

Finite element method – Nonlinear systems
FHL066
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 21/12 2013 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 analyzes the nonlinear behavior 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 nonlinear 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. The reader can also be assumed to have knowledge 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 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).

Nonlinear behavior of continuum

In the project two tasks will be considered, a static and a dynamic analysis.

Static analysis

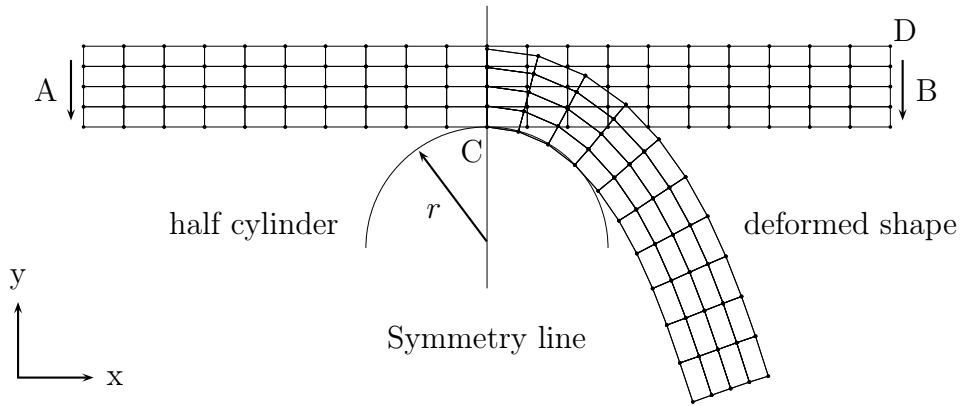


Figure 1: Contact problem

The geometry considered is given in Fig.1. Points A and B indicate the location of the loading.

The material is assumed to be described by the strain energy function

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

where K and G are the initial bulk and shear moduli, respectively, these can be obtained from the elastic modulus $E = 100\text{GPa}$ and Poisson's ratio $\nu = 0.3$. Moreover, $\mathbf{C} = \mathbf{F}^T \mathbf{F}$ where \mathbf{F} is the deformation gradient and $J = \det(\mathbf{F})$. For simplicity plane strain conditions are assumed, i.e. thickness can be set to 1mm. The radius of the cylinder is 3 mm.

In the static numerical solution procedures a total Lagrangian formulation should be used. 4-node isoparametric elements should be used in the analysis. Loading and boundary conditions are provided in the binary geometry files. Displacement loading is assumed and applied at node D and directed in negative y-direction. Symmetry is used in the calculations.

1) As a first approximation of the contact problem it is assumed that only a point contact exists at node C indicated in fig 1. The contact at C can be treated as an

essential boundary condition, i.e. provided by `bc` in the program. Write the script file `stat1.m` containing the Newton-Raphson scheme to solve the boundary value problem. The geometry file is given by `data2013_stat1.mat` and the geometry is given in [mm].

Load the structure such that the displacement in node D is 10mm in the negative y-direction. Plot the the force vs. the y-displacement in node D, as well as the deformed shape of the structure. To plot the undeformed and deformed geometries use the Caldem commands `eldraw2` and `eldisp2`. Show that contact problem is not well represented by drawing the shape of the cylinder in the deformed plot.

2) As a better approximation for 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 [1] with a special constitutive model will be used.

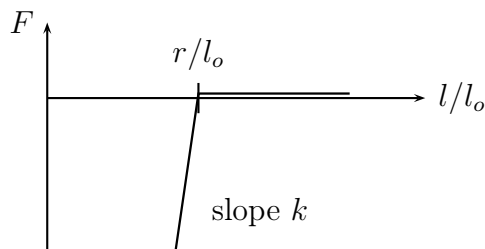
The constitutive model used is given by

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

where

$$\Lambda = \sqrt{2\epsilon_G + 1} = \frac{l}{l_o} \quad \Lambda_c = \frac{r}{l_o}$$

The strain ϵ_G is the usual Green strain and Λ is called stretch, i.e. equation (1) replaces equation (2.20) in Krenk (2009). The length r is the radius of the cylinder as shown in Fig. 1. The 'true' force F response of the bar will then be



Write two functions. The first one calculating the normal force N , as

$$N = \text{norfb}(ec, ee, k, r)$$

according to equation (1). The input arguments are defined in the manual pages for the bar3g element, the additional parameters are defined in equation (1).

