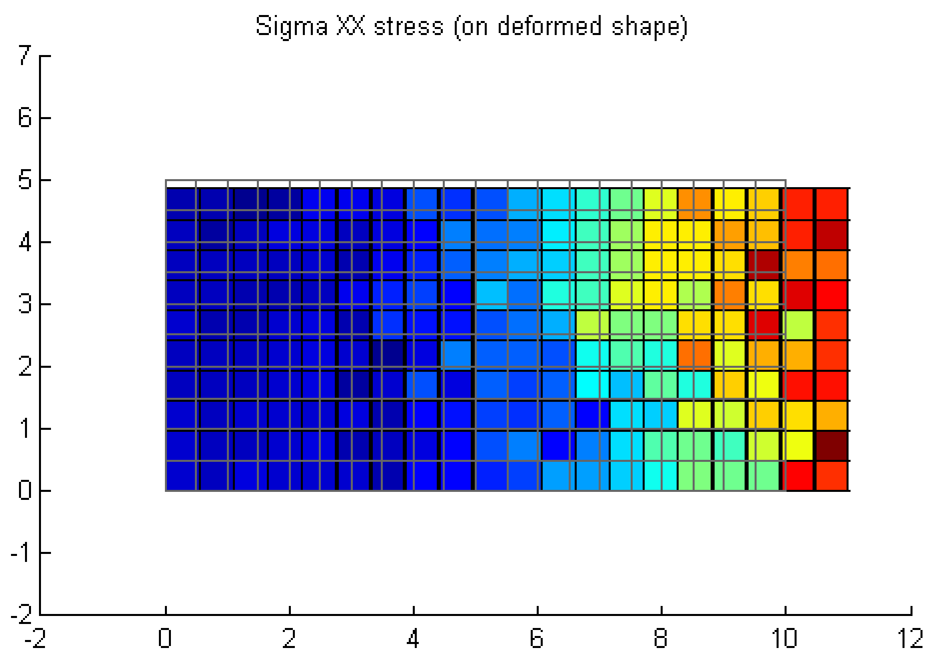
ISTD 01.110 Computational Fabrication

[Assignment 4] Finite Element Method (FEM)

Zhang Zhexian (1001214)

2D Plate FEM

Running the MATLAB simulation produces the following graph. We observe that the mesh has increased its length (horizontal displacement) while largely maintains its original width (vertical displacement). Also, the elements in the right-side experience greater stress than the ones in the left. This is because the traction force is applied in the positive x-direction.



Measurement result shows the following:

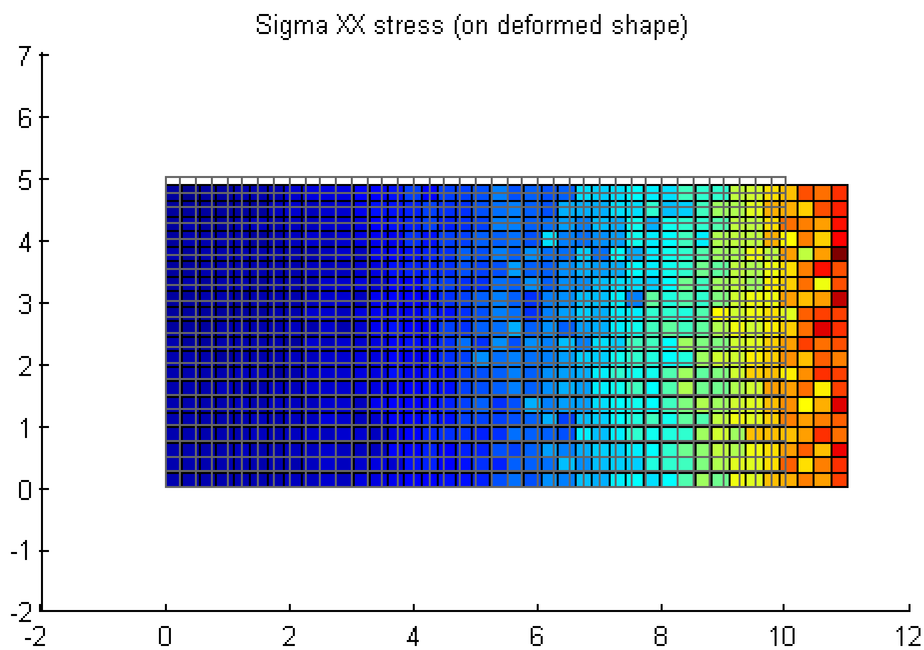
- Strain energy: 4.0372
- Poissons ratio: 0.3

Poisson’s Ratio is a property associated with the material that controls volume preservation, and it remains unchanged when tractor force is applied.

The 2D plane is conceptually similar to a linear spring, so its strain energy is calculated by Hooke’s Law.

Mesh refinement

Running the MATLAB simulation produces the following graph. Similar to the graph above, we also observe that the mesh has increased its length (horizontal displacement), and elements in the right-side experience greater stress than the left. In addition, the stress transition looks finer and more gradual after mesh refinement. This is because as more elements are used for finite element method, the model behaves closer to the behavior in real life, which has stress behavior that transits smoothly along the object's length.



