Computational Mechanics Final Report

Finite Element Simulation

Krit Surintraboon

C1TB1707

krit.surintraboon.q7@dc.tohoku.ac.jp

Submission date: 3/2/2024

1. Overview

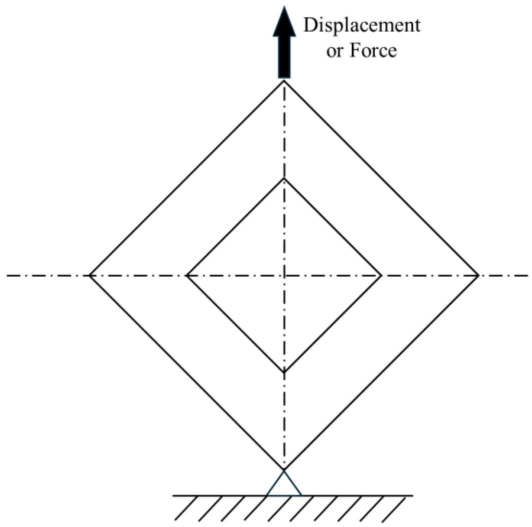

Figure 1: Applying force on the material.

In this report, the deformation of the object after force is applied, as depicted in figure 1, will be simulated by using a finite element method. The following report will contain the overall step to compute the simulation using python, the deformation of the object, the stress and strain distribution, and the discussion regarding element size, and plain stress or strain conditions. The details of the source code will not be included, since it is already explained in the source code file (jupyter notebook).

2. Computation steps

2.1. Creating elements

In the very first step of finite element simulation, the element must be divided into smaller elements. In this simulation, I decided to employ triangular-shaped elements instead of rectangular-shaped elements. This is done by defining the nodes and creating a triangular mesh on those nodes.

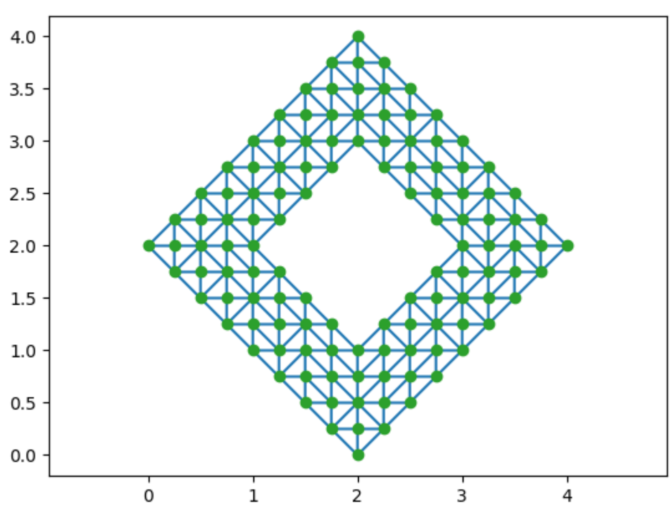

Figure 2: Creating triangular mesh.

2.2. Finding stiffness matrix of each element

The stiffness matrix $\boldsymbol{K_n}$ of each element can be computed using the following formula:

$$\boldsymbol{K_n} = a_n \boldsymbol{B_n^T D B_n}$$

, where a_n is the area of the element, $\boldsymbol{B}$ matrix is the matrix that shows the realationship between the displacement and strain of each element, and $\boldsymbol{D}$ matrix is the Elastic modulus matrix. Therefore, in order to find $\boldsymbol{K_n}$, it is necessary to compute $\boldsymbol{D}$ matrix and $\boldsymbol{B_n}$ matrix first.

