Assignment 2
Non-linear elasto-plastic problem

Constitutive Modeling 2011

Division of Solid Mechanics
Lund University
September, 2011.

Computational Inelasticity FHLN05

Assignment 2 2011

General instructions

The written report should be returned to the Division of Solid Mechanics no later than 24/10 2011 at 10.00.

The assignment serves as a part exam. A maximum of 20 points can be obtained. The task can be solved individually or two persons can collaborate. If two persons work together they will obtain the same amount of points. The task is, a calculation of the nonlinear inelastic response of a simple structure. To solve the problem the CALFEM-toolbox should be used. In CALFEM, certain general FE-routines are already established and the task is to establish the extra routines needed to solve the boundary value problem.

The report should be clear and well-structured and contain a description of the problem, as well as the solution procedure that is needed and the results from the calculations in form of illustrative figures and tables. The program code should be included as an appendix. It should be sufficient with a maximum of 15 pages, appendix excluded.

In the text it can be assumed that the reader posses a basic knowledge in Solid Mechanics. The reader can also be assumed to have a knowledge of the problem description but it has been a while since he/she dealt with this type of analysis.

After reading the report, the reader should be able to reproduce the results just by reading through the report, i.e. without using the included program. This implies that all derivations of necessary quantities such as stiffness tensor of the material etc. should be presented in some detail.

Problem description

A plate with crack in the middle will be analyzed using the finite element method. A figure of the plate is found below.

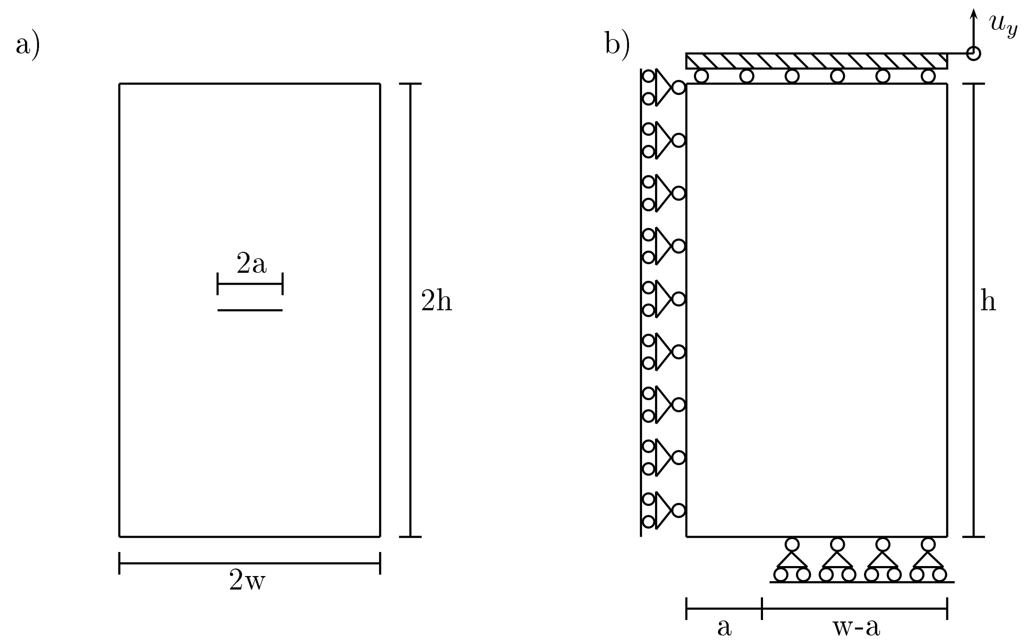

Figure 1: a) Entire plate with a crack in the middle. b) One quarter of the plate, showing the boundary conditions. Dimensions: w = 200 mm, h = 350 mm and a = 50 mm.

Due to symmetry, it is enough to consider one quarter of the plate. The plate will be pulled at the top a total distance of 1.5 mm. The plate is made out of steel and is assumed to obey plane stress conditions, i.e $\sigma_{33} = \sigma_{13} = \sigma_{23} = 0$. The elastic response is assumed to be linear isotropic and follow Hooke's law

