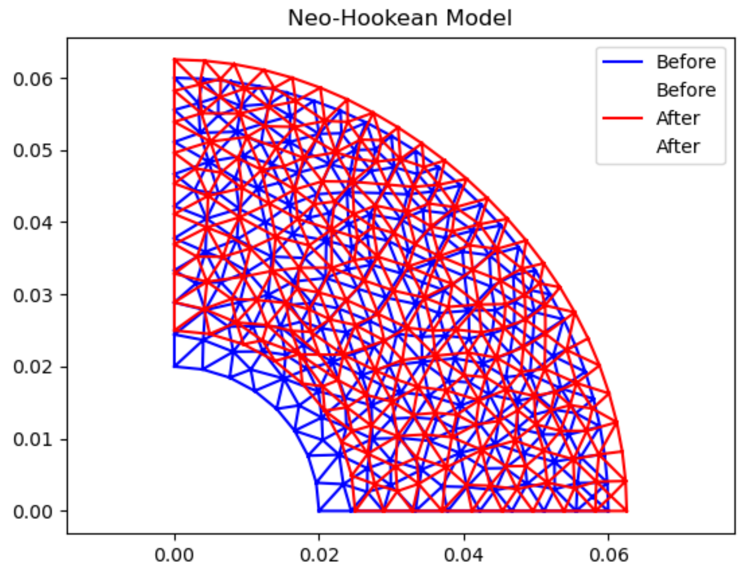

Figure 1: The original and deformed meshes for the Neo-Hookean elastoplasticity model under the first defined boundary conditions.

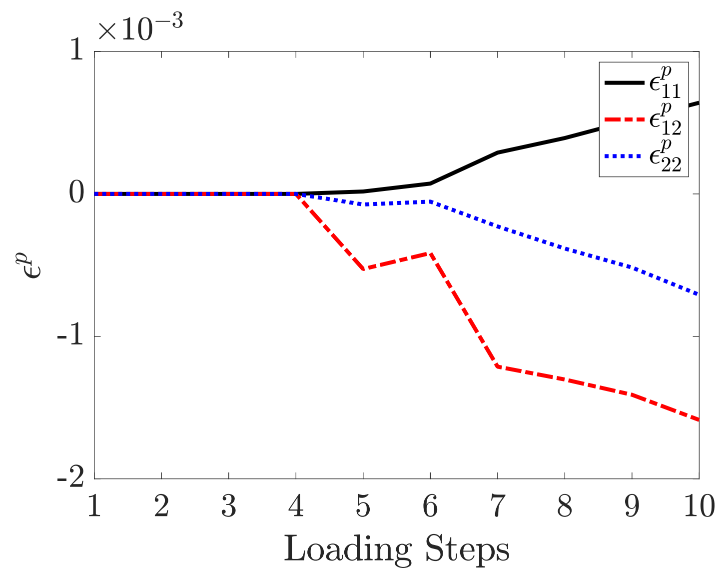

Figure 2: The evolution of the plastic strains (ϵ_{11}, ϵ_{12}, ϵ_{22}) for the Neo-Hookean elastoplasticity model under the first defined boundary conditions.

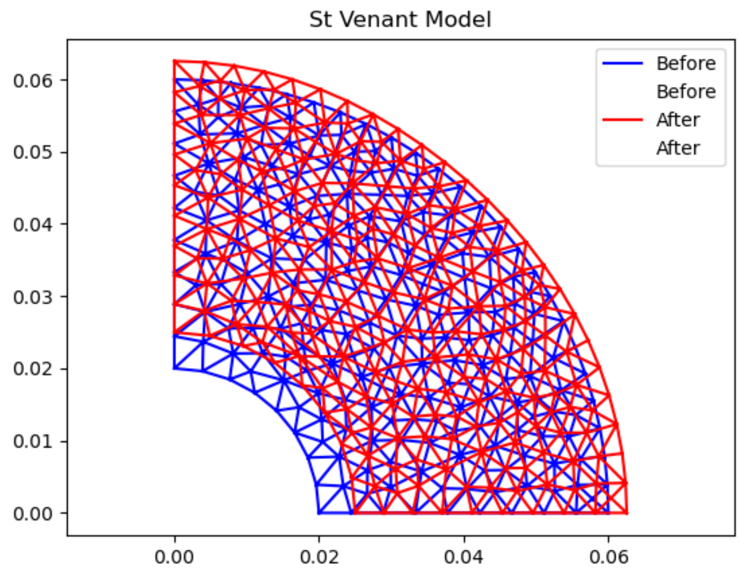

Figure 3: The original and deformed meshes for the St. Venant elastoplasticity model under the first defined boundary conditions.

