Computational Inelasticity FHLN05

Assignment 2016

A non-linear elasto-plastic problem

General instructions

A written report should be submitted to the Division of Solid Mechanics no later than 1 November 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. 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 a basic knowledge in 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

A thin steel plate should be examined as it undergoes elasto-plastic deformation. The geometry of the plate is shown in figure 1.

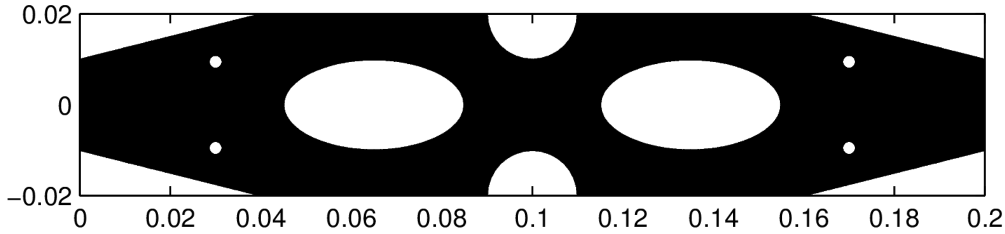


Figure 1: Geometry of the thin steel plate, dimensions in meter.

The structure have two symmetry axis; $x = 100$ mm and $y = 0$ mm, therefore only one quadrant of the profile is required for analysis. The small hole has a diameter of 3 mm and has its center at (30, 9.5) mm. The elliptic hole has its center at (65, 0) m and has the half-axis 20 mm along x and 10 mm along y. The half-circle has a radius of 10 mm and its center at $(100, 20)$ mm. The chamfering of the corner is 40 by 10 mm and the thickness of the profile is 1 mm.

In the development process of the plate, two different steel qualities are considered. Both qualities have the same elastic properties but they have different response to plastic deformation. The elastic modulus is $E = 200$ GPa and the Poisson's ratio is $\nu = 0.3$. In the elastic regime the material is considered linear and due to the small out of plane dimension plane stress is assumed, i.e, Hooke's law for plane stress can be used.

For the plastic loading, the materials can be modelled using von Mises yield surface with isotropic hardening, where associated plasticity can be assumed. The yield stress for the materials are given by the following expressions

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

$$\text{Material 2}: \quad \sigma_y = \sigma_{y0} + \alpha\sigma_{y0}(\varepsilon^p_{eff})^n \tag{2}$$

where the parameters $\sigma_{y0} = 250$ MPa, $K_\infty = 200$ MPa, $h = 20$ GPa, $\alpha = 15$, $n = 0.6$.

Assignment

The task is to calculate the elasto-plastic response of a given structure. 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_Assigmnent2016.m` may be used to obtain the topology matrices and Dirichlet boundary conditions `bc`. Figure 2 shows the part to be meshed using PDEtool in MATLAB.

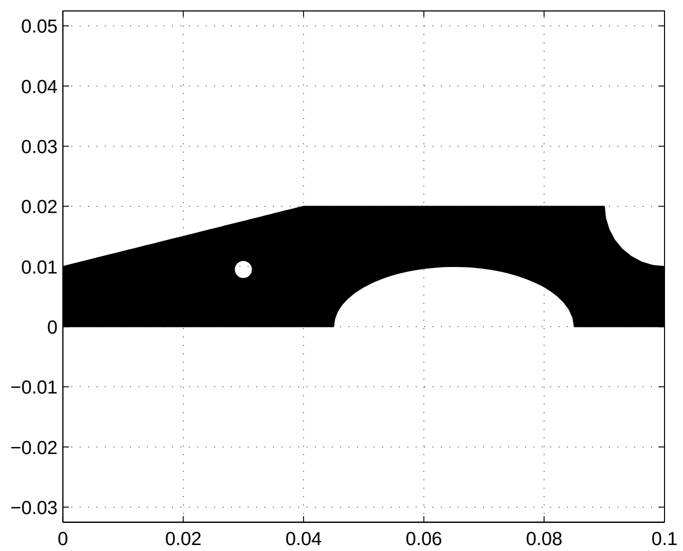


Figure 2: Geometry of the thin steel plate, dimensions in meter.

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 isotropic hardening of von Mises yield surface.
- Using a simple 2 element setup (illustrated in figure A.1 in appendix) plot the force-displacement curve for the different materials during a load-cycle where the material is plastically deformed (includes loading, unloading and re-loading).
- Investigate the elasto-plastic response of the steel profile by implementing a FE program using the Newton-Raphson algorithm with a fully implicit radial return method using displacement controled boundary conditions. This includes:
  - Implementation of the subroutines `update_variables1.m`, `update_variables2.m` that checks for elasto-plastic response and updates accordingly (a manual for the routines is appended). The number indicate which hardening model that is considered. The routines can be checked with data from `check_update.mat`.
  - Implementation of the subroutines `alg_tan_stiff1.m`, `alg_tan_stiff2.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.mat`.
- Use displacement controlled loading and load the structure well into the plastic region and then return to the original position i.e, at boundary displacement zero.
- The following results should be presented in an illustrative way:
  - A force-displacement curve for a load-cycle using the simple 2-element setup. The response for both materials should be presented.
  - The development of plastic response regions (maximum load, unloaded, and with one or more intermediate steps) for both materials.
  - The effective von Mises stress distribution at maximum load and after unloading, for both materials.

Remember that there are two different materials so the subroutines are slightly different because the materials have different hardening.

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.

Good luck!

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

4) 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.