$\boldsymbol{D}$ matrix is a constant for the material, only depending on Young's modulus (E) and Poisson ratio (ν). However, the value of $\boldsymbol{D}$ matrix varies between plain strain condition and plain stress condition. By selecting Aluminum alloy 1100, which has the Young's modulus of 69 GPa and the Poisson ratio of 0.33, as the material, the matrix in plain strain condition is expressed as:

$$\boldsymbol{D} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & 0 & 0 \\ 0 & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} = \begin{bmatrix} 102.23 & 50.35 & 0 \\ 50.35 & 102.23 & 0 \\ 0 & 0 & 25.94 \end{bmatrix} \text{GPA}$$

And in plain stress condition, the matrix is expressed as:

$$\boldsymbol{D} = \frac{E}{(1+\nu)(1-\nu)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} = \begin{bmatrix} 77.43 & 25.55 & 0 \\ 25.55 & 77.43 & 0 \\ 0 & 0 & 25.94 \end{bmatrix} \text{GPA}$$

On the other hand, every $\boldsymbol{B}$ matrix corresponds to each element, and they have different value if they are computed from different elements. The matrix can be expressed as follows:

$$\boldsymbol{B} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix}$$

, where

$$\frac{\partial N_1}{\partial x} = \frac{y_2 - y_3}{2a}, \quad \frac{\partial N_2}{\partial x} = \frac{y_3 - y_1}{2a}, \quad \frac{\partial N_3}{\partial x} = \frac{y_1 - y_2}{2a}$$

$$\frac{\partial N_1}{\partial y} = \frac{x_3 - x_2}{2a}, \quad \frac{\partial N_2}{\partial y} = \frac{x_1 - x_3}{2a}, \quad \frac{\partial N_3}{\partial y} = \frac{x_2 - x_1}{2a}$$

, and

$$a = \frac{1}{2} det \begin{bmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{bmatrix}$$

2.3. Finding total stiffness matrix ($\boldsymbol{K_t}$)

$$\boldsymbol{K_n} = \begin{array}{c} \\ f_{1x} \\ f_{1y} \\ f_{2x} \\ f_{2y} \\ f_{3x} \\ f_{3y} \end{array} \begin{array}{c} \begin{array}{cccccc} u_{1x} & u_{1y} & u_{2x} & u_{2y} & u_{3x} & u_{3y} \end{array} \\ \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & K_{15} & K_{16} \\ K_{21} & K_{22} & K_{23} & K_{24} & K_{25} & K_{26} \\ K_{31} & K_{32} & K_{33} & K_{34} & K_{35} & K_{36} \\ K_{41} & K_{42} & K_{43} & K_{44} & K_{45} & K_{46} \\ K_{51} & K_{52} & K_{53} & K_{54} & K_{55} & K_{56} \\ K_{61} & K_{62} & K_{63} & K_{64} & K_{65} & K_{66} \end{bmatrix} \end{array}$$

Each element of the computed $\boldsymbol{K_n}$ matrix is the displacement of the node in x or y direction corresponded to the force in x or y direction applied to one of the nodes in the element. Thus, to compute the total stiffness of the material, every $\boldsymbol{K_n}$ must superimposed according to their coordinate and force applied. Finally, the obtained $\boldsymbol{K_t}$ will have the dimension of $[2n(node) \times 2n(node)]$

2.4. Compute the displacement of each node to obtain the deformed shape

After obtaining the total stiffness matrix, the displacement vector ($\boldsymbol{u}$) from force vector ($\boldsymbol{f}$) can be obtained using the following relation:

$$\boldsymbol{K_t u} = \boldsymbol{f}$$

However, before using the above relation to compute $\boldsymbol{u}$, first, each vector and matrix needs to be cropped. In the $\boldsymbol{u}$ vector, each node occupies two entries, representing displacement in x and y directions. Since the base of the geometry is fixed, the base node will have both displacement in x and y directions equal to zero. Moreover, since the force is only applied vertically, the node located at the symmetric axis must have the displacement in x direction equal to zero as well. Because these nodes are not free to transform, there will be forces applied to those nodes in their fixed direction. The magnitudes of these forces are unknown, therefore, must be removed from the equation, otherwise, the unknown variable will exceed the number of equations. When cropping $\boldsymbol{u}$ and $\boldsymbol{f}$ vectors, rows and column $\boldsymbol{K_t}$ must also be cropped according to the unknown force elements and zero displacement elements. Finally, the displacement vector can be obtained through the following relation:

$$\boldsymbol{u_{cropped}} = \boldsymbol{K_{t_{cropped}}^{-1} f_{cropped}}$$

The deformed shape, then, can be obtained by adding node displacement to the original node coordinate. By applying vertical force of magnitude 5GN to the top node in the plane strain condition, the deformed shape is shown in figure 3. The deformed shape in plane stress condition will be shown in the discussion section.

