Computational Inelasticity FHLN05

Assignment 2013

A non-linear elasto-plastic problem

General instructions

The written report should be returned to the Division of Solid Mechanics no later than 16 of October at 10:00.

The assignment serves as a part exam. A maximum of 5 points can be obtained. The task can be solved individually or in groups of two. If two persons work together they will obtain the same amount of points.

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 10 pages, appendix excluded.

It can be assumed that the reader posses a basic knowledge in Solid Mechanics and a understanding 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 etc. should be presented in some detail.

Note, a report should be handed in even if you're not able to solve all tasks or if your program doesn't work!

Problem description

A plate with hole should be examined under elasto-plastic deformation. The geometry of the plate is shown in figure 1.

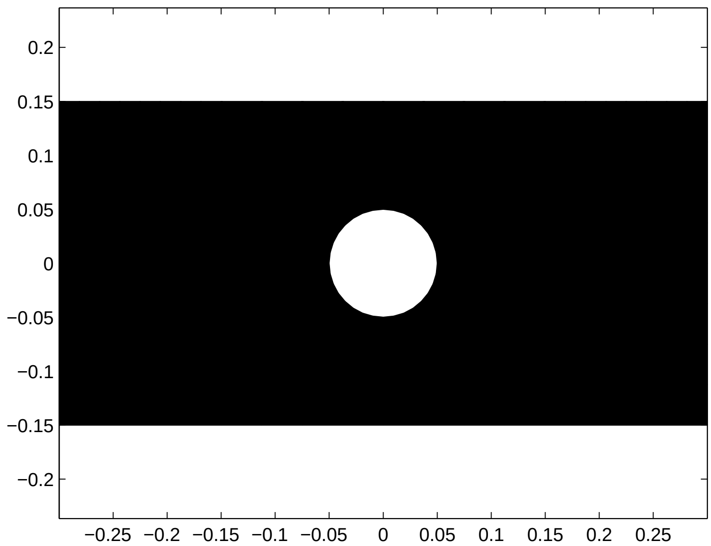

Figure 1: Plate with hole, units in m.

As in the computer lab the elastic response is assumed to be isotropic and independent of the third strain invariant, i.e. the stress strain relations are given by

$$\sigma_{kk} = 3K\epsilon_{kk} \qquad s_{ij} = 2Ge_{ij}$$

where σ_{ij} and ϵ_{ij} are the stress and strain tensors, respectively. Moreover, $s_{ij} = \sigma_{ij} - 1/3\delta_{ij}\sigma_{kk}$ and $e_{ij} = \epsilon_{ij} - 1/3\delta_{ij}\epsilon_{kk}$ are the deviatoric stress and strain tensors, respectively.

The elastic modulus of the material is $E = 140$ MPa and the Poisson's ratio is $\nu = 0.3$. In the elastic loading regime, the material can be considered linear, i.e. of the format given in eq. (4.89) in the course book (but reduced to plane stress conditions!). The thickness of the plate is 3 mm.

For the plastic loading, the material can be modelled using von Mises yield surface with isotropic hardening, where associated plasticity can be assumed. The yield stress is given by the format

$$\sigma_y = \sigma_{y0} + K_0 \left(\frac{\epsilon^p_{eff}}{\epsilon_0} \right)^{1/n}$$

where the parameters $\sigma_{y0} = 0.7$ MPa, $K_0 = 0.5$ MPa, $\epsilon_0 = 0.025$ and $n = 3$.

Instructions

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

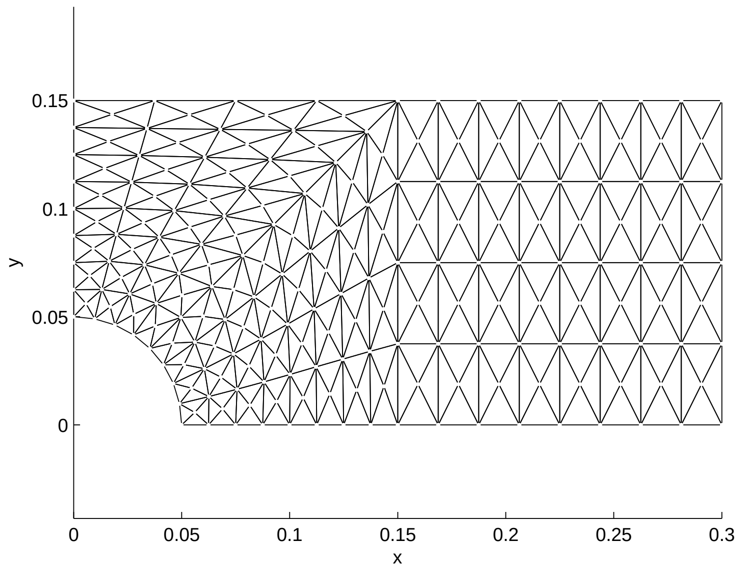

Figure 2: Plate with hole, units in m.

