Finite Element Analysis I 4300:609

Homework #6
Due is October 29 (Thursday)

Programming Assignment I: Stiffness Matrix of 2D Plane Stress Element using Isoparametric Formulation

Learning Objectives:
- FEM coding for computing elemental stiffness matrices using isoparametric formulation
- File I/O techniques for FE data
- Verification with ABAQUS

In this assignment, incomplete MATLAB codes are provided for computing element stiffness matrix of a 4-node 2D plane stress element (Q4). For detail computational algorithms, refer to Table 6.3-2 (R.Cook Book).

MATLAB codes:
- *HW6.m* : main program
- *ScanInput.m* : a function to read FE data from an input file
- *SetGaussQ.m* : a function to set Gauss Quadrature constants
- *GetElemStiff.m* : a function to compute the elemental stiffness matrix
- *GetGPV.m* : a function to get sampling point locations and weight factors (Incomplete)
- *GetSFD.m* : a function to get shape function derivatives (Incomplete)
- *GetCoord.m* : a function to get nodal coordinate values of the element (Incomplete)
- *GetDJacob.m* : a function to get Jacobian matrix and its determinant (Incomplete)
- *GetBMatrix.m* : a function to get B matrix (Incomplete)

Data files:
- *FEData.inp* : FE data file.
  (Note that this file will be read by ScanInput.m function. The syntax of this input file is very similar to ABAQUS input file.)
  Key MATLAB built-in functions used for this input file are fgetl( ); strtoken( ); . Refer to MATLAB help manual.
- *Abaqus_verif.inp* : ABAQUS input file to be used in verification (Incomplete)

Use one square element which has thickness 0.01 m. E=2.05E11 (Pa) and poisson ratio=0.29.

For verification of your FE codes, an incomplete ABAQUS input file (abaqus_verif.inp) is provided. Use '*element matrix output' command to write its elemental stiffness matrix in global coordinate system to a file 'estif'. Refer to ABAQUS keyword manual for the options of the command.

Use two different coordinate values
Case I) a square element
```
*node
1   0.0                 0.0                 0.0
2   1.0                 0.0                 0.0
3   1.0                 1.0                 0.0
4   0.0                 1.0                 0.0
```

Case II) a non-square and a non-parallelogram element
```
*node
1   1.0                 1.5                 0.0
2   3.1                 1.8                 0.0
3   5.0                 4.0                 0.0
4   0.8                 3.0                 0.0
```

Q1) Please complete the incomplete FE codes
Q2) For Case I and II, run your codes and ABAQUS. Submit 8 x 8 elemental stiffness matrices.
Q3) Print out and submit your complete FE codes
Q4) Print out and submit ABAQUS verification input file
Q5) Change nodal connectivity to '2, 3, 4, 1' (still counterclockwise) and state your observations of changes of the stiffness matrices
Q6) Change nodal connectivity to '4, 3, 2, 1' (clockwise) and state your observations of changes in the stiffness matrices

Note: This assignment is important for subsequent FE programming assignments!

Good luck!