Final Project

Due: 2024 Jan. 18

The grading of the final project is based on the code development displayed in your github repository (50 points) and your submitted report (50 points).

Code development:

1. (30 pts) Consider a plane elasticity problem defined on a unit square domain $\Omega = [0,1] \times [0,1]$. Implement a finite element code using bilinear quadrilateral element. You may assume that the mesh is uniform in each direction. In doing so, you may generate the mesh (including the nodal coordinates, the IEN connectivity, etc.) straightforwardly. Regarding the boundary conditions, there are four edges of the square (i.e., the top, bottom, left, and right edges). You need to allow the specification of either displacement (Dirichlet) or traction (Neumann) boundary condition on these edges in your code. Regarding the model, you may allow the users to specify either plane strain or plane stress for the two-dimensional analysis. The material is homogeneous isotropic elastic.

2. (20 pts) Modify your statics code into an elastodynamics code using the Newmark-β scheme. You do not need to account for the viscous damping term.

You are asked to report the following in your final project report.

1. (5 pts) State the strong-, weak-, and Galerkin formulation of the problem.

2. (5 pts) Discuss your chosen implementation of the element stiffness. You may need to consult Sec. 3.10 of the textbook for different implementations.

3. (15 pts) Verify the implementation of the static code using the manufactured solution method. You need to design your own manufactured solution. Report the errors and convergence rates measured in the L2 and H1 norms, respectively.

4. (15 pts) Verify the implementation of the dynamics code using the manufactured solution method for the trapezoidal rule (i.e. $\beta = \frac{1}{4}$ and $\gamma = \frac{1}{2}$). Hint: the spatiotemporal error behaves as $\mathcal{O}(\Delta x^p + \Delta t^q)$. To rule out the pollution caused by the spatial error, you may want to use a spatial mesh that is fine enough, so that the

Δx^p part is significantly smaller than the Δt^q part.

5. (10 pts) Consider a cantilever beam with length L=10m and width c=1m (see Fig. 1). Its left edge is fixed, and the rest boundaries are imposed with traction-free boundary conditions. We ignore the body force. The initial displacement is zero and the initial velocity is given as

$$\boldsymbol{v}(x_1, x_2, 0) = \tilde{v} \begin{pmatrix} 0 \\ x_1 \end{pmatrix},$$

where $\tilde{v} = 1\mathrm{s}^{-1}$. The Young's modulus is 1 kPa, Poisson's ratio is 0.3, and density is $1\,\mathrm{kg/m^3}$. Monitor the tip displacement and velocity up to $T = 10\mathrm{s}$. Compare and comment on the stability of the trapezoidal rule and the central difference rule (i.e. $\beta = 0$ and $\gamma = \frac{1}{2}$) with respect to the time step size.

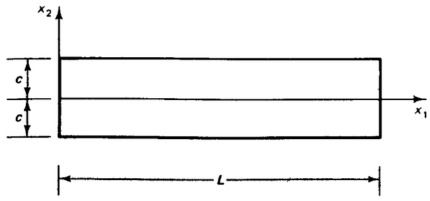

Fig. 1 Geometrical setting of cantilever beam

Hint: The trapezoidal rule is an unconditionally stable algorithm which does not pose restriction on the time step size. The central difference rule is an explicit scheme if the mass matrix is lumped to be diagonal. Its stability is conditional, meaning its calculation is stable if the time step size is less than or equal to Δt_{crit}, a critical time step size.

6. (bonus 5 pts) Determine the value of Δt_{crit} for your central difference rule. Hint: it depends on your finite element discretization.

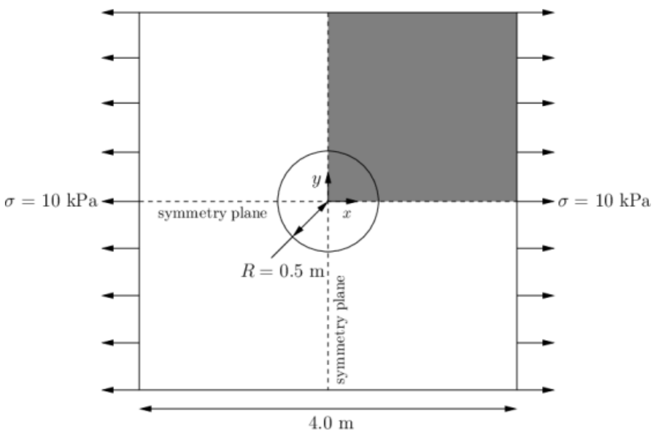

Fig. 2 Geometrical setting of a square plate with a hole.

7. (bonus 25 pts) The stress analysis for structures with holes can be common in

engineering practice. Consider the following plane elastic body with side length 4m and a hole in the center (see Fig. 2). The hole has radius 0.5m. It is loaded with a uniform traction of 10 kPa over its left and right faces. There is no body force. Due to the symmetry of the problem, only a quarter of the domain needs to be analyzed, which is shown as the dark area in the following figure. Over the symmetry plane, the symmetry boundary condition needs to be specified. The Young's modulus is 10^9 Pa and Poisson's ratio is 0.3. Use your own code to perform a plane stress analysis of this problem. You need to generate a mesh for this domain and refer to Fig. 3 for a representative mesh.

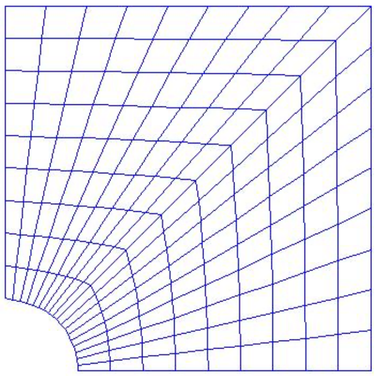

Fig. 3 A representative mesh for the domain.