Seoul National University
Department of Mechanical & Aerospace Engineering
Computational Structural Analysis

Prof. Gunjin Yun
M2795.008300
Spring 2019

Computational Structural Analysis

Programming Project II

Due June 17 (Monday)

The goal of the second FE programming assignment is to complete full finite element procedures to analyze a structural mechanics problem. The educational MATLAB codes are provided to you. The computer program is for two-dimensional stress analyses using the Q4 finite element. Note that the program is incomplete. You are supposed to complete the MATLAB codes, 1) a MATLAB function to compute elemental work-equivalent load vectors and assemble them to a global force vector and 2) a MATLAB function that computes stresses and strains for visualizations. E=2.05E11 (Pa), Poisson ratio=0.29, thickness =0.01 (m), h=0.3 (m), w=0.9(m), a=0.05 (m), and σ_0=10 KPa. Using the symmetry condition, make a quadrant model and apply symmetric boundary conditions.

(a)

(b)
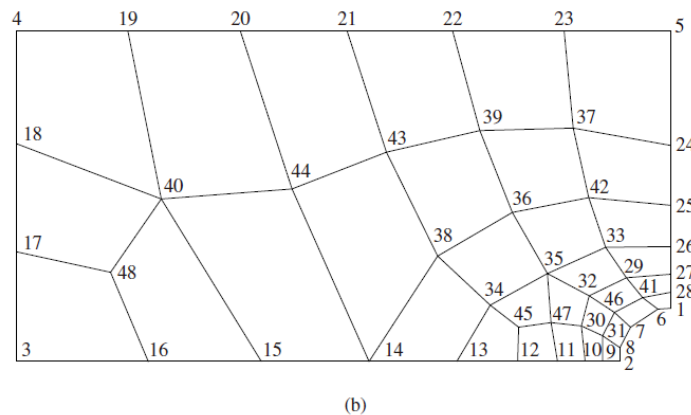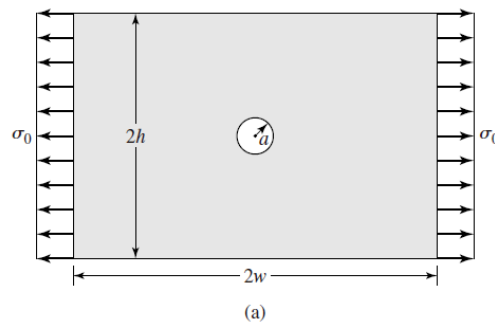
1. Main_FEProgram.m: Main FE program (Incomplete)
2. ScanInput.m: Read Finite Element Model Data
3. GetIDArray.m: Get ID Array
4. SetGaussQ.m: Set Gauss Quadrature Constants

5. GetElemStiff.m: Compute Elemental Stiffness Matrices
6. GetGPV.m: Get Sampling Point Locations and Associated Weight Factor
7. GetSFD.m: Compute Shape Function Derivatives
8. GetCoord.m: Get Coordinate Values of the Elements
9. GetDJacob.m: Get Jacobian Matrix and Determinant of Jacobian Matrix
10. GetBMatrix.m: Get [B] matrix
11. GetDLoad.m: Compute Consistent Load Vector (Incomplete)
12. GetStrsStrn.m: Compute Stresses and Strains at Gauss Points and Print out the Results (Incomplete)
13. Plot_result.m: Visualize displacements, stresses and strains.

Guidelines)
- Study the program provided to you and try to gain a clear understanding of the overall FEM procedures, and the logics behind each of the routines.
- Generate your FE input data file after studying the ScanInput.m program.
- Complete the following function files: Main_FEProgram.m, GetDLoad.m, and GetStrsStrn.m
- Create three different FE meshes with different number of elements (coarse, intermediate and fine meshes). For this use the ABAQUS CAE student edition and export FE data so that they can be used in the educational MATLAB FE codes.
- Run your Main_FEProgram.m program and print the stresses and the displacements for all the elements.
- You should submit your complete MATLAB codes.
- Load input data need Element, Surface direction, Magnitude
  e.g. [1, 2, 10] means Pressure applied 2nd surface of 1st element.
- Below table show data format for Visualization.

Element	Node	Value
1	1	U1
1	2	U2
1	3	U3
1	4	U4
2	3	U3
2	4	U4
2	5	U5
2	6	U6

Submit the followings
Q1) A graph showing nodal displacements on the right-end vertical edge for three different meshes
Q2) A graph showing nodal stresses (σ_{xx}) on the right-end vertical edge for three different meshes
Q3) Contour plots of stress components (σ_{xx}, σ_{yy}, and σ_{xy}) for three different meshes
Q4) Contour plots of strain components ($\epsilon_{xx}, \epsilon_{yy}$, and ϵ_{xy}) for three different meshes
Q5) Comparisons the above results with those from ABAQUS