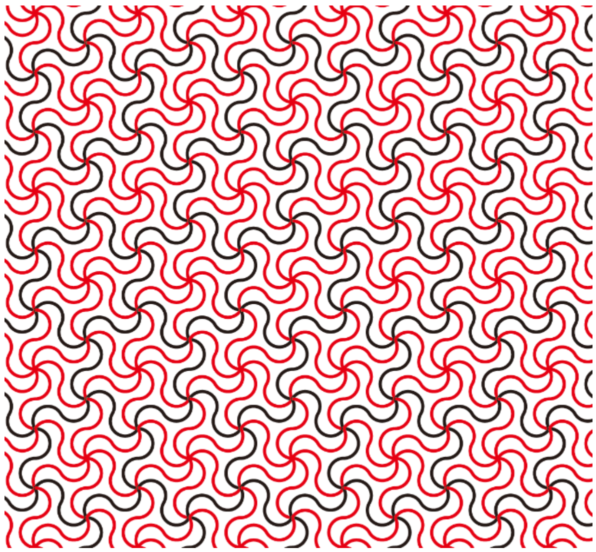
ENGN 2340 Final Project Report

Optimization of Mechanical Isotropy of Soft Network Material

Enrui Zhang 12/15/2017

1. Introduction of the Problem



This project deals with the stress-strain response of a 2-dimensional periodic wavy filamentary network under uniaxial finite displacement loading from arbitrary direction with ABAQUS, as shown in the figure above. Black and red refer to curved beams made of material I and II respectively. Beams of material I form a periodic honeycomb structure, whereas material II subdivide each "honeycomb" into 6 parts. All the beams are geometrically identical, and the width is much smaller than the curvature radius of beams. The arc angle of curved beam is fixed at 180°. It is assumed that the geometric scale of a single beam is much smaller than the macroscopic network. To simplify the modeling of this problem, one representative unit cell is picked from the network on which the simulation is conducted. To ensure that the behavior of the unit cell is the same as the behavior of full-scale network, periodic boundary conditions are applied on corresponding boundaries of the unit cell.

Previous research[1] has shown that when material II does not exist ($E_{\mathrm{II}} / E_{\mathrm{I}} \to 0$, where E_α is the Young's Modulus of material α), the pure honeycomb network is has the smallest stiffness under vertical loading and largest stiffness under horizontal loading. On the contrary, when material II is the same materials as material I ($E_{\mathrm{II}} / E_{\mathrm{I}} = 1$), the triangular network has the largest stiffness under vertical loading and the smallest stiffness under horizontal loading. Therefore, it may be postulated that there exist a $\beta \in (0,1)$, such that for $E_{\mathrm{II}} / E_{\mathrm{I}} = \beta$ this anisotropy of (nominal) normal stress-strain response reaches the minimum.

With different values of β, different stress-strain behavior can be obtained with the setup of ABAQUS with periodic boundary conditions. The final aim of this project is to find the β that minimize the anisotropy in terms of stress-strain behavior of the material.

2. Setup of Problem with ABAQUS

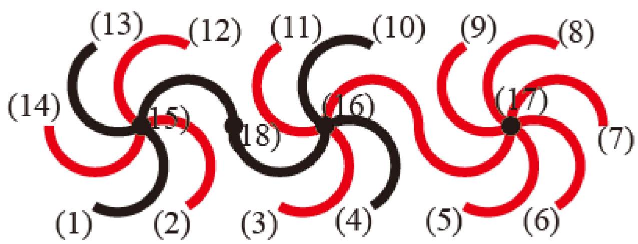
Although the macroscopic network is applied with finite displacement loading. However, due to the wavy microstructure of the network, the intrinsic strain of material is small. Therefore, the basic constitutive model of plain stress linear elasticity in ABAQUS is used in this simulation. A static step is set, and the network is meshed with quadrilateral quadratic 2D plain stress elements. To ensure the accuracy of simulation, the number of elements along the direction of width is no less than 4.

In this problem, establishing boundary conditions will be the most complicated and time-consuming step, compared to the simple constitutive model. In the following, three necessary types of boundary conditions will be discussed: (i) periodic boundary conditions, (ii) uniaxial displacement loading (iii) constraints of closed polygon.

Parameters in this problem include material modulus E_{I}, E_{II}, Poisson's ratio ν, curvature radius of beam R, width of beam w, prescribed macroscopic strain ε, loading angle θ. $E_{\mathrm{I}} = 1000, E_{\mathrm{II}} = 100, \nu = 0.25, R = 5, w = 1$.