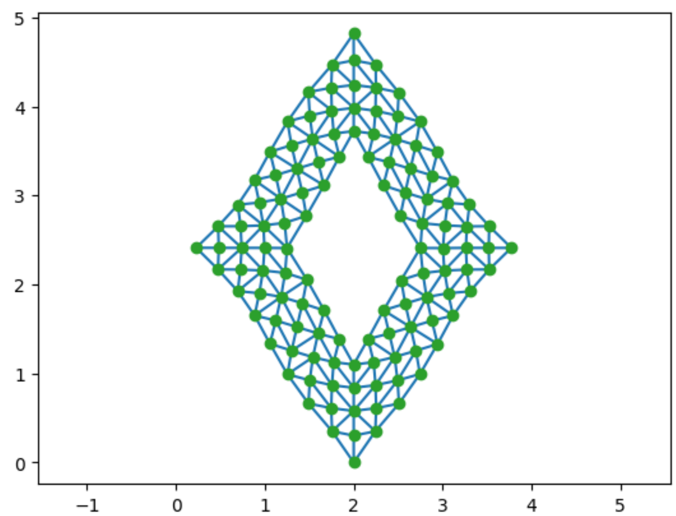

Figure 3: Deformation in plane strain condition

2.4. Compute the strain of each element

After obtaining the displacement of each node, the strain value of each element can be computed by:

$$\boldsymbol{\varepsilon} = \boldsymbol{Bd}$$

$$\begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ 2\varepsilon_{xy} \end{Bmatrix} = \begin{bmatrix} \frac{\partial N_1}{\partial x} & \mathbf{0} & \frac{\partial N_2}{\partial x} & \mathbf{0} & \frac{\partial N_3}{\partial x} & \mathbf{0} \\ \mathbf{0} & \frac{\partial N_1}{\partial y} & \mathbf{0} & \frac{\partial N_2}{\partial y} & \mathbf{0} & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix} \begin{Bmatrix} u_{1x} \\ u_{1y} \\ u_{2x} \\ u_{2y} \\ u_{3x} \\ u_{3y} \end{Bmatrix}$$

The strain of each element, when a vertical force of magnitude 5GN is applied to the object in the plain strain condition is shown in the following figures 4, 5 and 6.

