Finite element method – Nonlinear systems
FHL066 2016
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 16/1 2017, 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 the analysis of the nonlinear behavior of a simple structure. The task involves static and dynamic analysis as well as contact. To solve the problems 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 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. The reader should be able to obtain all the relevant results 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 of a frame

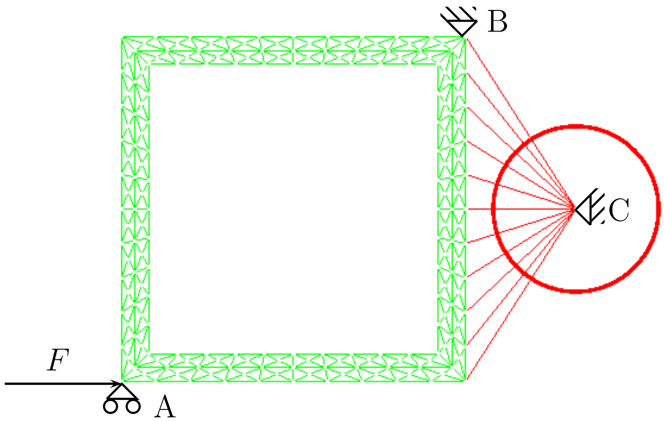


Figure 1: Loading and boundary conditions for the contact problem

The first task is to solve the boundary value problem shown in Fig 1.

The material of the frame structure is assumed to be described by the St. Venant-Kirchhoff model with the elastic modulus $E = 10$GPa and Poisson's ratio $\nu = 0.3$. For simplicity plane strain conditions are assumed, the thickness can be set to 0.04m.

A horisontal force is applied at point A. This will make the frame come into contact with a rigid cylinder of radius 0.12 m. The contact problem is addressed using a simplified version of the penalty method commonly 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 bar elements are positioned such that one end of the bar is located at the center of the cylinder, the other end is connected to a node in the frame structure, as shown in Fig. 1. When the length of the bar l becomes less than the radius of the cylinder r_c a contact force will be present. This is described by the constitutive law (1).

The constitutive model for the bar elements is given by

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

where

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

The strain ϵ_G is the usual Green strain and Λ is the stretch, i.e. equation (1) replaces equation (2.20) in Krenk (2009). The length r_c is the radius of the cylinder. A suitable value for k should be determined, such that the penetration into the cylinder becomes small.

In the numerical solution procedure a total Lagrangian formulation should be used.

The following task should be solved:

- Write two functions for the bar elements. The first one calculating the normal force N, as

  $$N = \texttt{norfb(ec,ee,k,rc)} \tag{3}$$

  according to equation (1). The second function should calculate the material stiffness, i.e $D = dN/d\epsilon_G$ as

  $$D = \texttt{stiffb(ec,ee,k,rc)} \tag{4}$$

  The input arguments are `ec`; the element nodal coordinates in the undeformed configuration, `ee`; the Green strain of the bar, `k`; the penalty parameter defined in (1) and `rc`; the radius of the rigid cylinder.

- Solve the boundary value problem considering contact between the rigid cylinder and the deformable frame. Plot the deformation of the structure and the internal force vs displacement of node A. Load the frame until the displacment of node A is at least 0.35 m.

  The material file `static_contact.mat` includes topology data for the problem. Quantities with the ending `B` are related to the bars e.g, `edofB, exB, eyB`.

Dynamic analysis of a swinging frame

The same geometry as in the static analysis is used for the dynamic problem. However, the boundary conditions are changed and the rigid cylinder is removed. Initially the structure is preloaded to the load level 60kN as shown in Fig 2a). After this the loading and the boundary conditions in point B are released and the structure is allowed to swing. The boundary conditions should be released using a ramp going from the maximum load to zero in 1 millisecond. The response of the frame at a some timestep is shown in Fig.2b).

The initial density is equal 1700kg/m^3. The material model is the same as in the static analysis, i.e. the strain energy is given by

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

where $\boldsymbol{D}$ is the constant elasticity matrix for plane strain conditions.

For the dynamic analysis the following tasks should be solved:

- Run the static analysis, but without the rigid cylinder, such that the load level 60kN is reached. Topology data without the cylinder is found in the file `static.mat`. The deformed frame should look similar to Fig. 2a. These results will be the initial condition for the dynamic analysis.
- Implement the Newmark algorithm so that the dynamic properties of the structure can be analysed. Choose suitable values for γ and β in the Newmark algorithm.

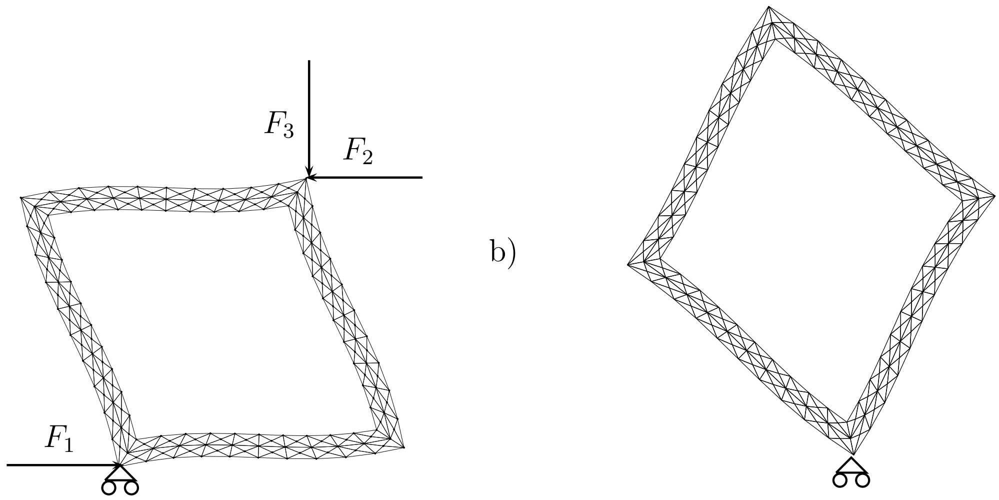
a)  b)

Figure 2: Preloaded frame in a) and a swinging frame in b) at some time instance.

- Plot the variation of the energy with time. Consider different time step lengths and comment upon the results obtained. For this purpose the element function

  [KinE, IntE]=plan3gEn(suitable arguments)

  calculating the kinetic and internal energies of a three node element should be written. The energies should be calculated for the specific material model. 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 *final* task consists of implementing an energy conserving dynamic algorithm. The same geometry and material data as considered previously will be used. 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.

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

References

[1] S. Krenk, (2009), 'Non-Linear Modelling and Analysis of Solid and Structures', Cambridge.

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
```

To draw circles the Matlab command: `viscircles(CENTERS, RADII)` can be used.