2.1 Periodic Boundary Conditions

To simplify the simulation, a representative cell in the network, which repeats periodically in the material, is picked up. To make corresponding cross sections able to join together, it is necessary to apply periodic boundary conditions on the unit cell. With this method, the deformation of the unit cell can represent that of the full-scale network. The unit cell is picked as shown in the following figure:



where the corresponding cross sections are: (1) and (10), (2) and (9), (3) and (8), (4) and (13), (5) and (12), (6) and (11), (7) and (14). The following equations must be satisfied as periodic boundary conditions (cross sections 1 and 10, for example):

$$u_x(1_i)-u_x(10_i)=u_x(1_1)-u_x(10_1)$$
$$u_y(1_i)-u_y(10_i)=u_y(1_1)-u_y(10_1)$$
$$(i=2,\ldots,n)$$

where the subscript i denotes the ith node on the cross section. The same subscript means that the two nodes are at the same location on the cross sections. u_x and u_y refer to displacement in x (horizontal) and y (vertical) directions respectively. n is the number of nodes on the cross section. This can be realized in ABAQUS by using “Equations” in Interaction. To add a large number of equations in this problem, I used MATLAB code to modify the “.inp” file from ABAQUS to add the periodic boundary conditions.

2.2 Uniaxial Displacement Loading

Uniaxial displacement loading: prescribe displacement only at the center node of each cross section, with direction along the desirable loading direction. The periodic boundary conditions will automatically spread the prescribed displacement at the center node to other nodes on the cross section. According to my previous research[1] analyzing the deformation of simple unit cell, for periodic curved beam network with simple unit cells, the value of prescribed displacement on cross sections should linearly depend on the location of the cross section in the direction of loading so that the mechanical behavior of the unit cell can approximately represent the behavior of full network.

However, the structure of unit cell in this project is more complex than a simple unit cell in either honeycomb or triangular network, but the displacement loading is still doable in a similar way. Here, the displacement loading is applied explicitly on the boundary cross section (1), (4), (10), and (13), which are on the material I and are regarded as “master” cross sections. Specifically, to apply a macroscopic strain ε in the direction θ to the network, the following displacement loading should be applied to a center node of cross sections with number i with initial coordinates (x_0, y_0):

$$u_x(i)\cos\theta + u_y(i)\sin\theta = \varepsilon(x_0(i)\cos\theta + y_0(i)\sin\theta), i = 1,4,10,13$$

which can be realized in ABAQUS using “Equation” in Interaction and reference points. Although this displacement loading is applied explicitly only on the cross sections of material I, it will be automatically spread to other cross sections of material II (regarded as “slave” cross sections) after the constraints described in the following paragraph are applied. In addition, the node (18) is fixed in both directions as the original point. Equations in this part are applied by writing Python script.

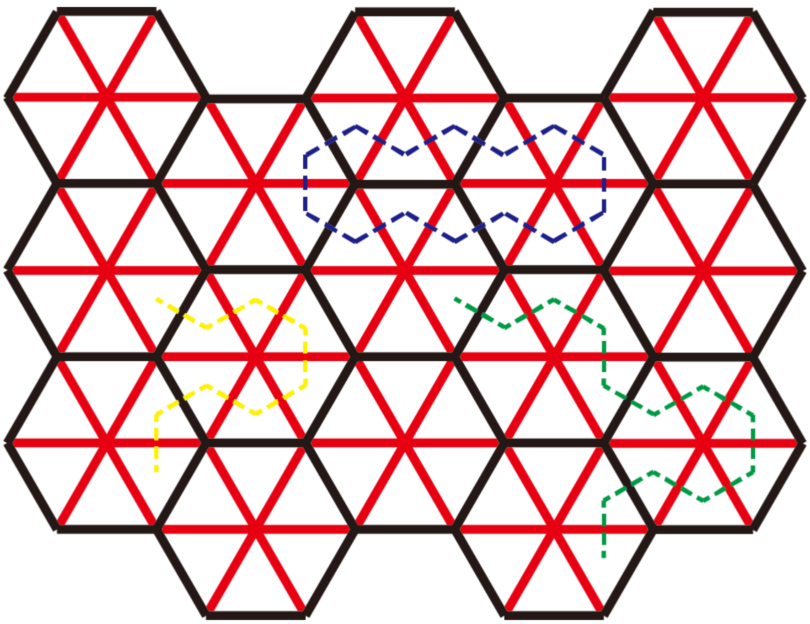
2.3 Constraints of Closed Polygon

(iii) Constraints of closed polygon: a closed triangle always keeps closed after elastic deformation. In this network, there are 6 types of fundamental triangles, in which deformation of cross section nodes related to 5 of the triangles must be constrained to keep the triangles closed. The equation of the remaining one triangle is the linear combination of the first 5 equations, so it can be omitted. Specifically, the following constraints must be applied by ABAQUS equations:

