Computational Inelasticity FHLN05

Assignment 2014

A non-linear elasto-plastic problem

General instructions

The written report should be returned to the Division of Solid Mechanics no later than 3 November at 10.00.

The assignment serves as a part of the examination. 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 link in a chain should be examined under elasto-plastic deformation. The geometry of the link is shown in figure 1. (Note that the same geometry was used in the computer lab)

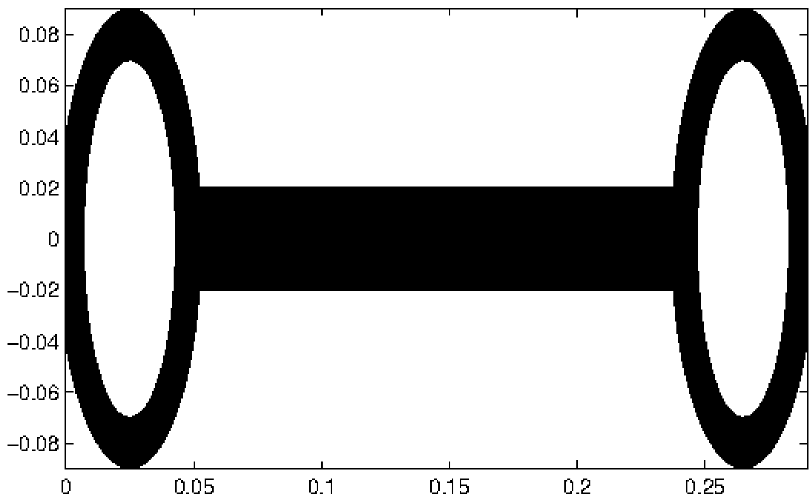

Figure 1: Geometry of link, dimensions in meter.

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

$$\sigma_{kk} = 3K\epsilon^e_{kk} \qquad s_{ij} = 2Ge^e_{ij}$$

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

The elastic modulus of the material is E = 200 GPa and 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 link is 2 mm.

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

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

where ε_{eff}^{p} is the effective plastic strain and the parameters $\sigma_{y0} = 600$ MPa, $K_{\infty} = 200$ MPa and $h = 20$ GPa.

Assignment

The task is to calculate the elasto-plastic response of the link 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.

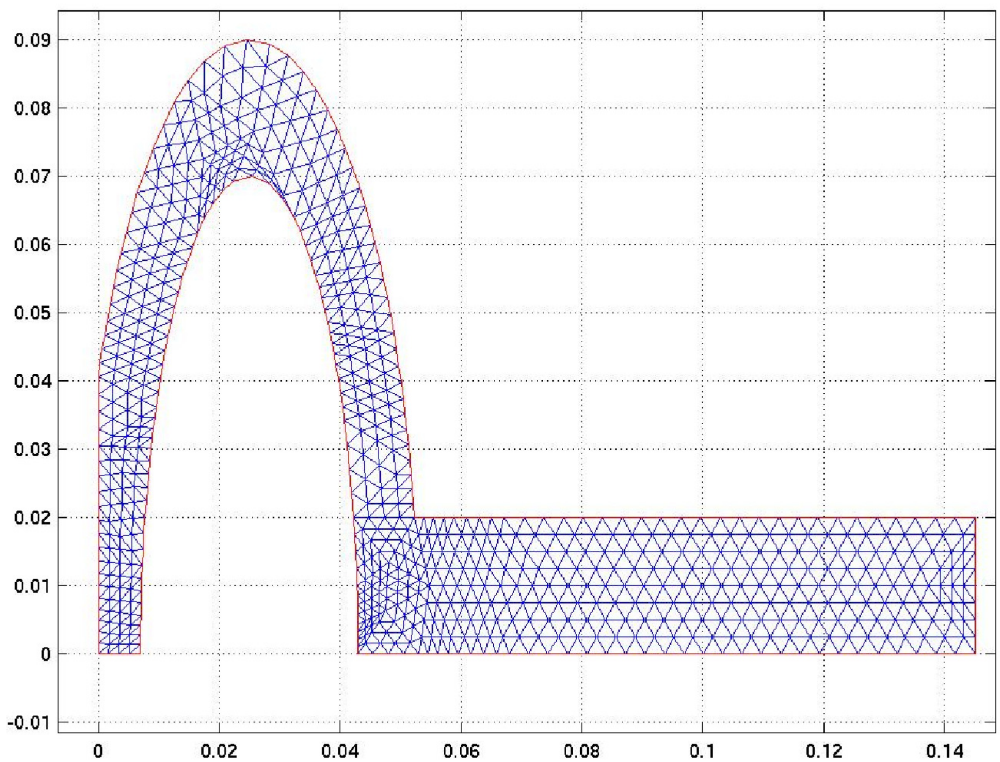

Figure 2: Example of mesh for link, units in m.

The geometry for the link is given in figure 1. Due to the present symmetry, only one fourth of the link should be modelled, see example of mesh in figure 2. As in the computer lab the file `TopConstMod.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 link 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.
- Derive the numerical algorithmic tangent stiffness $\mathbf{D_{ats}}$ and an update scheme for the stresses following the 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` 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.mat`).
  - Implement a sub-routine `alg_tan_stiff` 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.mat`).
- Using force controlled 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).
- Force-displacement curves of degree of freedom `plot_dof` given by `TopConstMod.m`.
- Effective von Mises stress distribution at maximum load 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. Note that this assignment serves as a part of the examination and therefore the teacher assistants are not allowed to assist in any programming. Furthermore, some useful hints are given in appendix.

Appendix A

A.1 Variables

Variable	Description	Size
`bc`	Dirichlet boundary conditions	
`coord`	Coordinates of nodes	[`nbr_node`×2]
`dof`	Degrees of freedom	[`nbr_node`×2]
`edof`	Element topology matrix	[`nbr_elem`×7]
`ex`	Element x-coordinates	[`nbr_elem`×3]
`ey`	Element y-coordinates	[`nbr_elem`×3]
`df`	External force increment vector	[`nbr_dof`×1]
`plot_dof`	Degree of freedom used for plotting	
`dtau_x`	Incremental traction stress in x-dir	
`du`	Incremental displacement in x-dir	
`th`	Thickness	
`control`	`control`=0 force controlled, `control`=1 displacement controlled	

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 $\Delta\lambda$;

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

   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.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!