Finite element method – Nonlinear systems
FHLN20 – 2017
Division of Solid Mechanics

Project 2 – General instructions

A written report including results/conclusions should be returned to the Division of Solid Mechanics no later than 15/1 10.00.

The assignment serves as part of the examination. A maximum of 20 points can be obtained. The task should be solved in groups of two or individually. If two persons work together they will obtain the same amount of points.

The assignment considers an analysis of the non-linear behaviour of a simple structure. To solve the problem Matlab should be used. In the toolbox Calfem, certain general FE-routines are already established and the task is to establish the extra routines needed to solve the non-linear boundary value problem.

The report should contain a description of the problem, the solution procedure that is needed as well as the results from the calculations in form of illustrative figures and tables. The program codes should be well commented and included in an Appendix.

When writing the text it can be assumed that the reader has 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 obtain all the relevant results just by reading through the report, i.e. without using the included program.

The report should be structured and give a professional description of the methods and the obtained results and be no longer than 20 pages (appendix excluded).

In the report, to every algorithm, a box should be included in the text, illustrating the used implementation.

Static and dynamic simulation of a contact problem

In this project the geometry in figure 1 will be studied under both static and dynamic analysis in form of a contact problem. All geometry data can be found in the file `input_data_2017`. The geometry is given in [m] and can be plotted with `eldraw2(ex,ey)`, the topology of the beam is provided by `edof`.

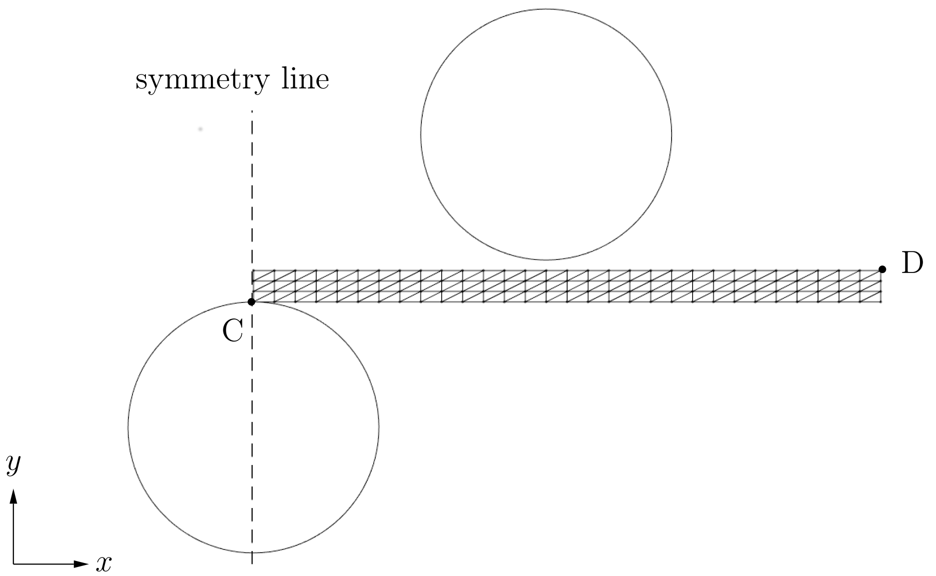

Figure 1: Undeformed structure, cylinder and symmetry line

The material is, in the three first parts of the project, assumed to be described by the following strain energy function:

$$U = \frac{1}{2}K\left(\frac{1}{2}(J^2-1)-\ln(J)\right)+\frac{1}{2}G(J^{-2/3}\mathrm{tr}(\mathbf{C})-3)$$

where K and G are the bulk and shear moduli respectively and can be obtained from the elastic modulus $E = 10$ GPa and Poisson's ration $\nu = 0.3$. $\mathbf{C}$ is the right Cauchy-Green tensor $\mathbf{C} = \mathbf{F}^T\mathbf{F}$ and $J = \det(\mathbf{F})$, where $\mathbf{F}$ is the deformation gradient. Plane strain conditions can be assumed, i.e. thickness can be put to 10 mm.

Static analysis

In the static numerical solution a total Lagrangian formulation should be used along with 3-node elements.

1) As a first approximation of the contact problem it is assumed that only one contact point exists between the cylinder and the bar, located at node C indicated in figure 1. The contact at node C is simply treated as an essential boundary condition, i.e. provided by `bc` in the program.