The second function should calculate the material stiffness, i.e. $D = dN/d\epsilon_G$, by

$$D = \text{bstiff}(ec, ee, k, r)$$

The bar elements are now located such that one end is located at the center of cylinder, the other end is connected to a node in the structure. Then when the length of the bar becomes less than the radius of the cylinder a ‘contact force’ will be present. Six bar elements should be used, the configuration is shown in Fig.2

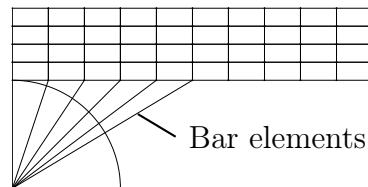


Figure 2: Location of the bar elements

Write the script file `stat2.m` where the bar elements are introduced. Load the structure such that the displacement at node D is 10mm in the negative y-direction. Plot the 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), try a small value then increase the value. The geometry file is given by `data2013_stat2.mat` in this file `ecb` and `edofb` are the element coordinates and topology matrices for the bar elements. Note `ecb` is a cell structure and coordinates for element ie is given as `ecb{ie}`.

Dynamic analysis

The geometry is shown in Fig.3 and the (binary file) `data2013_dynamic.mat` contains the geometry data. Note that the geometry is defined in [mm] and that 3 node elements are used here. It is assumed that the rotor can rotate around a fixed rotation point.

In the analysis it is assumed that the rotor already rotates, i.e. initial conditions exist. The initial conditions are given in the file `incond.m`. Note this initial

condition corresponds to a constrained situation where the length of the rotor is kept at its original length. This is released in the first step in the analysis and the dynamics of this situation is considered. To obtain the correct energy level, the first time step length must be chosen as $2 \cdot 10^{-5}$ s. No external loading exists.

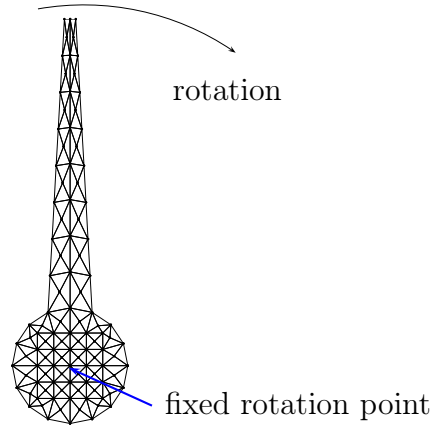


Figure 3: Geometry of rotor

The initial density is assumed to equal 1700kg/m^3 . Note that if [mm] is used in the calculations a correct scaling of ρ_o is needed such that $\rho_o \ddot{\mathbf{u}} V = [\text{N}]$, where V is the volume. The material response is here assumed to be described by the St. Venant-Kirchhoff model, i.e. the strain energy is given by

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

where \mathbf{D} is the constant elasticity matrix for plane strain conditions. The elastic properties are the same as in the static analysis.

1) Implementing the Newmark algorithm such that the dynamic properties of the structure shown in Fig. 3 can be analyzed. A description of the Newmark algorithm can be found in Krenk [1], Chapter 9.

Write a script file `dynNewmark.m` containing the Newmark algorithm. Choose suitable values for γ and β in the Newmark algorithm.

Plot the variation of the energy during rotation. Consider different time step lengths, comment upon the results obtained. Consider also different initial conditions, i.e. angular velocities. For this purpose the element function

$$[\text{KinE}, \text{IntE}] = \text{plan3gEd}(\text{suitable arguments})$$

should be written. The energies should be calculated for the specific material model. Kinetic energy as well as internal energy should be calculated. The specific format for the function should be described in a manual page. The manual page should be included as an Appendix in the report.

Let the structure "rotate" without any Dirichlet (essential) boundary conditions. Also here, consider different time step lengths. What will happen with the system?

2) The *final* task consists of implementing an energy conserving dynamic algorithm. The same geometry and material data as considered previously will be used here. A description of the energy conserving dynamic algorithm can be found in Krenk [1], Chapter 9.

The element function

`Ke=plan3Ege(suitable arguments)`

calculating the tangential element stiffness matrix used in the energy conserving algorithm should be written. The specific format for the function should be described in a manual page. The manual page should be included as an Appendix in the report. The element commands `plan3gs` and `plan3gf` should be used with suitable input arguments.

Write a script file `dynConser.m` containing the energy conserving dynamic algorithm.

Plot the variation of the energy during loading. Consider different time step lengths, comment upon the results obtained.

Let the structure rotate without any Dirichlet (essential) boundary conditions. Also here, consider different time step lengths and comment upon the obtained response.

References

[1] S. Krenk, (2008), 'Non-Linear Modeling and Analysis of Structures and Solids', lecture notes.

Hints

To speed up the program you should instead of `zeros` use `sparse` to initialize matrices.

The `assem` command in Calfem is very slow, a slightly faster algorithm is obtained by using

```
nd=edof(element,2:nrdof_element)
```

```
K(nd,nd)=K(nd,nd)+Ke
```