Testing the Constant Strain Mesh

5.0 Principal Stresses

Previously we developed the FEA mathematics for a constant strain triangular mesh. This method is applicable to problems that can be idealized as two dimensional plates. All constraints and loads must be in the plane of the plate.

We developed the equation

$$\sigma = DBq \tag{4.73}$$

or expanding

$$\sigma = \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \frac{E}{1-v^2} \begin{bmatrix} 1 & v & 0 \\ v & 1 & 0 \\ 0 & 0 & \dfrac{1-v}{2} \end{bmatrix} \frac{1}{\det J} \begin{bmatrix} y_{23} & 0 & y_{31} & 0 & y_{12} & 0 \\ 0 & x_{32} & 0 & x_{13} & 0 & x_{21} \\ x_{32} & y_{23} & x_{13} & y_{31} & x_{21} & y_{12} \end{bmatrix} \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \end{Bmatrix} \tag{4.74}$$

for computing the stresses for each triangular element.

We used a linear equation to interpolate the displacements and this yielded a constant strain or stress across each triangular element. Knowing that this is only an approximation, we decided to place the computed stress near the centroid of the triangle.

Equation 4.74 indicates we are computing two axial stresses and a shear stress. These are illustrated in Figure 1 below.

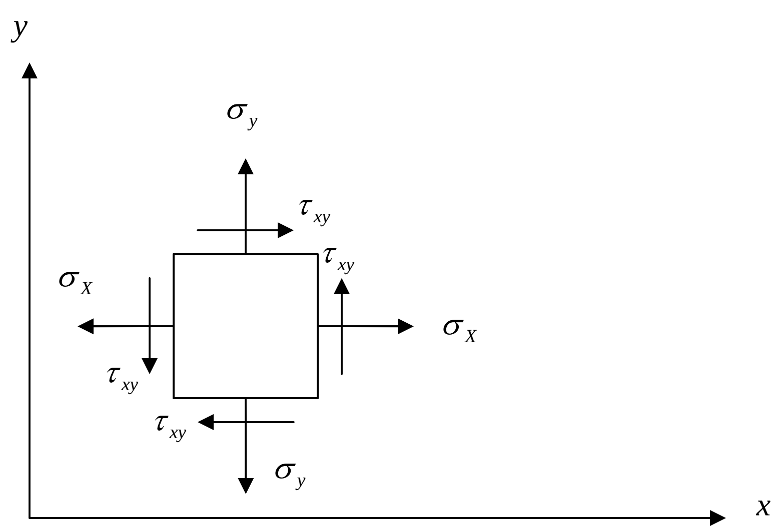

Figure 1 - Material element showing the axial and shear stresses.

The axial and shear stresses vary with the orientation of the small element and we can compute these with Mohr's circle.

5.1 Mohr's Circle

Mohr's circle is a graphical method for computing axial and shear stresses in the element of material shown in Figure 1. The circle is plotted on a coordinate system with the axial stresses plotted along the horizontal axis and the shear stresses plotted along the vertical axis.

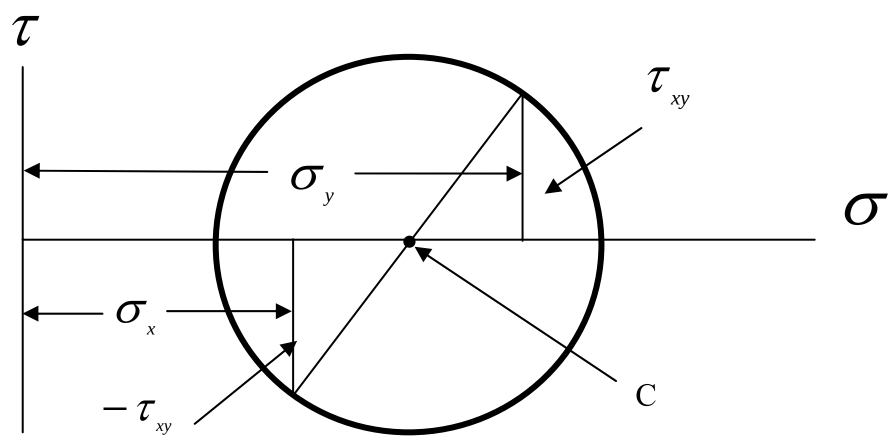

Figure 2 - Mohr's circle drawn from the computed stresses.