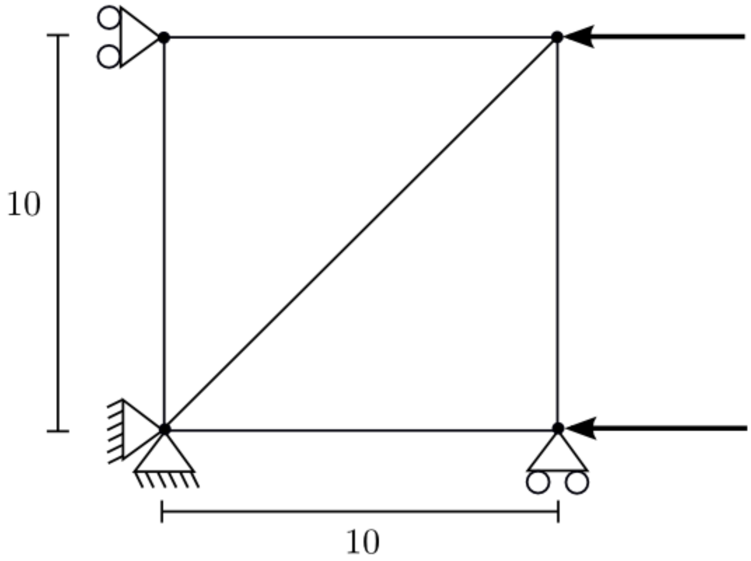
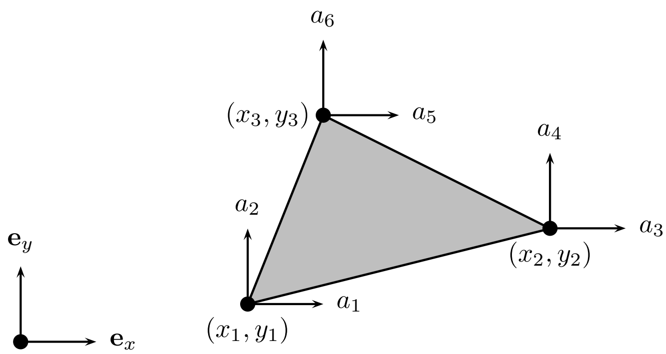


Figure A.1: Simple two element structure with boundary conditions and loaded nodes prescribed (thickness 1 mm). Dimensions in mm.

.

Purpose:

Compute the algorithmic tangent stiffness matrix for a triangular 3 node element under plane stress conditions.



Syntax:

Dats=alg_tan_stiff(sigma,Dstar,ep_eff,dlambda,mp)

Description:

alg_tan_stiff provides the algorithmic tangent stiffness matrix D_{ats} for a triangular 3 node element. The in-plane stresses are provided by sigma;

$$\mathsf{sigma} = \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix}$$

Dstar is the linear elastic material tangent for plane stress case. ep_eff is the effective plastic strains ε^p_{eff}, dlambda is the increment $\Delta\lambda$ and mp a vector containing the material parameters needed.

The algorithmic tangent stiffness is defined according to equation (18.64) in the course book;

$$\mathbf{D}_{ats} = \mathbf{D}^a - \frac{1}{A^a}\mathbf{D}^a \frac{\partial f}{\partial \boldsymbol{\sigma}} \left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{D}^a$$

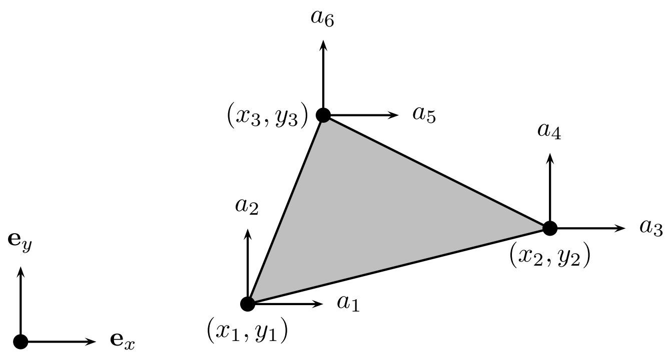
where

$$\mathbf{D}^a = \left(\mathbf{D}*^{-1} + \Delta\lambda \frac{\partial^2 f}{\partial \boldsymbol{\sigma} \partial \boldsymbol{\sigma}}\right)^{-1}, \quad A^a = \left(\frac{\partial f}{\partial \boldsymbol{\sigma}}\right)^T \mathbf{D}^a \frac{\partial f}{\partial \boldsymbol{\sigma}} - \frac{\partial f}{\partial K} d^a$$

$\mathbf{D}*$ denotes the linear elastic material tangent given by Dstar.

Purpose:

Check for elasto-plastic response in a triangular 3 node element under plane stress conditions and update variables accordingly.



Syntax:

```
[sigma,ep_eff,dlambda]=update_variables(sigmaEq,delta_eps,ep_effEq,Dstar,mp)
```

Description:

update_variables provides the following updates;

sigma - in-plane stresses $\boldsymbol{\sigma}^T = \begin{bmatrix}\sigma_{11} & \sigma_{22} & \sigma_{12}\end{bmatrix}$

dlambda - increment $\Delta\lambda$

ep_eff - effective plastic strain ε^p_{eff} using the radial return method for isotropic von Mises hardening plasticity.

The variables are computed with help of the stress and effective plastic strain from the last equilibrium state, sigmaEq and ep_effEq respectively and the increment in strains between the last equilibrium state and the current; delta_eps.

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

Moreover Dstar denotes the linear elastic material tangent and mp is a vector containing the material parameters.