

Finite element method – Nonlinear systems
FHL066 – 2015
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 2016 18/1, 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 nonlinear behavior of three simple structures. 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 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).

Static analysis of sheet pressing

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

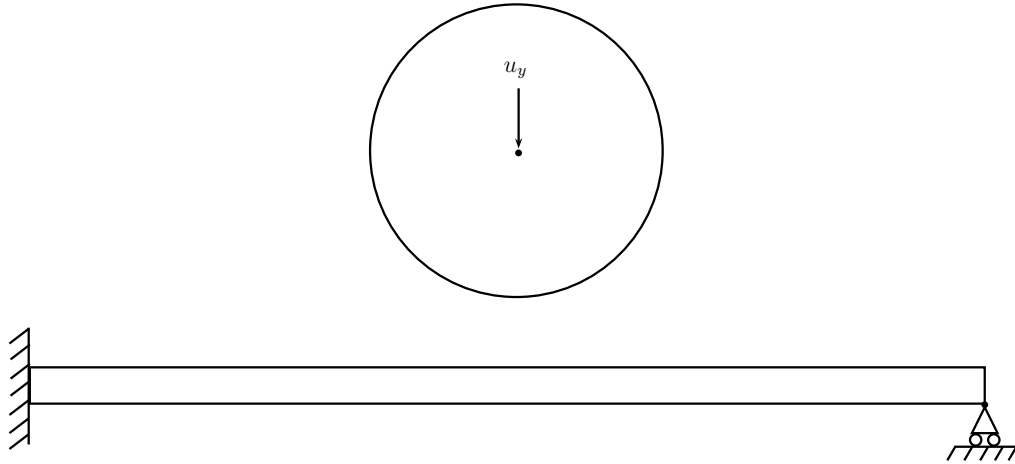


Figure 1: A sheet deformed by displacing a rigid cylinder.

The material is assumed to be described by the strain energy function (Neo-Hooke)

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

where K and G are the initial bulk and shear moduli, respectively, these can be obtained from the elastic modulus $E = 10$ MPa and Poissons ratio $\nu = 0.35$. 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 0.1 m. The radius of the cylinder is 0.03 m.

In the static numerical solution procedures a total Lagrangian formulation should be used. The geometry, loading, 4-node isoparametric mesh and boundary conditions are provided in the material file `static2015_1.mat`.

The rigid cylinder is assumed to be displaced downwards (negative y-direction) such that the cylinder comes in contact with the sheet and contact forces cause the sheet to deform.

- Solve the boundary value problem using a static algorithm without introducing the cylinder by writing the script `staticNR_force.m`. Instead of the cylinder introduce a downward force in `fDof` and plot the deformation of the structure as well as the force vs displacement of `fDof`.
- Solve the boundary value problem considering contact between the rigid cylinder and the deformable sheet by writing a script `staticNR_contact.m`. Plot the deformation of the structure and the internal force vs displacement of the node in the center of the cylinder. Load the structure until a displacement of the rigid cylinder is $u_y = 55$ mm. The contact problem is addressed with a simplified model 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. Fig 2 shows how the 'contact' bars are connected to the deformable body.

The constitutive model for the bar elements is given by

$$N = \begin{cases} \frac{k}{\Lambda}(\Lambda - \Lambda_c) & \text{if } \Lambda < \Lambda_c \\ 0 & \text{if } \Lambda \geq \Lambda_c \end{cases} \quad (2)$$

where

$$\Lambda = \sqrt{2\epsilon_G + 1} = \frac{l}{l_0}, \quad \Lambda_c = \frac{r}{l_0}. \quad (3)$$

The strain ϵ_G is the usual Green strain and Λ is the stretch, i.e. equation (2) replaces equation (2.20) in Krenk (2009). The length r is the radius of the cylinder as shown in Fig 1. A suitable value for k should be determined, such that the penetrations through the cylinder becomes small.