The stresses computed in Equation 4.74 are plotted by plotting the point defined by $\{\sigma_y, \tau_{xy}\}$ and $\{\sigma_x, -\tau_{xy}\}$. This defined two points on the circle.

We know that the circle is centered (point C) on the σ axis so we can compute the coordinates of this point by averaging the two axial stresses.

$$C = (\sigma_x + \sigma_y)/2 \tag{5.1}$$

The distance R from point C to the circle can be computed with Pythagoras's formula. The hypotenuse of the triangle is the distance from the center point to the circle. One leg of the triangle is τ_{xy} and the other can be computed as:

$$A = \left(\frac{\sigma_y - \sigma_x}{2} \right) \tag{5.2}$$

The distance R is computed with equation 5.3.

$$R = \sqrt{A^2 + \tau_{xy}^2} \tag{5.3}$$

The principal stresses, where the circle crosses the axial stress axis σ can be computed with:

$$\sigma_1 = C + R \tag{5.4}$$

and

$$\sigma_2 = C - R \tag{5.5}$$

These points are shown in Figure 3.

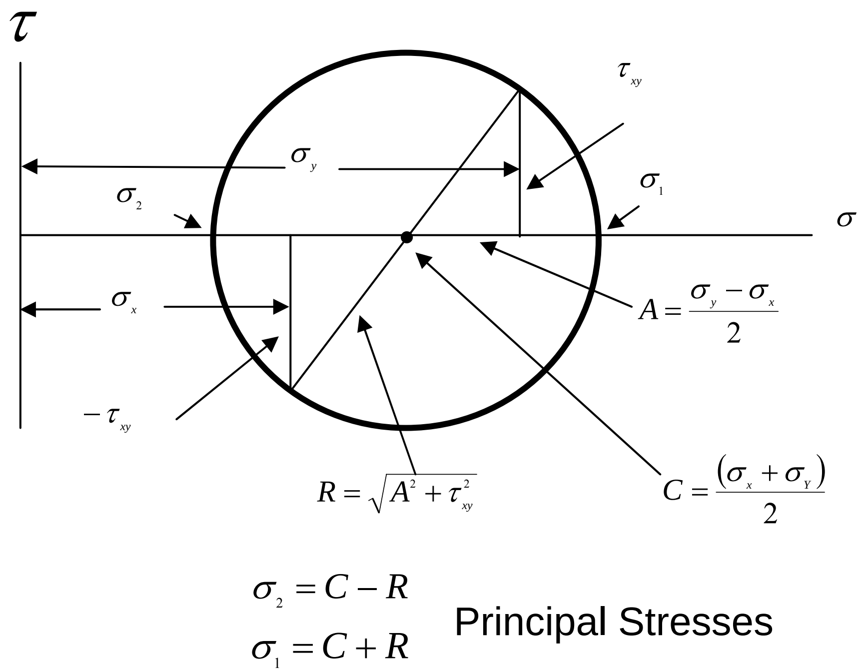

Figure 3 - Mohr's circle with principal stresses.

The circle represents various orientations of the element shown in Figure 1. Each degree of rotation of the element is represented by two degrees on Mohr's circle. When an element is oriented so that all of the stresses are axial, the stresses in the element will relate to the principal stresses. The maximum shear stresses occur when the element is oriented 45 degrees from the principal stress orientation. On Mohr's circle, this corresponds to the top and bottom of the circle. The maximum shear stresses can be computed with:

$$\tau_{max} = R \tag{5.6}$$

5.2 Von Mises Stress

As designers, it is usually important to limit the stresses so that our designs do not deform permanently. Richard von Mises and several other researchers studied this problem and determined that the material will yield when the distortion energy per unit of volume equals the tension yield stress of the material. We call this measure of distortion energy, the von Mises stress. It can be computed with:

$$I_1 = \sigma_x + \sigma_y \tag{5.7}$$

$$I_2 = \sigma_x \sigma_y - \tau_{xy}^2 \tag{5.8}$$

$$\sigma_{VM} = \sqrt{I_1^2 - 3I_2} \tag{5.9}$$

Equation 5.9 shows what is commonly called the von Mises stress. It can be computed with the axial and shear stresses computed in Equation 4.74.