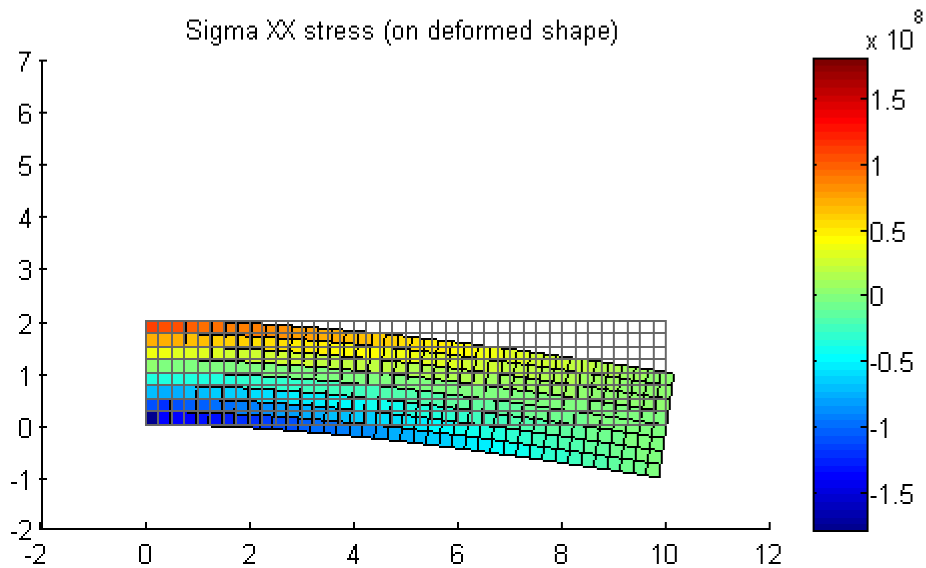
Measurement result shows the following:

- Strain energy: 14.8603
- Poissons ratio: 0.3

Similarly, the Poisson's Ratio remains unchanged. However, the strain energy of the refined mesh is higher, indicating that the material has deformed to a greater extent with elastic behavior.

2D Beam FEM

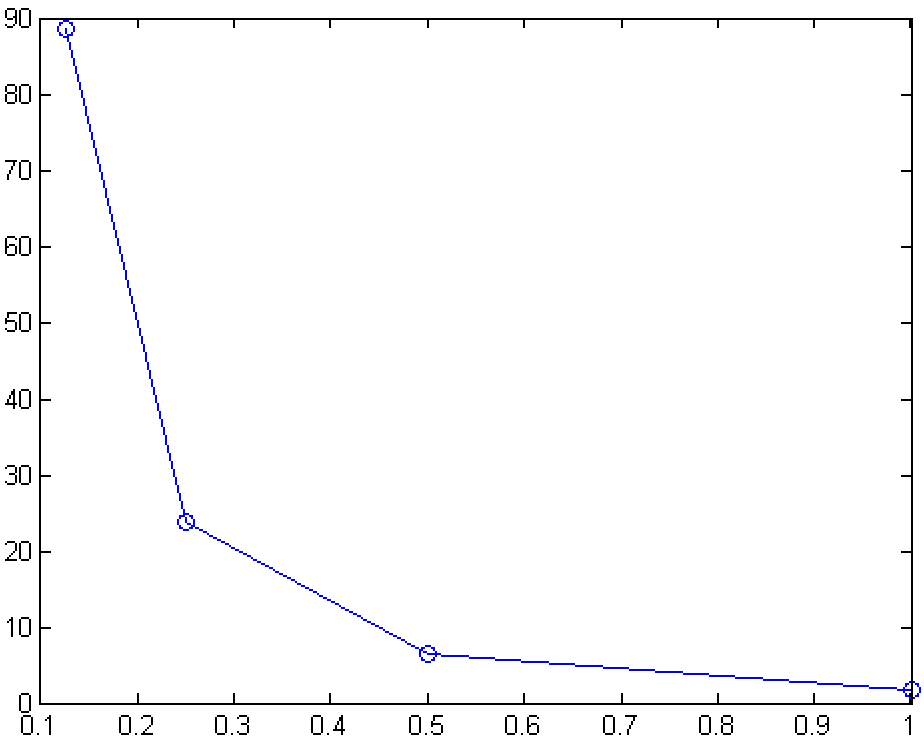
Running the MATLAB simulation produces the following graph. The mesh has bent downwards, with the right-side lower than the left, and appeared to be curvy instead of straight. The stress is increasing as we move from the bottom up, and from the right to the left, resulting in the top left corner having the highest stress, while the bottom left corner having the lowest. This is because the boundary condition is applied to the left side, and the traction force is to the negative-y direction.



The table of h and U under different settings are shown:

lx	***ly***	***h*: mesh size parameter**	***U*: strain energy**
10	2	1	1.7931
20	4	0.5	6.5744
40	8	0.25	23.7356
80	16	0.125	88.6084

The plot of U over h:



The plot shows that *U*, the strain energy, is inversely proportional to *h*, the mesh size parameter. This means the more refined the mesh, the greater the strain energy it experiences.

This also shows that as more elements are used for finite element method, the model behaves closer to the behavior in real life, which has stress behavior that transits smoothly along the object's length. We may thus model the behavior of elastic material by dividing the object into small elements (the smaller the better), and integrate over the result of each element.