$$
\begin{aligned}
&(\mathbf{u}(5)-\mathbf{u}(4))+(\mathbf{u}(13)-\mathbf{u}(12))=0\\
&(\mathbf{u}(6)-\mathbf{u}(5))+(\mathbf{u}(12)-\mathbf{u}(11))=0\\
&(\mathbf{u}(2)-\mathbf{u}(1))+(\mathbf{u}(10)-\mathbf{u}(9))=0\\
&(\mathbf{u}(3)-\mathbf{u}(2))+(\mathbf{u}(9)-\mathbf{u}(8))=0\\
&(\mathbf{u}(1)-\mathbf{u}(14))+(\mathbf{u}(11)-\mathbf{u}(10))+(\mathbf{u}(7)-\mathbf{u}(6))=0
\end{aligned}
$$

where all the constraints are applied on the center nodes of each cross section, and the coordinates also refer to those of center nodes of each cross section. Equations in this part are also applied by writing Python script.

3. Post-processing: method for calculating the averaged normal stress



After FEA with ABAQUS, we can get the stress response at the cross sections of the unit cell by exporting related data from ABAQUS. The schematic diagram of the network structure is shown as above, where the wavy microstructure is simplified as a straight beam to help understand. Material I and II are shown in black and red respectively. The boundary of one unit cell is marked with blue dashed line. We can distribute the stress on all the cross sections on the macroscopically "continuous" material network.

To study the anisotropy of mechanical properties, we can calculate the macroscopically averaged normal stress with displacement loading along the 2 principle axes (Loading angle θ=0°, 30° respectively. Note that θ=30° and θ=90° are completely the same because of the 6-fold symmetry.). In this two directions, the anisotropy is the most obvious. We need to properly cut the material network and sum the nodal forces on the cut. The dashed line in yellow and green in the diagram above shows how to cut the network for calculating averaged normal stress with θ=0°, 30° respectively. All of the cut is along the boundary of unit cells. The straight line connecting the two ends of yellow and green line is perpendicular to loading direction θ=0°, 30° respectively. After summing up the force component in the loading direction on all of the cut along each dashed line, we need to divide the value by the area of the cut, which is exactly the distance between two ends of each dashed line.

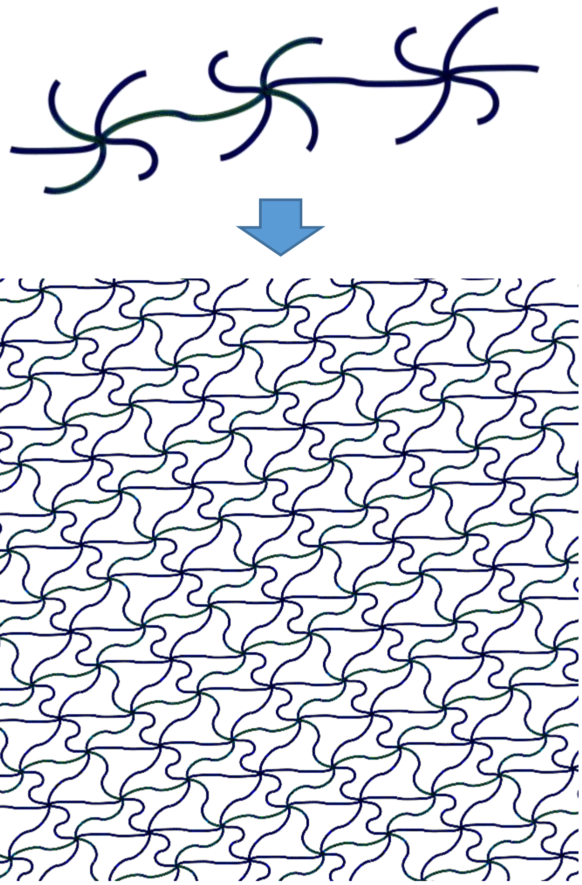
This postprocessing is conducted with MATLAB code after getting output data from ABAQUS. With the method proposed above, the normal stress response under the prescribed displacement (strain) will be known. With the stress output for a certain loading direction and different strain values, we can get the nominal normal stress-strain curve. By comparing the curve in the two loading directions, we can know the degree of anisotropy for the given $E_{\mathrm{II}} / E_{\mathrm{I}} = \beta$. By comparing the anisotropy for different β, we can know the value of β that can minimize the anisotropy of stress-strain curve. In this way, the aim of this project can be reached.

4. Results

4.1 Verification of Correctness of the Model

To ensure that the FEA give convincing results rather than garbage, I verified the correctness by checking continuity of boundary displacement and stress.

