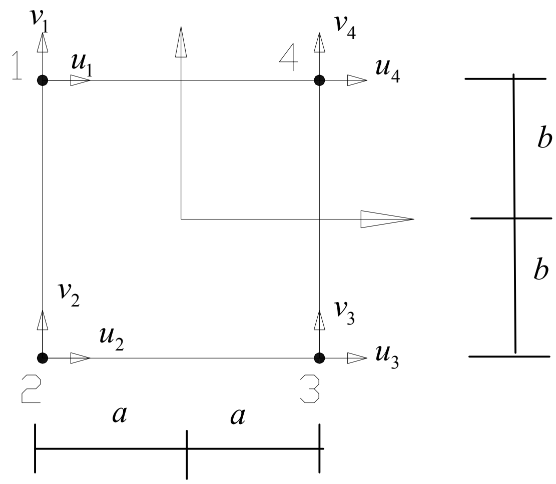
CEE-530/ME-524
Introduction to the Finite Element Method
Homework# 6
Due on: 10/31/2016
Electronic Submission in Sakai

Problem 1

Consider a Q4 element shown below used in an elastostatics finite element formulation. Take $a = b = 1$.



a) Derive the shape functions for this element. Notice that this is a Lagrangian element. You can use a generalized approach or products of 1D shape functions in this exercise.

b) Write a Matlab function that evaluates the shape functions of this element at a given point. That is, given point coordinates (x,y), your function will return a vector of shape functions, [N] evaluated at this point. Use this interface for your function:

```
function [N] = getN_Q4(x, y)
```

c) Write a Matlab function that evaluates the derivatives of the functions of this element at a given point. That is, given point coordinates (x,y), your function will return a vector of matrix functions, [G] evaluated at this point. Use this interface for your function:

```
function [G] = getGradN_Q4(x, y)
```

For this problem, the matrix G will look like this

$$[G] = \begin{bmatrix} \dfrac{\partial N_1}{\partial x} & \dfrac{\partial N_2}{\partial x} & \dfrac{\partial N_3}{\partial x} & \dfrac{\partial N_4}{\partial x} \\ \dfrac{\partial N_1}{\partial y} & \dfrac{\partial N_2}{\partial y} & \dfrac{\partial N_3}{\partial y} & \dfrac{\partial N_4}{\partial y} \end{bmatrix}$$

Remember that this is NOT a symbolic calculation and you need to evaluate the derivatives at a given point.

d) Write another Matlab function that accepts nodal values, point coordinates (x, y), and returns the interpolated displacements $u^e(x, y)$ and $v^e(x, y)$ at the given point. Report the value of the displacements at Point (0.5,-0.5) for the displacement vector $\{d^e\} = \{\ 0 \quad 0 \quad 0 \quad 0 \quad 1 \quad 1 \quad 1 \quad -1\}^T$. This code should make use of the function that you wrote in b). An interface for this function would be

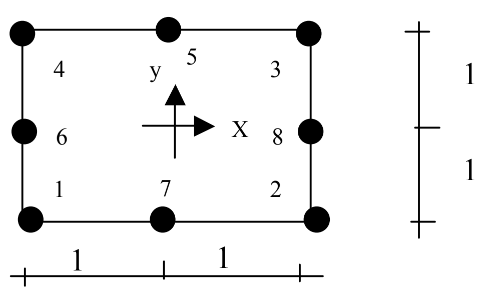
```
function [u, v] = getDisp (d, x, y)
```

e) Write a Matlab function that accepts nodal values, point coordinates (x, y), and returns the strains $\varepsilon_{xx}, \varepsilon_{yy}$, and ε_{xy} at the given point. Evaluate and report the strains at the point shown in d) for the same displacement vector. Use the function that you wrote in c) for this code. An interface for this function would be

```
function [epsilon] = getStrain(d, x, y)
```

Turn in your code for all parts of this Problem.

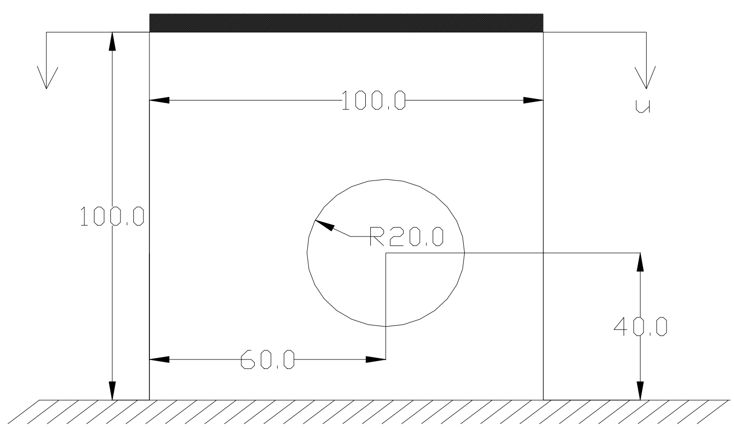
Problem 2:
Consider a solid Q8 element as shown in the figure. This element is used in a heat conduction FE formulation.



a) Derive the shape function corresponding to Node 2 using the approach shown in class for serendipity elements.
b) Assuming this is a 2D element for a heat conduction problem, write a Matlab program to compute the conductivity matrix for the element. Hand in your code (neatly and clearly commented), and the output for the stiffness matrix.

Use the following numerical values:
$\kappa = 100$ Conductivity
$t = 1$ Thickness of the element

Problem 3: A very long rectangular prism with a cylindrical hole is loaded with a rigid plate (i.e. all the points on the top surface displace vertically by the same amount) as shown in the figure below. Assume that the prism is supported on a frictionless surface so it can expand freely. Set the horizontal displacement to zero on the lower left corner so that there is no rigid body motion allowed. Carry out a finite element analysis and determine the vertical displacement u_y at which the material in the prism starts yielding. Also, determine the location where first yielding occurs. In order to determine first yielding in 2D and 3D problems, you have to compute the Von Mises stress (σ_e). Yielding occurs when the Von Mises stress reaches the uniaxial yield stress of the material. Perform a mesh convergence study to select a suitable mesh for the problem. For instance, the value of the maximum Von Mises stress should not be changing by more than some predefined tolerance of 1% from one mesh to another. Show a plot of the mesh used for the final analysis. Use a Q8 element fo this problem. [Hint: Once you have a converged mesh, the yield displacement can be obtained using the linearity of the problem].



All dimensions are in mm

Material properties

E = 200 GPa
$\upsilon = 0.3$
σ_Y = 420 MPa (Uniaxial yield stress)

Von Mises Stress

$$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_1 - \sigma_3)^2 + (\sigma_3 - \sigma_2)^2\right]}$$