Finite element method – Nonlinear systems
FHLN20 - 2018
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 21/1, 10.00 (2019).

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.

Nonlinear behavior of continuum

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

Static analysis

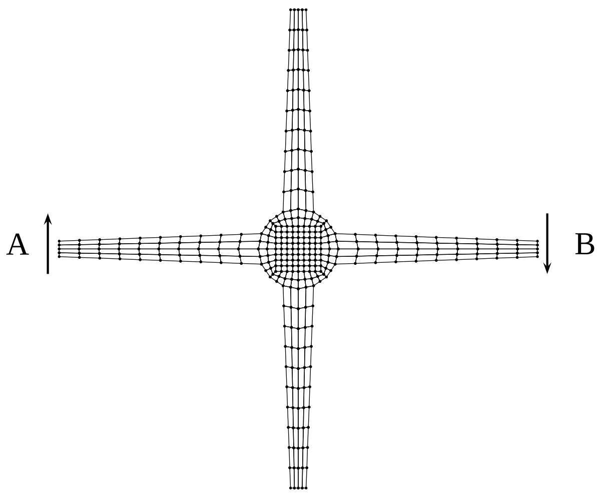

Figure 1: Fan

The geometry considered is given in Fig.1. Point A and B indicate the location of the applied force F whereas C and D indicate the location of the where the essential boundary conditions are applied, i.e. zero displacements. Nodes at location D are allowed to move in horizontal direction. The radius of the fan is 300mm and thickness is 1mm. The geometry is given in the (binary) file data2018_static.mat where coordinates are given in millimeters.

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

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

where K and G are the initial bulk and shear moduli, respectively, these can be obtained from the elastic modulus $E = 1$GPa and Poisson's ratio $\nu = 0.3$ (Corresponding to a polymer). For simplicity plane strain conditions are assumed.

In the static numerical solution procedure a total Lagrangian formulation should be used. 4-node isoparametric elements should be used in the analysis. The loading and boundary conditions are given in the file data2018_static.mat as bc and f, respectively. The fan should be loaded well into the deformed stated.

Write a function calculating the internal energy for an element

```
Energy=plan4gEs(ec,t,ed,ep).
```

The arguments are only included for illustration purposes, you should defined these by yourself.

1) Write the script file statTotal.m containing the Newton-Raphson scheme that use a total Lagrangian algorithm.

2) Plot the total internal energy vs. the applied force, as well as the deformed shape of the fan. To plot the undeformed and deformed geometries use the Calfem commands eldraw2 and eldisp2.

Dynamic analysis

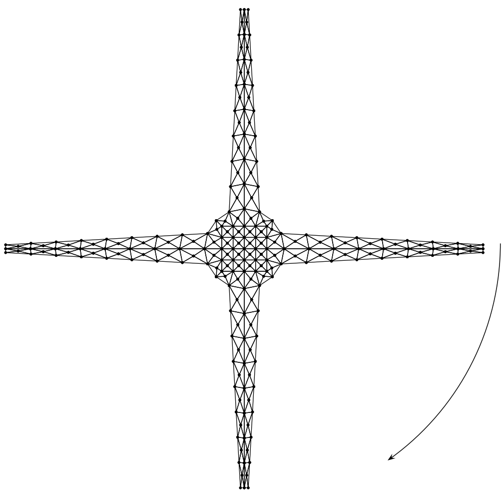

Figure 2: Geometry

Task 1

The *first* task consists of implementing the Newmark algorithm such that the dynamic properties of the structure shown in Fig. 2 can be analyzed. A description of the Newmark algorithm can be found in Lecture notes [1] and Steen Krenk [2], Chapter 9. The initial density is assumed to equal 1700kg/m^3. The material response is here assumed to be described by the St. Venant-Kirchhoff model. The material parameter are the same as in the static analysis.

The geometry is given in the (binary file) data2018_dynamic.mat and for simplicity the thickness is set to 1m. Note that the geometry is defined in meters and that 3 node elements are used here. Essential boundary conditions are applied on the center node, which makes the fan to rotate around its center point.

In the analysis it is assumed that the fan already rotates, i.e. initial conditions exist. The initial conditions are given in the file incond_2018.m. Note that this initial condition corresponds to a constrained situation where the lengths of the fins are kept at their original lengths. 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 $1 \cdot 10^{-4}$s. No external loading exists.

The element function

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

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.

1) Write a script file dynNewmark.m containing the Newmark algorithm. Consider different values for γ and β in the Newmark algorithm. The values $\gamma = 1/2$ and $\beta = 1/4$.

2) Plot the variation of the energies, total energy, kinetic energy and internal energy, during loading for time step length $1 \cdot 10^{-4}$. Consider two different values for γ and β in the Newmark algorithm. One that will have numerical damping as well as the values $\gamma = 1/2$ and $\beta = 1/4$.

3) Consider different time step lengths, comment upon the results obtained.

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

Task 2

The *second* 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 the Lecture notes [1] and Steen Krenk [2], 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.

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

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

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

4) Compare the results obtained from task 2 with those from task 1.

References

[1] M. Ristinmaa, (2018), Introduction to Non-linear Finite Element Method", Div. of Solid Mechanics, Lund University.

[1] S. Krenk, (2008), ’Non-Linear Modelling 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 faster algorithm is obtained by using

```
nd=edof(element,2:nrdof_element)
K(nd,nd)=K(nd,nd)+Ke
```