Computational Inelasticity FHLN05

Assignment 2015

A non-linear elasto-plastic problem

General instructions

A printed version of the report should be handed in at the Division of Solid Mechanics no later than 3 November at 10.00. The report and all matlab code should also be sent to henrik.askfelt[at]solid.lth.se or ylva.mellbin[at]solid.lth.se no later than 3 November 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 students 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 15 pages, appendix excluded.

It can be assumed that the reader possesses 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 section of a iron profile should be examined under elasto-plastic deformation. The geometry of the section is shown in figure 1.
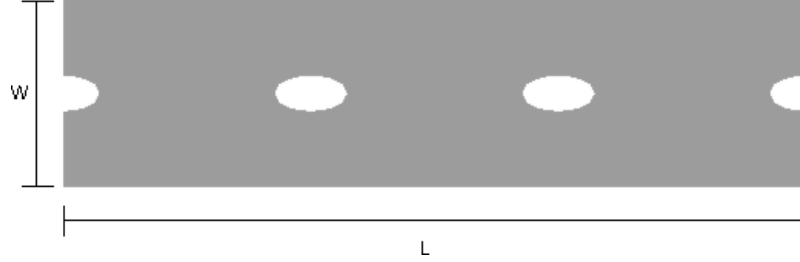

Figure 1: Geometry of the profile section.

The width of the profile is $W = 5cm$ and the length of the section is $L = 20cm$. Furthermore the holes in the profile are elliptical with a vertical radius of $0.5cm$ and a horizontal radius of $1cm$. Positioning origin in the lower left corner of the structure in figure 1, the four holes have there centres in the following four points $(0, \frac{W}{2}), (\frac{L}{3}, \frac{W}{2}), (\frac{2L}{3}, \frac{W}{2})$ and $(L, \frac{W}{2})$.

The elastic response is assumed to be linear, isotropic and independent of the third strain invariant. The stress strain relations are derived in chapter 4.9 in the course book and are given by

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

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

The elastic modulus of the material is $E = 200$ GPa 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 2 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 expression

$$\sigma_y = \sigma_{y0} + k\sigma_{y0}(\varepsilon_{eff}^p)^n$$

where ε_{eff}^p is effective plastic strain and the parameters $\sigma_{y0} = 600$ MPa, $k = 50$ and $n = 0.7$.

Assignment

The task is to calculate the elasto-plastic response of the given structure when
1) a force is applied on the external vertical boundaries and 2) when the vertical
boundaries are given a known displacement. The elasto-plastic response should be
retrieved by using the FE-method to solve the equilibrium equation (body forces
may be neglected). When solving this problem the CALFEM-toolbox should be
used. In CALFEM, certain general FE-routines are already established but you
need to establish extra routines in order to solve the elastic-plastic boundary
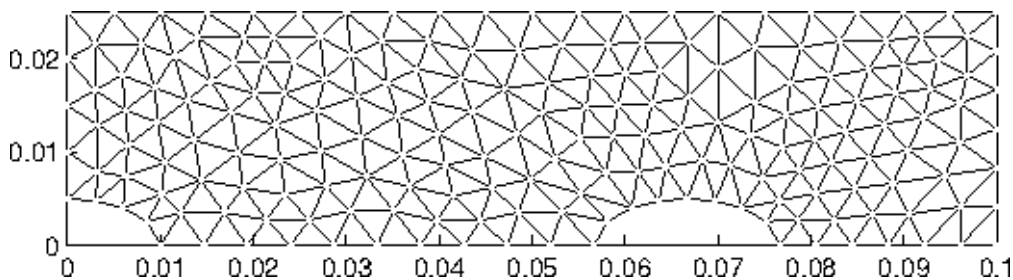value problem.

Figure 2: Example of mesh for profile section, units in m.

The geometry for the problem is given in figure 1. Due to symmetry, only one
fourth of the profile section is modelled, see example of mesh in figure 2. As in
the computer lab a file `TopConstMod_Assignment2015.m` may be used to retrieve
topology matrices, global incremental load vector `df` and Dirichlet boundary
conditions `bc`. For the global equilibrium loop a 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 (cf. chapter 18 in the course
book, note that plane stress conditions prevail!). 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 iron profile
should be loaded well into the plastic region.

The assignment includes the following:

- Derive the FE formulation of the equation of motion.

- Derive the equilibrium iteration procedure by defining and linearizing a
  residual, i.e Newton-Raphson procedure.

3

- Derive the numerical algorithmic tangent stiffness $\mathbf{D_{ats}}$ and the radial return method for isotropic hardening of von Mises yield surface.

- Calculate the elasto-plastic response of the link by implementing a FE program using the Newton-Raphson algorithm with a fully implicit radial return method, both load and displacement control should be investigated.

  - Implement a sub-routine `update_variables.m` in Matlab that checks for elasto-plastic response and updates accordingly (a manual for this routine may be found on the course homepage and the routine may be checked with `check_update_2015.mat`).

  - Implement a sub-routine `alg_tan_stiff.m` in Matlab that calculates the algorithmic tangent stiffness (a manual for this routine may be found on the course homepage and the routine may be checked with `check_Dats_2015.mat`).

- Using force controled loading load the structure well into the plastic region and then unload the structure.

- Using displacement control, load the structure well into the plastic region.

- Present the resulting response of the two load cases. The following results should be presented in an illustrative way:

  - The development of plastic response regions (initial, one or more intermediate steps and at maximum load).

  - The total force applied on the left boundary should be plotted against the displacement, in x-dir, of any node placed at the left boundary, e.g. the displacement of `plot_dof` given by `TopConstMod_Assignment2015.m` may be used.

  - Effective von Mises stress distribution at maximum load and after unloading.

The report should be well structured and contain sufficient details of the derivations with given assumptions and approximations for the reader to understand. Furthermore, some useful hints are given in appendix.

Appendix A

A.1 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 $\Delta\lambda$;

$$\frac{3}{2}(\boldsymbol{\sigma}^t)^T \mathbf{M}^T \mathbf{P} \mathbf{M} \boldsymbol{\sigma}^t - \sigma_y^2 = 0 \tag{A.1}$$

Note, the report should contain a derivation of this expression in order to get maximum number of points on the assignment. Note that σ_y in the expression above should be calculated in current state and that $\mathbf{M}$ also depends on $\Delta\lambda$!

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:size(coord,1)
    [c0,c1] = find(enod==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 nodal points and in the elements respectively. The matrix `enod` contains the nodal topology and is provided from `TopConstMod_Assignment2015.m`. 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.

Good luck!