Write the script file `stat1.m` containing a force controlled Newton-Raphson scheme to solve the boundary value problem.

Load the structure such that the total force in node D is 10 [kN] in the negative y-direction. Plot the force vs. the y-displacement in node D (degree of freedom # 248), as well as the deformed shape of the structure. Show that the contact problem is not well represented by drawing the shape of the cylinder in the deformed plot.

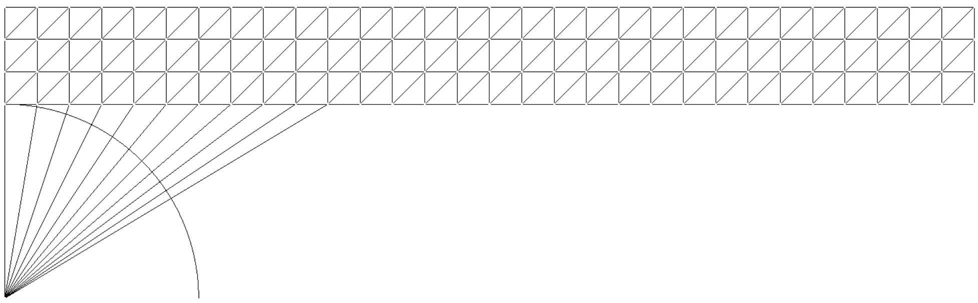

Figure 2: Location of bar elements connected to the lower cylinder.

2) In order to improve the approximation of the contact problem, nodal contact is introduced. The problem is a simplified model of the penalty method used in FE-codes. For this purpose the three dimensional bar elements defined in chapter 2 in Krenk [?], together with a special constitutive model will be introduced. The constitutive model, that replaces equation (2.20) in Krenk, is chosen as:

$$N = \begin{cases} \frac{k}{\Lambda}(\Lambda - \Lambda_c) & \text{if } \Lambda < \Lambda_c \\ 0 & \text{if } \Lambda \geq \Lambda_c \end{cases} \tag{1}$$

where

$$\Lambda = \sqrt{2\epsilon_G + 1} = \frac{l}{l_0}, \qquad \Lambda_c = \frac{r}{l_0} \tag{2}$$

and ϵ_G is the Green strain and Λ is the so called stretch. The length $r = 60$ [mm] is the radius of the cylinder as shown in figure 1. The force response F in the bars is illustrated in figure 4.

Write two functions, one calculating the normal force N according to equation 1, and one function that calculates the material stiffness $D = dN/d\epsilon_G$, as:

N = norfb(ec,ee,k,r) D=bstiff(ec,ee,k,r)

where the input arguments ec and ee are defined in the manual pages for the bar3g functions.

The bar elements are placed such that one end of the bar is located at the center of the cylinder and the other to nodes in the structure, as seen in figures 2 and 3. When the length of the bar becomes less then the radius of the cylinder a "contact force" is applied. The centers of the cylinders are (xc1,yc1) and (xc1,yc2).

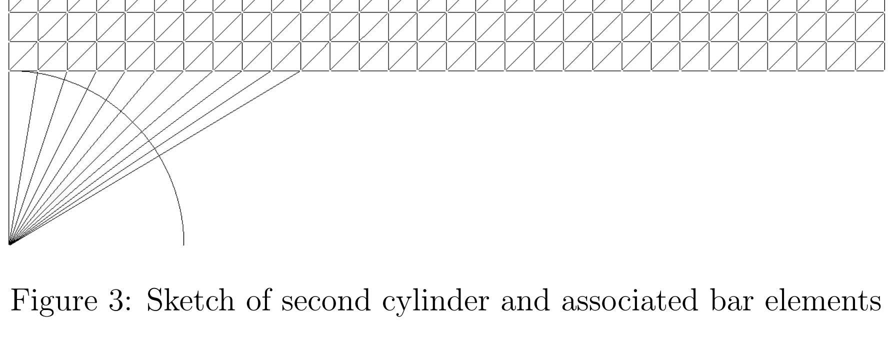

Figure 3: Sketch of second cylinder and associated bar elements

Write the script file stat2.m where the bar elements are introduced. Load the structure to the same load level as in task 1, but here apply the load first in negative y-direction then change to load direction and load to 10 [kN] in positive y-direction. Plot the applied force vs. the y-displacement in node D, as well as the deformed shape of the structure. Investigate different values of k in equation (1), begin with a small value and increase. In the geometry file ecb and edofb are the element coordinates and topology matrices for the bar elements. Note that ecb is a cell structure and coordinates for element i.e. is given as ecb{ie}.

