Computational Inelasticity FHLN05

Assignment 2017

A non-linear elasto-plastic problem

General instructions

A written report should be submitted to the Division of Solid Mechanics no later than **October 30 at 10.00**, both a printed version and a digital version should be handed in. The digital version is sent via e-mail to marcus.alexandersson@solid.lth.se.

The assignment serves as a part exam, thus help with coding and debugging will not be provided. 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; including necessary derivations 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 posses basic knowledge of Solid Mechanics 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

Three different geometries are considered for manufacturing of a metal detail. The purpose of the component is to carry a distributed load exerted at $x = 0$ and $0.025 \leq y \leq 0.05$ m in the x-direction. Along the bottom boundary ($y = 0$, $0 \leq x \leq 0.05$ m) the detail is fixed from moving in both x- and y-direction. The first design is seen Figure 1.

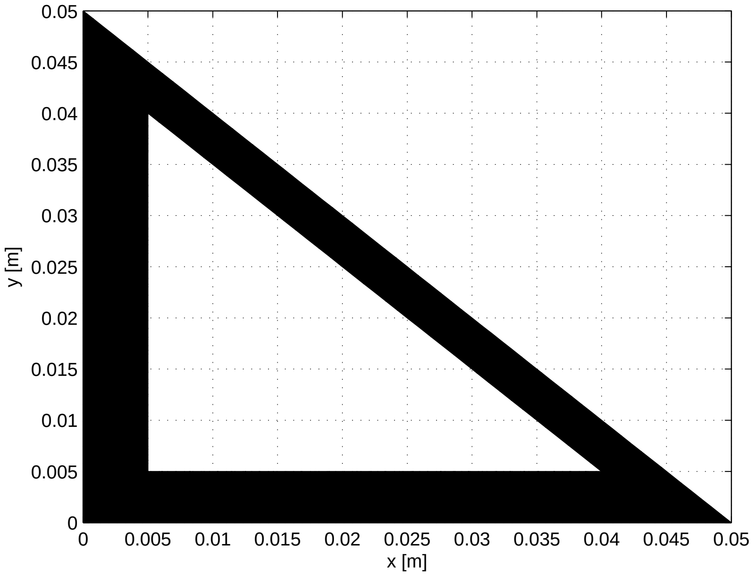

Figure 1: Geometry A considered for a metal profile.

The other two designs (B and C) are given in Figure 2 and Figure 3. All designs have the same thickness $t_h = 2$ mm and material volume V which can be used to calculate the dimensions of geometry B and C based on the area of A. Furthermore in the development process two different materials are considered. These are an annealed steel and a precipitation hardened aluminium alloy. The materials behave linear elastic in the elastic regime and due to the small thickness **plane stress is assumed**, i.e, Hooke's law for plane stress can be used to model elasticity. The plastic response of the materials can be modelled using von Mises yield surface with kinematic hardening:

$$f = \sqrt{\frac{3}{2}(s_{ij} - \alpha^d_{ij})(s_{ij} - \alpha^d_{ij})} - \sigma_{y0} = 0 \tag{1}$$

where associated plasticity can be assumed. The evolution of the back-stress α^d_{ij}

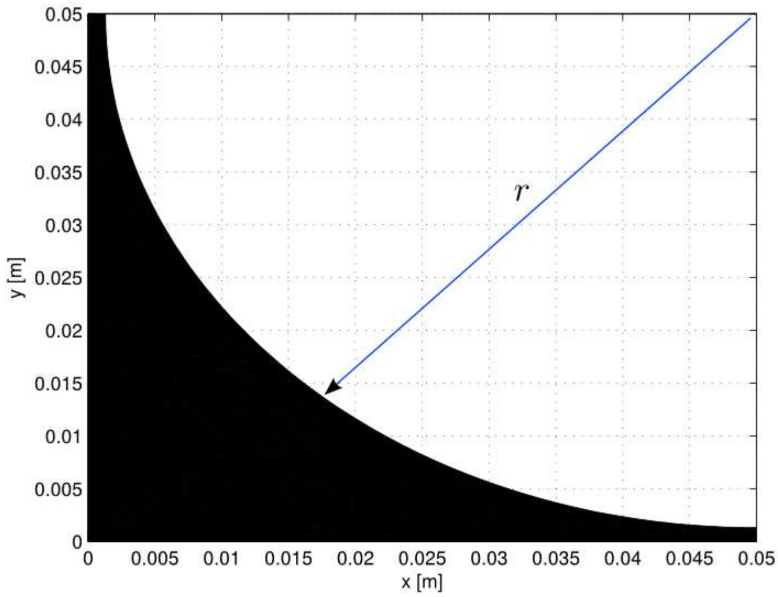

Figure 2: Geometry B considered for a metal profile.