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

$$N = \text{norfb}(\mathbf{ec}, \mathbf{ee}, \mathbf{k}, \mathbf{r}) \quad (4)$$

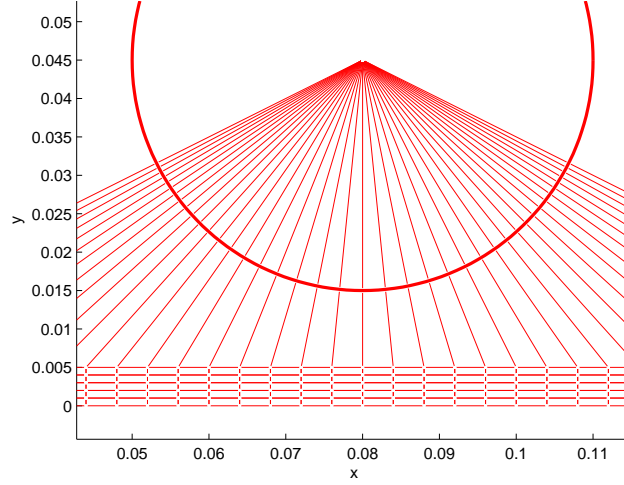


Figure 2: The contact bars connected to the nodes of the deformable mesh (dimensions in meter).

according to equation (2). The second function should calculate the material stiffness, i.e $D = dN/d\epsilon_G$ by

$$D = \text{bstiff}(\mathbf{ec}, \mathbf{ee}, \mathbf{k}, \mathbf{r}) \quad (5)$$

The input arguments are \mathbf{ec} are the element nodal coordinates in the undeformed configuration, \mathbf{ee} is the Green strain of the bar, \mathbf{k} is the penalty parameter defined in (3) and \mathbf{r} is the radius of the rigid cylinder. The bar elements are now positioned 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 l becomes less than the radius of the cylinder r a contact force will be present. This is described by the constitutive law (2). The material file `static2015_2.mat` includes topology data for the problem, quantities with \mathbf{B} are related to the bars e.g, `edofB`, `exB`, `eyB`, `nelmB`.

Dynamic analysis of a rotor

The geometry is shown in Fig. 3 and the file `2015_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 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_rot.m`. This is released in the first step in the analysis and the dynamics of this situation is considered. No external loading exists.

The initial density is assumed to equal 1700kg/m^3 . Note that if `[mm]` is used in the calculations a correct scaling of ρ_0 is needed such that $\rho_0 \ddot{\mathbf{u}} V = [N]$, 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:

$$U = \frac{1}{2} \mathbf{E}^T \mathbf{D} \mathbf{E} \quad (6)$$

where \mathbf{D} is the constant elasticity matrix for plane strain conditions. The Young's modulus is set to $E = 5 \text{ GPa}$ and $\nu = 0.3$.

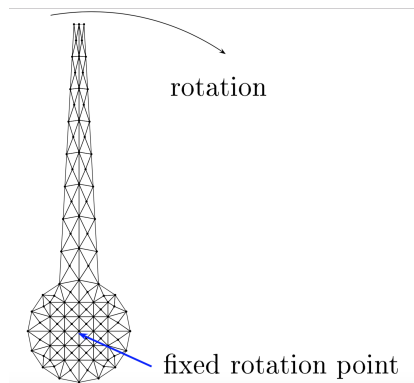


Figure 3: Geometry of rotor

- Implement the Newmark algorithm such that the dynamic properties of the structure shown in Fig. 3 can be analyzed. Write the script

file `dynNewmarkRotor.m`. Choose suitable values for γ and β in the Newmark algorithm.

- Plot the variation of the energy during the process. For this purpose the element function

$$[\text{KinE}, \text{IntE}] = \text{plan3gEd}(\text{ suitable arguments}) \quad (7)$$

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. What will happen with the system?

Dynamic analysis of a box hitting the floor

A dynamic analysis of a falling box is considered. The initial height over the floor is 0.2 m (the floor has the coordinate $y = -0.36\text{m}$). To simulate different drop heights the initial velocity may be changed. The geometry of the box is seen in Fig 4. The thickness can be set to 0.1 m.

It is assumed that the box has some initial velocity and rotation. Furthermore the box is accelerated by a body force due to gravity, \mathbf{f}_b .

The density of the box is assumed to equal 1000 kg/m^3 . The material response is here assumed to be described by a Neo-Hooke material with , i.e., the same strain energy as equation (1). The same material parameters as in the static problem are used.