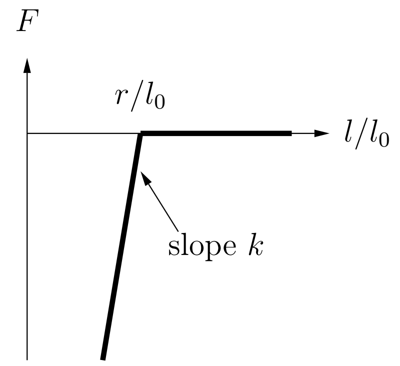

Figure 4: Force response of bars

Dynamic analysis

The same geometry as in the static analysis is considered. In the dynamic analysis the structure is assumed to start at the ending point of the static analysis, i.e. as a deformed structure. These variables are then loaded into the dynamic analysis and used as a starting point. The load is then released and the structure allowed to swing. In order to do this it is therefore required that you save appropriate variables from the second static analysis. The external force applied in the static analysis should be released using a ramp going from the maximum load to zero in 1 ms.

The initial density ρ is assumed to be 1700 kg/m^3. Note that if [mm] is used in the calculations a correct scaling of ρ is needed such that $\rho\ddot{u}V = [N]$, where V is the volume. The elastic properties are the same as in the static analysis.

1) Run the static analysis in task 2 again, but stop at maximum load in y-direction. The starting point for the dynamic analysis will then be when the beam it bent down at a load level of 10 [kN].

Save chosen values in a mat-file, use the command save in Matlab;

```
savefile='name of file'
save(savefile,'variable1','variable2','variable3','...')
```

Implement the Newmark algorithm such that the dynamic response of the structure can be analysed. Use the same non-linear material model as in the static analyses. Write a script file dynNewmark.m containing the Newmark algorithm. Choose suitable values for γ and β. More than one choice of γ and β should be evaluated, also in the calculation of the energies.

Run the analysis such that the beam will impact the upper cylinder and let it allow to swing up and down for some cycles. Plot the variation in energy during the analysis. Consider different time step lengths and comment upon the results obtained for these different values. For this purpose write a element function plan3gEd that calculates the kinetic and internal energy of the structure

```
[KinE, IntE] = plan3gEd(suitable arguments)
```

The specific format of the function should also be described in a manual page and included as an appendix in the report.

2) The *final* task consists of implementing an energy conserving dynamic algorithm. The same geometry and material data as considered previously is used here but the constitutive model is now the linear St. Venant-Kirchhoff, i.e. the strain energy is now given by

$$U = \frac{1}{2}\mathbf{E}^T\mathbf{D}\mathbf{E} \tag{3}$$

where $\mathbf{D}$ is the constant elasticity matrix for plane strain conditions and $\mathbf{E}$ the Green strain.

Run the static analysis in task 2 again, i.e. contact is considered. Save chosen values in a mat-file, use the command save in Matlab;

```
savefile='name of file'
save(savefile,'variable1','variable2','variable3','...')
```

As in the previous case release the load during a 1 ms ramp down to zero load. Modify the the routine for calculating the energies such that it corresponds the current model. Plot the energies during the simulation. As a final evaluation add the energies in the bar element to the total energy and plot the findings. Plot the variation of the energy during loading. Consider different time step lengths and comment upon the results obtained, especially during the impact phase.

A description of the the energy conserving dynamic algorithm can be found in Krenk, chapter 9 as well as in the lecture notes. Write the element function Ke=plan3Ege(suitable arguments) that calculates the tangential element stiffness matrix used in the algorithm. The specific format of the function should also be described in a manual page and included as an appendix in the report. Write a script file dynConser.m containing the energy conserving algorithm.

Krenk Steen, *Non-linear Modeling and Analysis of Solids and Structures*. Cambridge University Press, Cambridge, 2008.