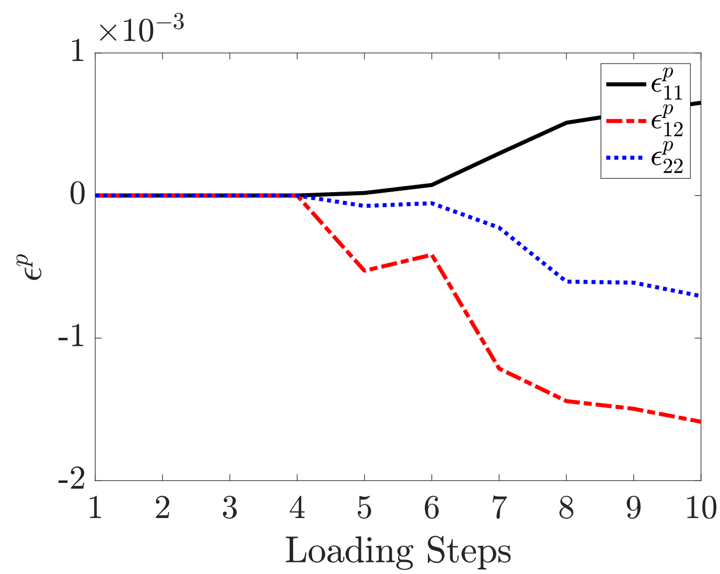

Figure 4: The evolution of the plastic strains (ϵ_{11}, ϵ_{12}, ϵ_{22}) for the St. Venant elastoplasticity model under the first defined boundary conditions.

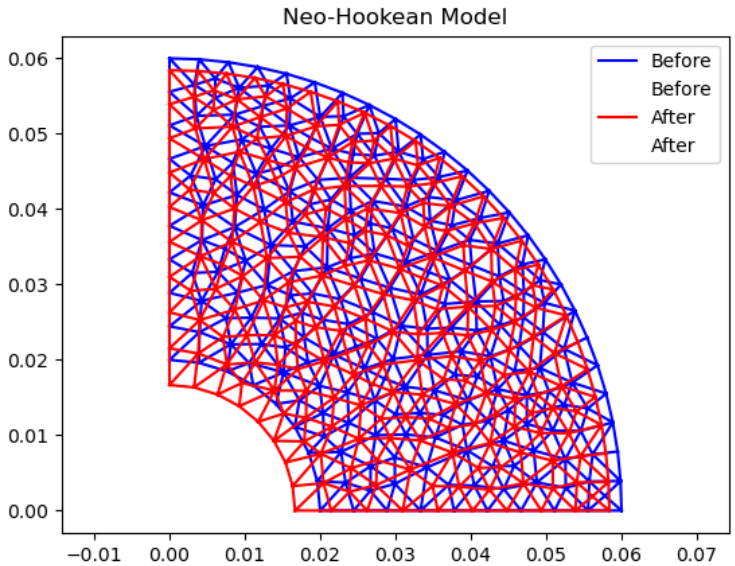

Figure 5: The original and deformed meshes for the Neo-Hookean elastoplasticity model under the second defined boundary conditions.

model is shown in Figure 5. The cumulative plastic strains for the ten different loading steps are shown in Figure 6. The deformed mesh (red) and the original mesh (blue) comparison for the St. Venant model is shown in Figure 7. The cumulative plastic strains for the ten different loading steps are shown in Figure 8.

