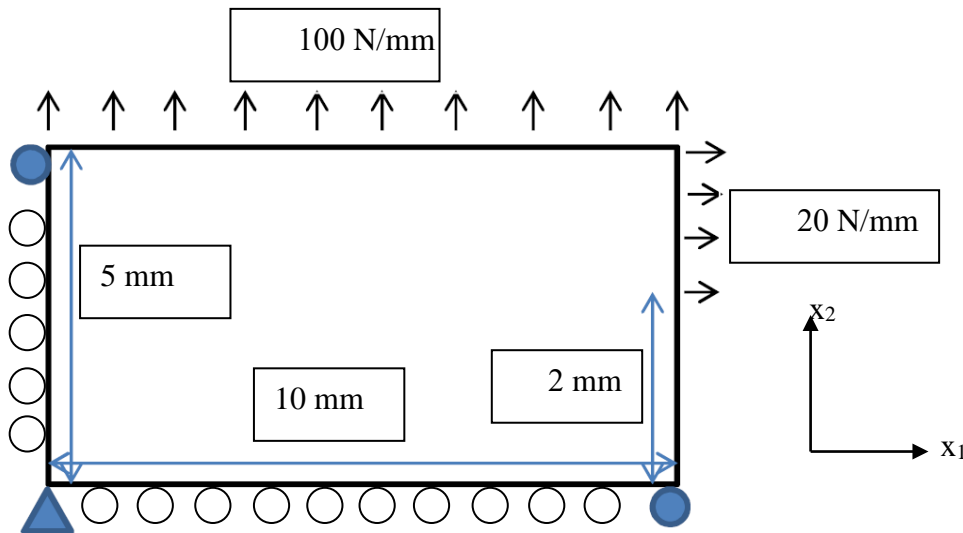


# AE420/ME471/CSE453 – Introduction to the finite element method Coding Project

**Upload to Canvas by 11.59 pm 12-4-2024**

You have been introduced the various steps of FEM modeling in class (i.e. shape function interpolation, mapping parent to global element, numerical integration, matrix assembly, applying boundary conditions, etc.). The goal of this HW problem is to familiarize you with the algorithm that goes into software packages like Abaqus, and Ansys. The licenses of these packages typically cost around \$60k per year. For a simple structural problem (or any problem in general), you can write your own finite element code instead.

## Problem 7.1: FEM Coding (10 points)



Consider the above 2D plane strain linear elastic structural problem. The left side has roller boundary conditions allowing the structure to move only in the  $x_2$  direction, while the bottom side has roller boundary conditions allowing the structure to move only in the  $x_1$  direction. Assuming an elastic modulus of 1 GPa and poisson's ratio of 0.3, and that the distributed loads are applied across a thickness of 5 mm, write an FEM code in any programming language you wish to solve this problem (recommend the problem be coded in MATLAB). (10 points)

Your code should:

- use four noded quadrilateral plane-strain elements with the mesh as outlined below,
- read in the nodal coordinates and element connectivity,
- formulate the shape functions, stiffness matrix, and residual force vector accounting for local-global transformation and invoking gauss quadrature for the integration,
- solve for the nodal displacements,
- find the forces at the imposed boundary nodes, and
- compute the  $\sigma_{11}$ ,  $\sigma_{22}$  stresses at the four integration points of element A.

**For those doing 4 credits, please complete this additional task (additional 5 points)**

Validate your results in (e) and (f) by solving the same problem in Abaqus. Do you expect the results of your coding to be the same as Abaqus? If the results do not happen to be the same, can you explain why?

Hint: To create this mesh in Abaqus, it would be easier to run the .inp file using Abaqus command prompt rather than creating this in .CAE.