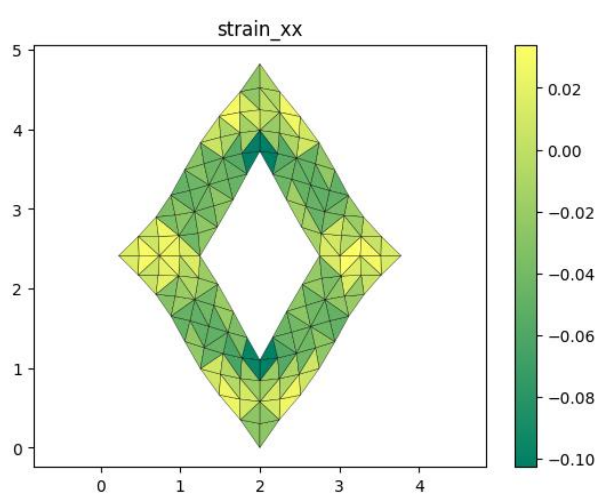

Figure 4: Strain in x direction

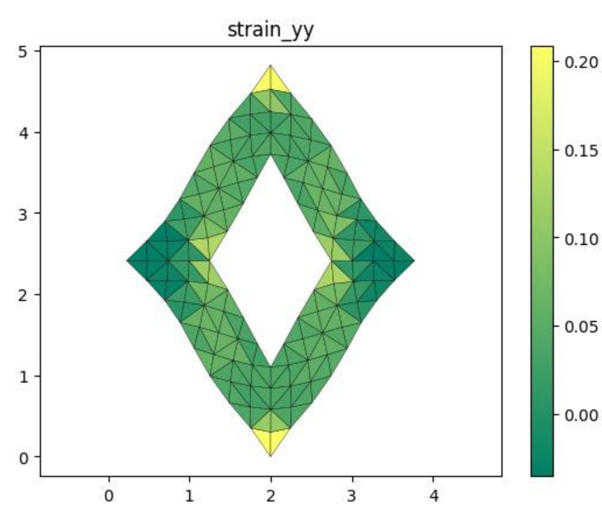

Figure 5: Strain in y direction

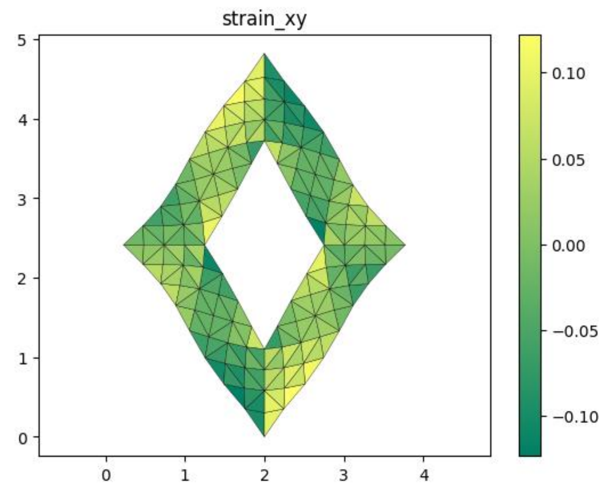

Figure 6: Shear strain

From figure 4, it can be seen that most of the element is compressed in x direction, having the negative value for strain. The most compressed area concentrates around the top and bottom of the hole. On the other hand, in y direction, the material is stretched, having positive strain value as shown in figure 5. The highest strain in y direction concentrates at the top and bottom of the material, where the force is applied, and the node is fixed. For shear strain shown in figure 6, the strain value is antisymmetric about the $x = 2$ axis. The shear strain also concentrates at the top and bottom of the material.

2.4. Compute the stress of each element

The computed strain vector from the last section can be used to calculate the stress vector by using the following relation:

$$\boldsymbol{\sigma} = \boldsymbol{D}\boldsymbol{\varepsilon}$$

$$\begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{xy} \end{Bmatrix} = \boldsymbol{D} \begin{Bmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ 2\varepsilon_{xy} \end{Bmatrix}$$