It is important to keep the von Mises stress below the yield stress of the material we are using in our design. If the von Mises stresses go beyond the yield stress, the object we are designing will permanently deform.

An example of a stress strain curve is shown in Figure 5. The FEA method we have developed only works in the linear range of the curve. If the von Mises stress is below the "Yield Point" stress, the material will be in the linear elastic range.

Figure 4 - Richand von Mises

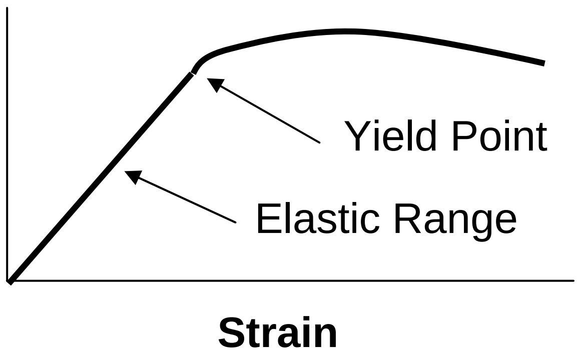

Figure 5 - Typical stress / strain curve for many metals.

5.3 Testing the Method

One way to test the method we have developed is to write a program based on the method then compare the results with that produced by other programs. We will look at a rectangular plate that is fixed at one end and free on the other. We will place a 1000 pound load on the free end.

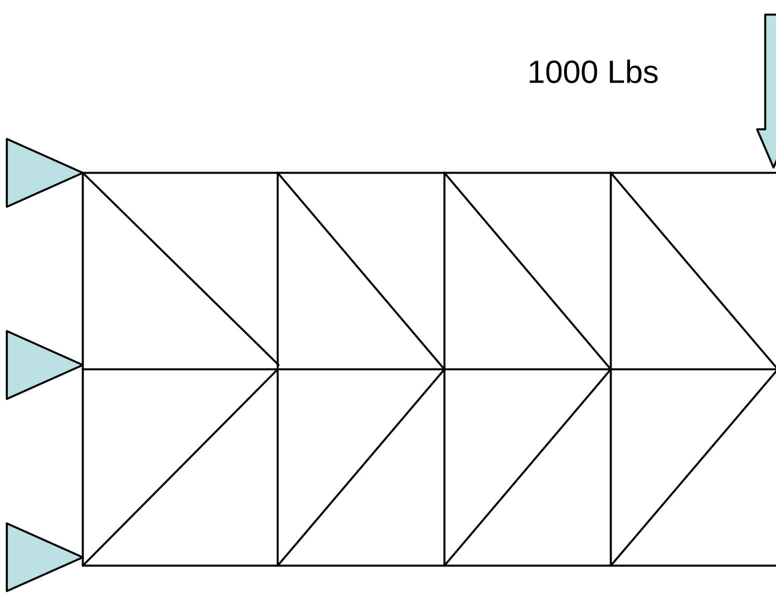

Figure 6 - Plate with elements shown.

The plate is 3 inches wide, 2 inches high and 0.5 inches thick. The modulus of elasticity is $30x10^6$ and Poison's ratio is 0.25. We will divide the plate into 16 elements. The actual data is shown in the two tables below. The table labeled "Nodal Coordinates" gives the node numbers and the coordinates of the node. The table labeled "Element Connectivity" defines each triangular element and the three nodes defining its vertices.