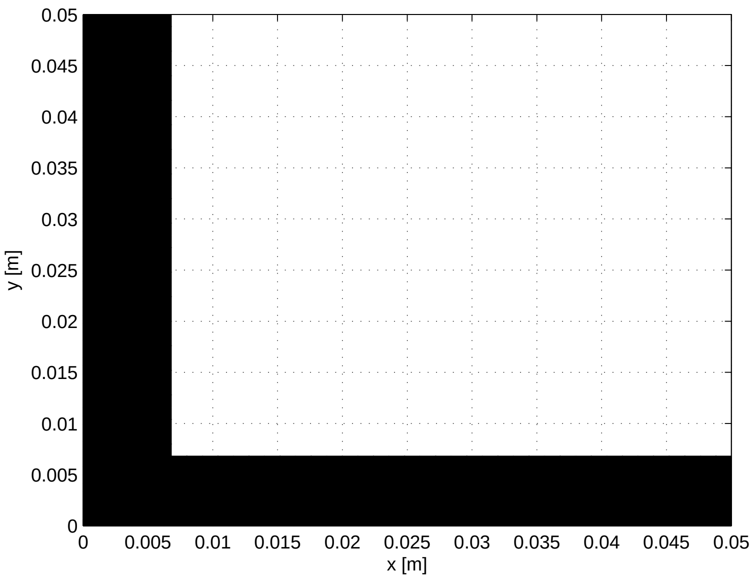

Figure 3: Geometry C considered for a metal profile.

is assumed to follow a Melan-Prager relation, i.e,

$$\dot{\alpha}_{ij}^{d} = c\dot{\varepsilon}_{ij}^{p} \tag{2}$$

	E [GPa]	ν [-]	σ_{y0} [MPa]	c [GPa]
Steel	210	0.3	250	10
Aluminium	70	0.32	310	30

Table 1: Material data

The task is to calculate the elasto-plastic response of the different designs. The elasto-plastic response is the solution to the equation of motion (static conditions may be assumed and body forces may be neglected). To solve the 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.

The routine `TopConstMod_Assigmnent2017.m` may be used to obtain the topology matrices and Dirichlet boundary conditions `bc` as well as the incremental external force `df`.

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 carried out using the plane stress assumption, which is closer to the real physical loading situation than plane strain conditions.

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 the radial return method for kinematic hardening of von Mises yield surface.
- Investigate the elasto-plastic response of the different designs with different materials by implementing a FE program using the Newton-Raphson algorithm with a fully implicit radial return method using **force control**. This includes:

- Implementation of the subroutines `update_variables.m` that checks for elasto-plastic response and updates accordingly (a manual for the routines is appended). The routines can be checked with data from `check_update_assignment2017.mat`.
- Implementation of the subroutines `alg_tan_stiff.m` that calculates the algorithmic tangent stiffness (a manual for this routine is appended) of the corresponding material. The routines can be checked with data from `check_Dats_assignment2017.mat`.

- Use force controlled loading and load the structures well into the plastic region (twice the load of initial yielding), return to the original position by unloading and then reverse the load and unload again (i.e, a load cycle). Investigate the stresses at peak load and after the structure has been unloaded after a the load cycle.
- The following results should be presented in an illustrative way:
  - An σ-ε curve for a load-cycle (loaded into the plastic regime) for the particular model under uniaxial loading, i.e, analytic expressions can be used. The response for both materials should be presented.
  - The development of plastic response regions of the different designs and materials when the structure is loaded well into the plastic region.
  - The effective von Mises stress distribution at maximum load and after unloading (take maximum load as twice that which cause initial yielding for the particular design and material).
  - The deformation pattern for the different structures at maximum load.

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.

Some interesting questions to consider are:

- Which design and what material is preferable and why?
- At what load is plasticity initiated?
- What are the limitations for application of the approach, i.e, what are the main assumptions?
- Are the results reasonable?

- How would you suggest to make improvements on the design?

Good luck!

Appendix A

A.1 Variables

Variable	Description
`bc`	Dirichlet boundary conditions
`coord`	Coordinates of nodes
`dof`	Degrees of freedom
`edof`	Element topology matrix
`ex`	Element x-coordinates
`ey`	Element y-coordinates
`df`	Incremental external force vector
`pressure`	Incremental traction stress in x-dir
`du`	Incremental displacement in x-dir
`th`	Thickness
`control`	= 0 force control, = 1 displacement control

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 $\Delta\lambda$;

$$\frac{3}{2}(\boldsymbol{\sigma}^t - \boldsymbol{\alpha}^{(1)})^T \mathbf{M}^T \mathbf{P} \mathbf{M} (\boldsymbol{\sigma}^t - \boldsymbol{\alpha}^{(1)}) - \sigma_{y0}^2 = 0 \tag{A.1}$$