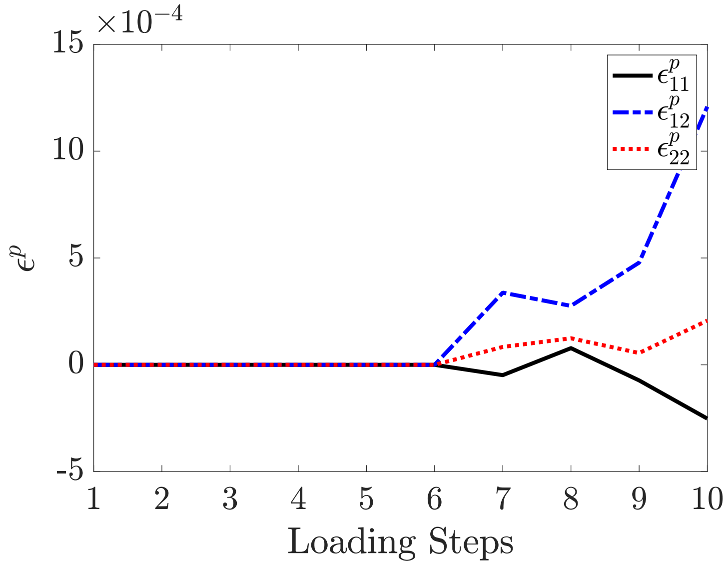

Figure 6: The evolution of the plastic strains (ϵ_{11}, ϵ_{12}, ϵ_{22}) for the Neo-Hookean elastoplasticity model under the second defined boundary conditions.

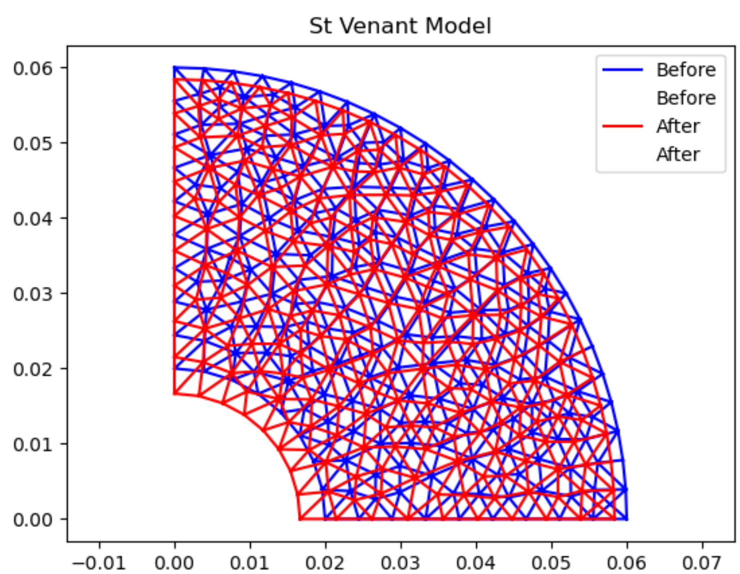

Figure 7: The original and deformed meshes for the St. Venant elastoplasticity model under the second defined boundary conditions.

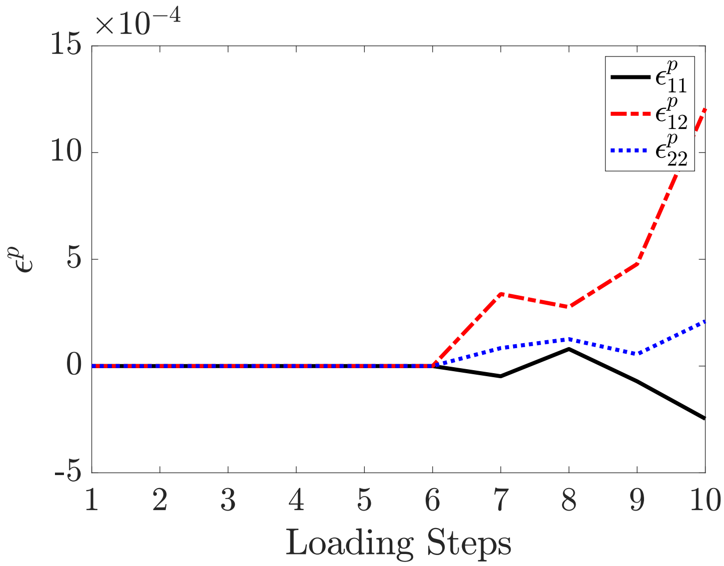

Figure 8: The evolution of the plastic strains (ϵ_{11}, ϵ_{12}, ϵ_{22}) for the St. Venant elastoplasticity model under the second defined boundary conditions.