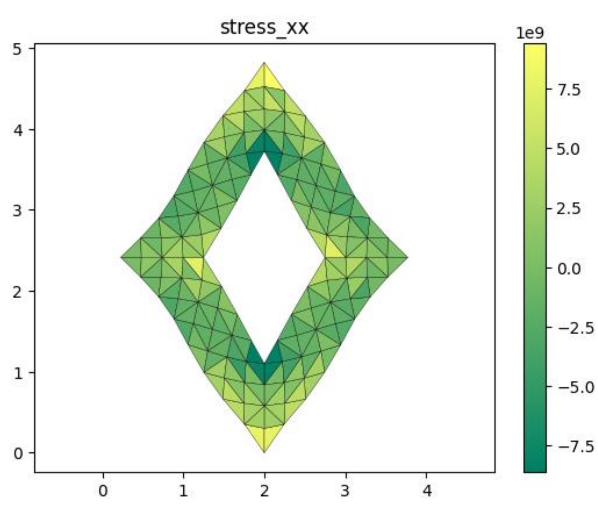

Figure 7: Stress in x direction

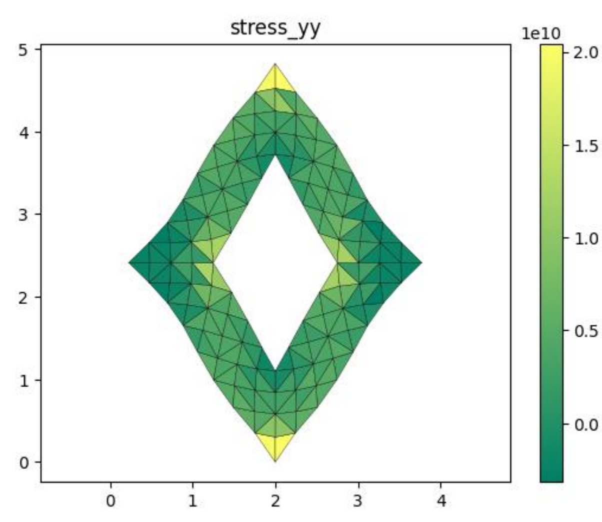

Figure 8: Stress in y direction

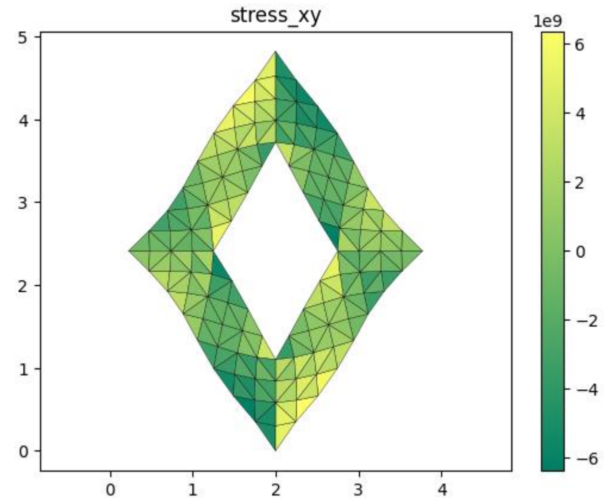

Figure 9: Shear stress

From figures 7, 8 and 9, it can be seen that the stress distributions are very similar to the strain distribution. However, since the stress distribution in x and y direction are computed from both strain in x and y direction with different proportions (can be seen from ***D*** matrix), the gradient of the distribution seems to be more steadily distributed.

3. Discussion

3.1. Effect of the number of element divisions

In the simulation above, 120 nodes are defined on the element. To observe the effect of the number of divisions or the size of element, simulation with a smaller number of nodes (12 and 36 nodes) will be performed with the same condition as above.

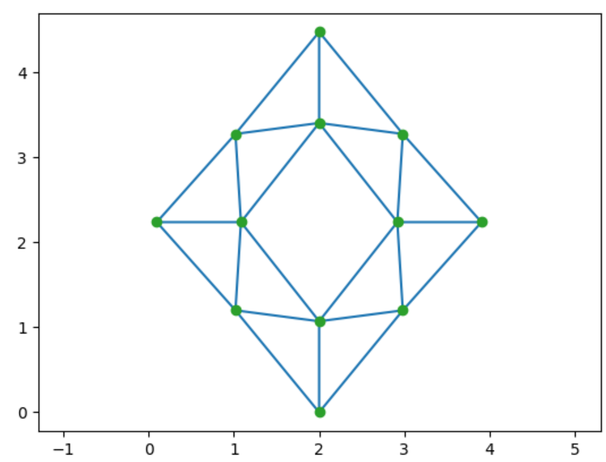

Figure 10: Deformation simulation with 12 nodes

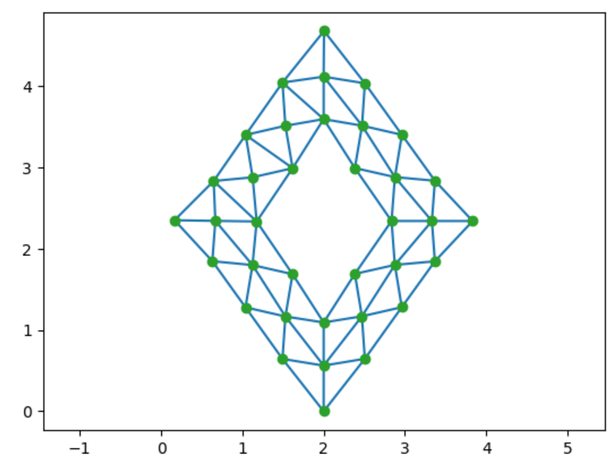

Figure 11: Deformation simulation with 36 nodes

Compared to the 120 nodes simulation in figure 3, the simulation shown in figures 10 and 11, which use larger elements, simplify the deformation structure and seems not able to show the localized effect accurately, leading to less precise result. On the other hand, smaller elements in the 120 nodes simulation can capture detailed changes in how a structure deforms, especially in places where there's a lot of stress or strain, more accurately.