Note, the report should contain a derivation of this expression in order to get maximum number of points on the assignment. Note that $\mathbf{M}$ 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!);

$$f = \sqrt{\frac{3}{2}\bar{\boldsymbol{\sigma}}^T \mathbf{P} \bar{\boldsymbol{\sigma}}} - \sigma_{y0} = 0 \tag{A.2}$$

where $\mathbf{P}$ is a matrix allowing a matrix representation of the von Mises yield criterion in the reduced stresses $\bar{\boldsymbol{\sigma}}$. Where;

$$\mathbf{P} = \frac{1}{3}\begin{bmatrix} 2 & -1 & 0 \\ -1 & 2 & 0 \\ 0 & 0 & 6 \end{bmatrix}, \qquad \bar{\boldsymbol{\sigma}} = \boldsymbol{\sigma} - \boldsymbol{\alpha} = \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} - \begin{bmatrix} \alpha_{11} \\ \alpha_{22} \\ \alpha_{12} \end{bmatrix}$$

3) The matrix format of the plastic strain rate and the Melan-Prager evolution is provided as

$$\dot{\boldsymbol{\varepsilon}}^p = \dot{\lambda}\frac{\partial f}{\partial \boldsymbol{\sigma}}, \qquad \dot{\boldsymbol{\alpha}} = c\dot{\lambda}\boldsymbol{Q}\frac{\partial f}{\partial \boldsymbol{\sigma}},$$

where

$$\dot{\boldsymbol{\varepsilon}}^p = \begin{bmatrix} \dot{\varepsilon}^p_{11} \\ \dot{\varepsilon}^p_{22} \\ 2\dot{\varepsilon}^p_{12} \end{bmatrix}, \quad \boldsymbol{Q} = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1/2 \end{bmatrix}$$

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.

`alg_tan_stiff`

Purpose: Compute the algorithmic tangent stiffness matrix for a triangular 3 node element under plane stress conditions for a Melan-Prager kinemtatic hardening von Mises material.

Syntax: `Dats = alg_tan_stiff(sigma,alphad,Dstar,dlambda,mp)`

Description: `alg_tan_stiff` provides the algorithmic tangent stiffness matrix `Dats` for a triangular 3 node element. The in-plane stress is provided by `sigma` and `alphad` is the backstress provided as;

$$\texttt{sigma} = \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} \qquad \texttt{alphad} = \begin{bmatrix} \alpha_{11} \\ \alpha_{22} \\ \alpha_{12} \end{bmatrix}$$

`Dstar` is the linear elastic material tangent for plane stress, `dlambda` is the increment $\Delta\lambda$ and `mp` a vector containing the material parameters needed. The algorithmic tangent stiffness is given as

$$\boldsymbol{D}_{ats} = \boldsymbol{D}^a + \Big[c\Delta\lambda \boldsymbol{D}^a \frac{\partial^2 f}{\partial \boldsymbol{\sigma} \partial \boldsymbol{\sigma}} \boldsymbol{Q} \frac{\partial f}{\partial \boldsymbol{\sigma}} - \boldsymbol{D}^a \frac{\partial f}{\partial \boldsymbol{\sigma}} \Big] \frac{1}{A^a} \Big(\frac{\partial f}{\partial \boldsymbol{\sigma}} \Big)^T \boldsymbol{D}^a$$

$$\boldsymbol{D}^a = \Big((\boldsymbol{D}^*)^{-1} + \Delta\lambda \frac{\partial^2 f}{\partial \boldsymbol{\sigma} \partial \boldsymbol{\sigma}} \Big)^{-1}$$

$$A^a = c \Big(\frac{\partial f}{\partial \boldsymbol{\sigma}} \Big)^T \boldsymbol{Q} \frac{\partial f}{\partial \boldsymbol{\sigma}} + \Big(\frac{\partial f}{\partial \boldsymbol{\sigma}} \Big)^T \boldsymbol{D}^a \frac{\partial f}{\partial \boldsymbol{\sigma}} - c\Delta\lambda \Big(\frac{\partial f}{\partial \boldsymbol{\sigma}} \Big)^T \boldsymbol{D}^a \frac{\partial^2 f}{\partial \boldsymbol{\sigma} \partial \boldsymbol{\sigma}} \boldsymbol{Q} \frac{\partial f}{\partial \boldsymbol{\sigma}}$$

Note that you need to prove this in the report!

`update_variables`

Purpose: Check for elasto-plastic response and update variables accordingly for a triangular 3 node element under plane stress conditions for a Melan-Prager kinemtatic hardening von Mises material.

Syntax:

```
[sigma,alphad,dlambda] =
update_variables(sigma_old,alphad_old,Dstar,delta_eps,mp)
```

Description: `update_variables` provides updates of the in-plane stress `sigma`, the increment in plastic multiplier `dlambda` and the back-stress `alphad`. The variables are calculated from stress and back-stress at the last accepted equilibrium state `sigma_old` and `alphad_old`, respectively and the increment in strains between the last equilibrium state and the current state; `delta_eps`.

The increment $\Delta\lambda$ needed to update the stresses and strains are also computed and could be used as an indicator of plasticity later on in the code and will therefore also be used as output from this function.

Moreover `Dstar` denotes the linear elastic material tangent for plane stress and `mp` is a vector containing the material parameters needed.