The geometry for the problem is given in the file `assignment_2013_geom.mat`, available on the course homepage. Due to symmetry, only one fourth of the plate is modelled, see the mesh in figure 2. A description of the variables supplied by the geometry file can be found in appendix A.1. Note that you may have to scale the increment in `bc` so that your steps don't become too large. For the global equilibrium loop a displacement controlled Newton-Raphson scheme should be implemented and for the integration of the elasto-plastic constitutive laws a fully implicit radial return method should be used. Three-node triangle elements are used for the finite element calculations.

The calculations should be done for plane stress condition, which is closer to the real physical loading situation than plane strain conditions and the plate should to be loaded well into the non-linear region.

Solve the following steps;

a) Implement two routines in Matlab. One that checks for elasto-plastic response and updates accordingly and one that calculates the algorithmic tangent stiffness $\mathbf{D}_{ats}$. See manuals on the course homepage.

b) Check that your routines work by comparing with the files; `check_elastic_update.mat`, `check_plastic_update.mat`, `check_elastic_Dats.mat` and `check_plastic_Dats.mat` on the course homepage. Note that both elastic and plastic behaviour can be checked.

c) Plot the effective stress distribution of the plate during displacement controlled loading.

d) Plot the global response during displacement controlled loading; total force versus displacement during two load cycles where the material should be loaded well into the plastic region.

e) Identify 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).

The report should contain a detailed derivation of the numerical integration for the isotropic hardening.

Appendix A

A.1 Variables in the supplied geometry files

Table A.1: Variables in the supplied geometry file.

Variable	Description	Size
bc	Boundary cond. during disp.-controlled loading	
edof	Element topology matrix	[nbr_elem×7]
enod	Node topology matrix	[nbr_elem×4]
ex	Element x-coordinates	[nbr_elem×3]
ey	Element y-coordinates	[nbr_elem×3]
nbr_dof	Number of degrees of freedom	
nbr_el	Number of elements	
nbr_node	Number of nodes	
plot_dof	Degree of freedom used for plotting	

A.2 Hints

1) From $f = f(\boldsymbol{\sigma}^{(2)}, K^{(2)}) = 0$ it is possible to derive a constraint that can be used to find the increment $\Delta\lambda$;

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

Note, the report should contain a derivation of this expression in order to get maximum number of points on the assignment. It is allowed to use Maple in order to simplify and identify that you have arrived at the same expression. Note that σ_y in the expression above should be calculated in current state.

2) In order to simplify the integration of the variables, the von Mises yield condition can be written as (verify this!);

$$f = \sqrt{\frac{3}{2}\boldsymbol{\sigma}^T \mathbf{P} \boldsymbol{\sigma}} - \sigma_y = 0$$

where $\mathbf{P}$ is a matrix that maps the stresses $\boldsymbol{\sigma}$ to the deviatoric stresses $\mathbf{s}$, i.e. $\mathbf{s} = \mathbf{P}\boldsymbol{\sigma}$. The matrix $\mathbf{P}$ is given by;

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

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

```
for i=1:nbr_node
    [c0,c1]=find(enod(:,2:4)==i);
    eff_nod(i,1)=sum(eff_el(c0))/size(c0,1);
end
```

where `eff_nod` and `eff_el` is the von Mises effective stress at nodal points and in the elements respectively. The matrix `enod` contains the nodal connectivity and is included in the supplied geometry file. The effective stress is then extracted element wise (in the same way as the displacements are) to the matrix `edeff` which can be used to draw contour-plots in Matlab with the command `fill`;

```
fill(ex',ey',edeff');
```

4) In order to solve the constraint for $\Delta\lambda$ the command `fzero` in Matlab could be used.

5) You could use a modified Newton-Raphson scheme to solve the problem, i.e. use the elastic tangent stiffness instead of $\mathbf{D}_{ats}$. The convergence will then be impaired but it could be useful when developing your program. Note that for a maximum number of points on the assignment you will need to use the full Newton-Raphson.

Have fun!