3.2. Difference between plane strain and plane stress conditions

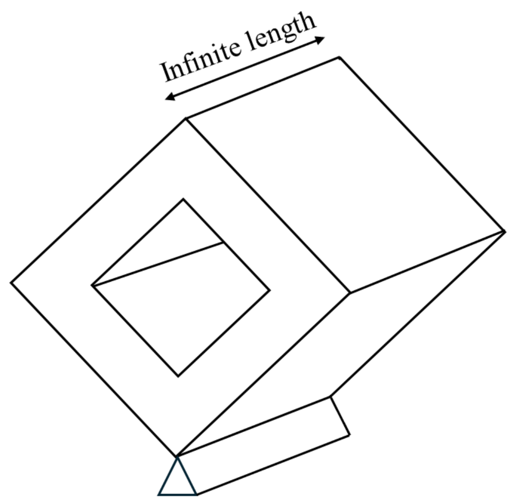

Figure 12: Plane strain condition

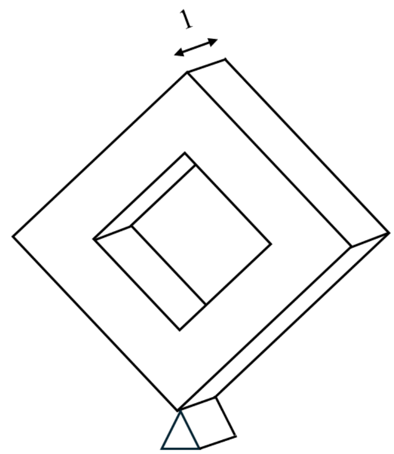

Figure 13: Plane stress condition

In plain strain condition, a material is constrained in one dimension while free to deform in the other two dimensions, often assumed to analyze deformation section whose length is much larger than the cross-sectional size. On the other hand, plain stress assumes stress-free conditions in one direction, typically applied to thin structures, where stresses in the thickness direction are considered small or neglected.

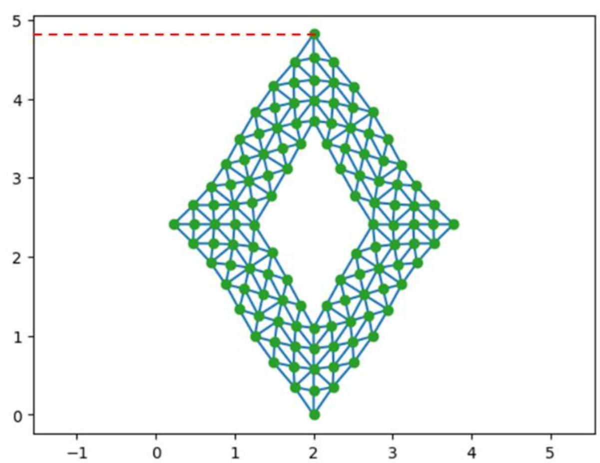

Figure 14: Deformation in plane strain condition

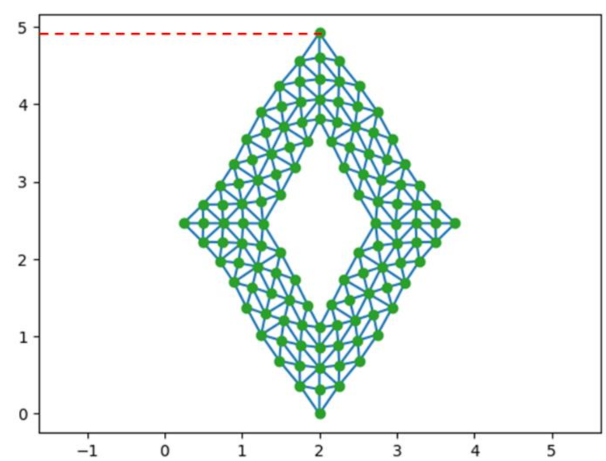

Figure 15: Deformation in plane stress condition

By comparing deformation in plane strain and plane stress conditions in figures 14 and 15, it can be seen that in plane stress condition, the object elongates in y-direction slightly more than that in plane strain condition.