Nodal Coordinates		
Node	**X**	**Y**
1	0	0
2	0.75	0
3	1.5	0
4	2.25	0
5	3	0
6	0	1
7	0.75	1
8	1.5	1
9	2.25	1
10	3	1
11	0	2
12	0.75	2
13	1.5	2
14	2.25	2
15	3	2

	Element Connectivity		
Element	**Node 1**	**Node 2**	**Node 3**
1	1	7	6
2	1	2	7
3	2	8	7
4	2	3	8
5	3	9	8
6	3	4	9
7	4	10	9
8	4	5	10
9	6	7	11
10	7	12	11
11	7	8	12
12	8	13	12
13	8	9	13
14	9	14	13
15	9	10	14
16	10	15	14

The von Mises stresses are shown below in Figure 7 below. The method we developed assumed a linear shape function and this implied that the stress and strain were constant across each element. We know that the stress is not constant across the entire element so we assume that it applies to the center of each triangle near where the stresses are shown in the figure.

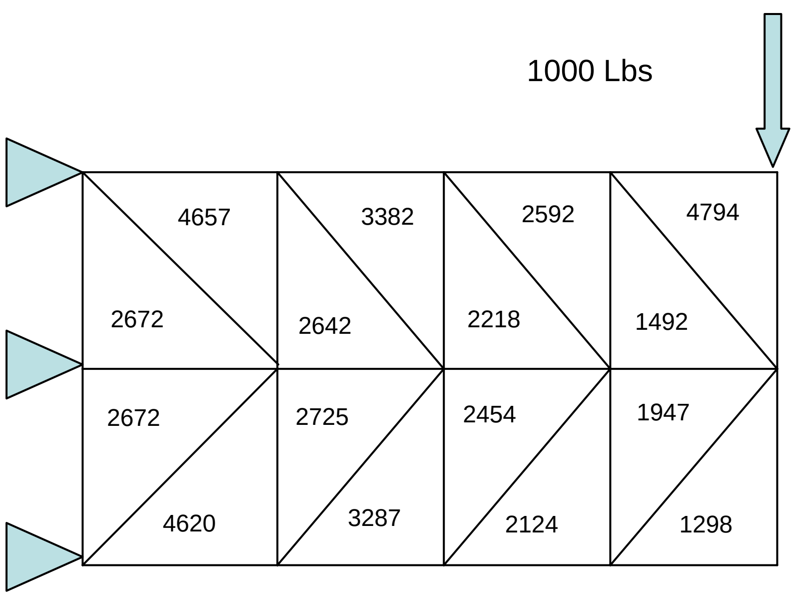

Figure 7 - Plate with von Mises stresses shown.

The same problem was solved using ANSYS, a commercial finite element program. ANSYS allows the exact same shape of elements to be defined but it uses a different algorithm. ANSYS uses a quadratic shape function which should produce a more realistic solution than the method we developed in class. The ANSYS results and the results from our class developed method are shown in the table below.

Elem	Class	ANSYS
1	2672	2672
2	4620	4620
3	2725	2725
4	3287	3287
5	2454	2454
6	2124	2124
7	1948	1948
8	1298	1298

Elem	Class	ANSYS
9	2672	2672
10	4657	4620
11	2642	2725
12	3382	3287
13	2218	2454
14	2592	2124
15	1492	1948
16	4794	4467

As you can see from the table above, the solutions using the method we developed compare quite well with the ANSYS solutions. The solutions match very well on the left side of the cantilever but diverge from the ANSYS solutions as we move closer to the point load. There will be a force concentration at this point and the quadratic elements used in ANSYS handle this stress concentration better than our constant strain elements. As you move farther away from this high stress area, the stresses match very closely.

5.4 Homework

Using the method we developed in class, write a Matlab program to compute the displacements at each node, and the stresses, principal stresses, and von Mises stresses for each element in the constant strain mesh. Test your program using the problem solved in the notes. Secondly, test your program using the plate problem shown above. Email the program to me when you have finished.