The contact with the floor will be modeled using a node-based penalty formulation. This is introduced by including a force vector \mathbf{f}_{cont} to the residual, i.e.

$$\mathbf{r} = \mathbf{f}_{int} + \mathbf{M}\ddot{\mathbf{u}} - \mathbf{f}_b - \mathbf{f}_{cont} \quad (8)$$

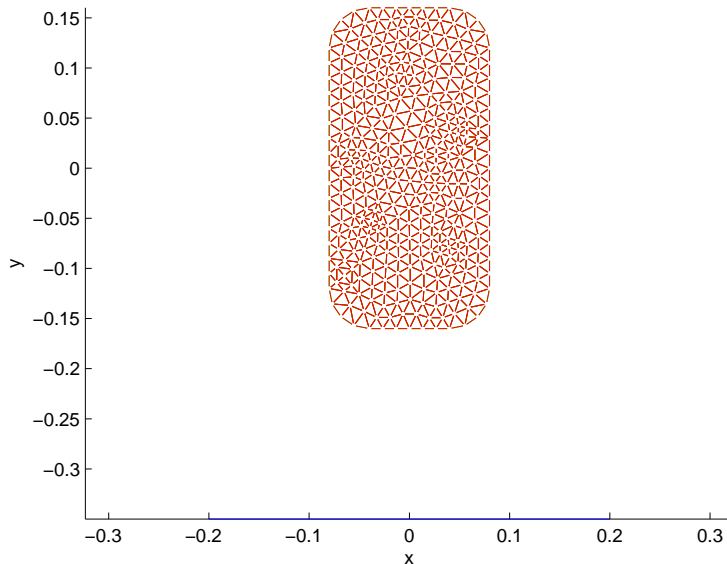


Figure 4: Set up for the dynamic analysis of a box hitting the floor (dimensions in meter).

The contact force vector is assumed to be given by

$$\mathbf{f}_{cont} = \sum_{i=1}^{n_c} \bar{\epsilon}_N \bar{g}_{Ni} \mathbf{n}_i^w \quad (9)$$

where \mathbf{n}^w is the normal vector to the floor, $\bar{\epsilon}_N < 0$ is a penalty parameter and \bar{g}_{Ni} is the penetration function, defined by

$$\bar{g}_{Ni} = \begin{cases} (\mathbf{x}^i - \mathbf{x}^w)^T \mathbf{n}^w & \text{if } (\mathbf{x}^i - \mathbf{x}^w)^T \mathbf{n}^w < 0 \\ 0 & \text{otherwise} \end{cases} \quad (10)$$

where \mathbf{x}^i is the nodal position along the boundary of the box and \mathbf{x}^w is the closet point projection, i.e. the point on the floor which gives the smallest distance to the node \mathbf{x}^i . Note that a parametrization of the floor can be written as $\mathbf{x}_t^w = (s, 0)$ where $s \in \mathbb{R}$. The linearisation of the contact force vector \mathbf{f}_{cont} provides the stiffness matrix given by

$$\mathbf{K}_{cont} = \sum_{i=1}^{n_c} \bar{\epsilon}_N \mathbf{n}^w (\mathbf{n}^w)^T \quad (11)$$

- Solve the dynamic boundary value problem using the Newmark method. Write a script file `dynNewmarkBox.m` with the Newmark parameters $\gamma = 0.5$ and $\beta = 0.25$ and find a suitable value for $\bar{\epsilon}_N$ (less than 0) in the Newmark algorithm which gives a convergent solution. Plot the motion of the box before hitting the ground and during impact. The geometry of box is provided in `dynamic2015_box.m` (quantities with **S** are associated with the surface).

The function `(v0,a0)=incon2015Box(omega,Vx,Vy,Coord)` generate initial velocity (m/s) and rotation around the center of mass. `omega` is the counter-clockwise angular velocity in revolutions per second.

- Vary some parameters in the model, such as the initial velocity, elastic modulus and/or penalty parameter. And comment upon the what happens at impact and why.

Note: the contact introduces a non-smoothness of the problem and therefore it is not expected that the problem have quadratic convergence at during impact. Also the Newmark algorithm is sensitive to the time step length, therefore small time increments during impact is recommended.

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
```

The Calfem command: `[empty,fbe] = plante(ex(e1,:),ey(e1,:),[1,th], zeros(3),[0;-rho*g])` can be used to calculate the elementwise body forces.

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