$$\boldsymbol{\sigma} = \boldsymbol{D}^{*}\boldsymbol{\varepsilon}$$

where $\boldsymbol{\sigma} = [\sigma_{11}, \sigma_{22}, \sigma_{12}]^T$ and $\boldsymbol{\varepsilon} = [\varepsilon_{11}, \varepsilon_{22}, 2\varepsilon_{12}]^T$ are the stresses and strains respectively. The matrix $\boldsymbol{D}^{*}$ is Hooke's matrix for plane stress (eq.(4.112) in the course book). The elastic modulus of the material is $E = 200$ GPa and the Poisson's ratio is $\nu = 0.3$. The thickness of the plate is 2 mm. For the plastic loading, it is assumed that the material can be modelled a von Mises yield function and isotropic hardening. The internal variable for the hardening is given by $\dot{\epsilon}^p_{eff} = \sqrt{\frac{2}{3}\dot{\epsilon}^p_{ij}\dot{\epsilon}^p_{ij}} = \dot{\lambda}$. The yield stress is given by the format

$$\sigma_y = \sigma_{y0} + K_\infty \left(1 - e^{-\frac{h}{K_\infty}\epsilon^p_{eff}}\right)$$

where the parameters $\sigma_{y0} = 600$ MPa, $K_\infty = 200$ MPa and $h = 20000$ MPa.

Instructions

The task is to perform a calculation of the nonlinear response of the given structure. To solve the problem the CALFEM-toolbox should be used. In CALFEM, certain general FE-routines are already established and the task is to establish the extra routines needed to solve the nonlinear elastic boundary value problem.

The geometry for the problem is given in the two files `geomCrackedPlateC.mat`, containing a coarse mesh, and `geomCrackedPlateF.mat` containing a fine mesh. Both files are available on the course homepage. Use the coarse mesh when developing the program.

Three node linear triangle elements are used in the finite element calculations. In the calculations plane stress conditions are assumed, which is closer to the real physical loading situation than plane strain conditions.

For the global equilibrium loop a Newton-Raphson scheme should be implemented. For the integration of the elasto-plastic constitutive laws the radial return method should be used. Routines should be established that calculates the stresss and the algorithmic tangent stiffness matrix $\mathbf{D}^{ats}$.

The report should contain a detailed derivation of the numerical integration for the isotropic hardening and how the $\mathbf{D}^{ats}$ matrix is established.

In addition to described theory and necessary derivations as well as the description of the numerical implemention the following results should be provided

a) Plot of the stress distribution of the plate.

b) Plot of the global response during displacement controlled loading; total force versus displacement.

c) Identification of the area where plastic yielding first occurs and how the plastic area evolves with further loading (keep track of which elements have developed plastic strains).

d) Plot of the evolving effective stress in an element close to the crack tip (select one and indicate in a figure its location).

Note that it is possible to use a modified Newton-Raphson scheme to solve the problem, however, then it is not possible obtain the maximum number of points for the assignment.

Hints:

1.) From $f = 0$ the following can be obtained

$$\frac{(\sigma_{11}^t + \sigma_{22}^t)^2}{(2 + \frac{\Delta\lambda E}{(1-\nu)\sigma_y})^2} + \frac{3(\sigma_{11}^t - \sigma_{22}^t)^2 + 12(\sigma_{12}^t)^2}{(2 + \frac{3\Delta\lambda E}{(1+\nu)\sigma_y})^2} - \sigma_y^2 = 0$$

This can be used to find the quantity $\Delta\lambda$.

2.) Since plane stress is assumed, some care needs to be taken to ensure $f = 0$ during plastic loading in the radial return algorithm. In order to simplify the integration algorithm, the von Mises yield condition for plane stress can be written as (verify this!)

$$f = \sqrt{\frac{3}{2}\boldsymbol{\sigma}^T\boldsymbol{P}\boldsymbol{\sigma}} - \sigma_y$$

where $\boldsymbol{P}$ is a matrix which maps the stresses $\boldsymbol{\sigma}$ to the deviatoric stresses given by $\boldsymbol{s} = [s_{11}, s_{22}, 2s_{12}]^T$, i.e $\boldsymbol{s} = \boldsymbol{P}\boldsymbol{\sigma}$. The matrix $\boldsymbol{P}$ is given by

$$\boldsymbol{P} = \frac{1}{3}\begin{pmatrix} 2 & -1 & 0 \\ -1 & 2 & 0 \\ 0 & 0 & 6 \end{pmatrix}$$

3.) To plot the stress distriubtion, the stresses needs to be interpolated to the nodes. This can be done in the following manner:

```
for i=1:size(coord,1)
        [c0,c1]=find(Elements(:,2:4)==i);
        Seff_nod(i,1)=sum(Seff_el(c0))/size(c0,1);
end
```

where *Seff_nod* and *Seff_el* is the von Mises effective stress at the nodal points and in the elements, respectively. The matrix `Elements` contains the nodal conectivity and is included in the supplied geometry files.

Good luck!