You should be able to write this program by modifying the previous FEA Truss program. The overall structure is the same. The only difference is the method used to form the global stiffness matrix. The individual element stiffness matrices are a little more complex than they were in the FEA Truss program.

In the FEA Truss program, it was not necessary to actually generate the stiffness matrix for each element. It was easy enough to add the terms of the individual stiffness matrix directly to the global stiffness matrix without actually forming the individual matrix.

The individual stiffness matrix for the elements in the Constant Strain Mesh program are more complex and it will probably be easier to create the element stiffness matrix then go through the matrix term by term adding them to the global stiffness matrix. As soon as you use the elemental stiffness matrix, you can discard it by using the variables for the next element.

The program that you create will read data from a file as did the other programs you have written. The format for the input date file is shown below.

```
30e6                        Young's Modulus
.25                         Poisson's ratio
0.5                         Plate thickness
15                          Number of Nodes
1  0.0    0.0               Nodal Data
2  0.75   0.0
3  1.50   0.0
4  2.25   0.0
5  3.00   0.0
6  0.0    1.0
7  0.75   1.0
8  1.50   1.0
9  2.25   1.0
10 3.0    1.0
11 0.0    2.0
12 0.75   2.0
13 1.50   2.0
14 2.25   2.0
15 3.0    2.0
16                          Number of Elements
1   1   7   6               Element Data
2   1   2   7
3   2   8   7
4   2   3   8
5   3   9   8
6   3   4   9
7   4  10   9
8   4   5  10
9   6   7  11
10  7  12  11
11  7   8  12
12  8  13  12
13  8   9  13
14  9  14  13
15  9  10  14
16 10  15  14
6                           Number of constraints
1   1  X                    constraint at node 1
2   1  Y
3   6  X                    constraint at node 6
4   6  Y
5   11 X                    constraint at node 11
6   11 Y
1                           Number of loads
15  1000  -90               Load at node 15
```

The output from the program is shown below.

```
>> Plate('Bigtest.txt')

Youngs Modulus = 3e+007
Poisons Ratio = 0.25
Plate Thickness = 0.5
Nodal Displacements
 Node          DX              DY
   1            0               0
   2    -0.000116461   -0.000117267
   3     -0.00019675   -0.000296573
   4    -0.000241064   -0.000516959
   5    -0.000247554   -0.000731622
   6            0               0
   7   -9.66529e-007  -9.63957e-005
   8   -3.01734e-006   -0.000278868
   9   -7.11942e-006   -0.000510546
   10  -1.32574e-005    -0.00075542
   11            0               0
   12    0.000118135   -0.000115888
   13    0.000202327   -0.000296892
   14    0.000258718   -0.000537295
   15    0.000308076   -0.000874441

Element Stresses
Element  Stress X       Stress Y         Shear     Principal 1    Principal 2      VM Stress
1        -41.2386          -10.3      -1542.33        -1568.18        1516.63        2671.65
2        -4802.01           -574       -490.35        -4858.14       -518.222        4620.88
3         79.4727            646      -1533.62        -1196.82        1922.31         2725.5
4        -3284.03           -290      -544.103        -3379.84       -194.034        3287.12
5        -33.3778            523      -1382.05        -1165.03        1654.48           2454
6        -1839.41           -267      -718.838        -2118.56        11.6892        2124.43
7        -210.582            140      -1110.66         -1159.8        1088.97        1947.81
8        -467.294           -831      -623.059        -1298.04           0.00        1298.04
9        -41.2386          -10.3      -1542.33        -1568.18        1516.63        2671.65
10        4884.49            636      -424.987         594.258        4926.59        4657.98
11       -243.439           -646      -1490.33        -1948.37        1059.31        2642.39
12           3448            321      -431.938         262.691        3506.57        3382.88
13       -319.218           -621      -1242.71        -1721.69        781.932        2218.53
14        2192.01           -254      -656.396        -419.465           2357        2592.31
15       -475.883           -921       -727.94        -1459.94        62.6006        1492.22
16        1153.76      -3.28e+003     -1538.35        -3763.47        1635.03        4794.83
```