Continuity of displacement requires that neighbor unit cells can join together to form the network. This is assured by the periodic boundary conditions and constraints of closed polygons. As long as these equations are correct, the continuity of displacement should be satisfied. Here I simply show a figure with a number of deformed unit cell joined together as follows, as an intuitive verification of displacement continuity: ($\varepsilon = 0.6, \theta = 15°$)



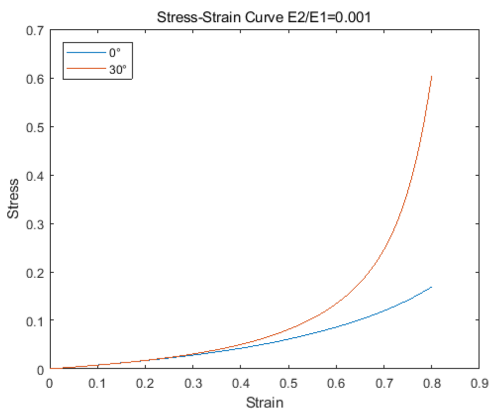
The next step is to verify the continuity of stress. Using the same example as above, I compared the stress S11 on the 9 nodes on the corresponding cross sections (1) and (10). The result is shown as follows:

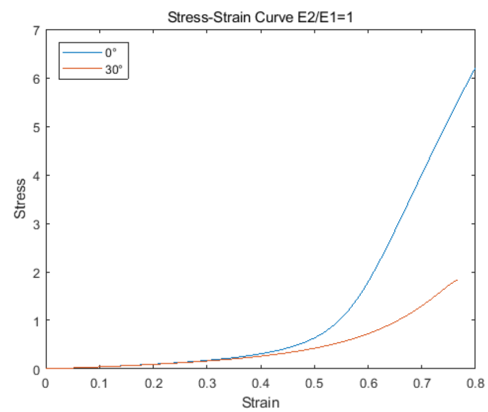
Node # (from top to bottom)	S11 on (1)	S11 on (10)
1	2.65	2.59
2	3.51	3.49
3	4.33	4.33
4	4.48	4.48
5	4.68	4.68
6	4.48	4.48
7	4.33	4.33
8	3.49	3.51
9	2.59	2.65

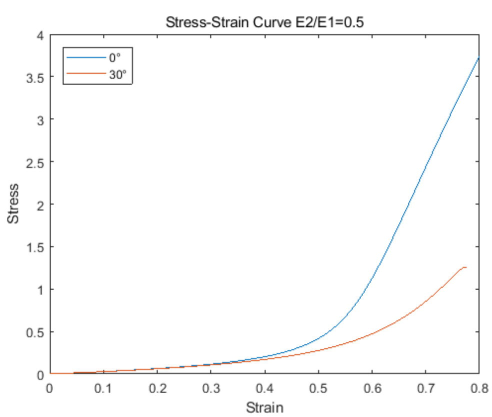
From the table above we learn that the continuity of stress is satisfied with very small error of ~2%.

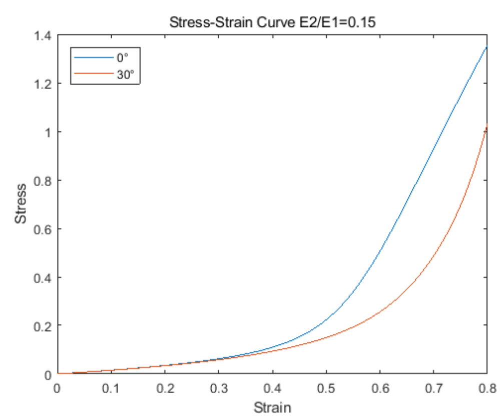
4.2 Stress-strain Curve and Optimization of Isotropy

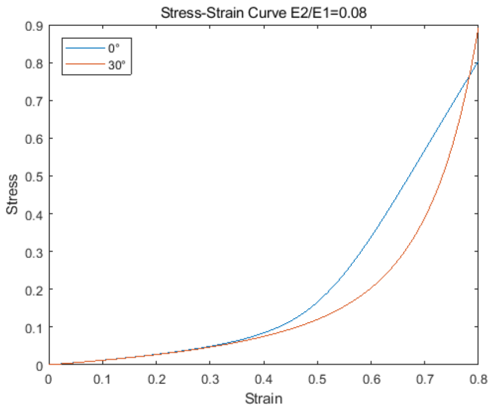
The nominal normal stress-strain curve with loading angle θ=0°, 30° and different values of $\beta = E_{\mathrm{II}} / E_{\mathrm{I}}$ are shown as follows. The first two figures with $\beta = 0.001, 1$ correspond to simple honeycomb and triangular network in the previous research[1]. The curve is consistent with the curve in the previous work.

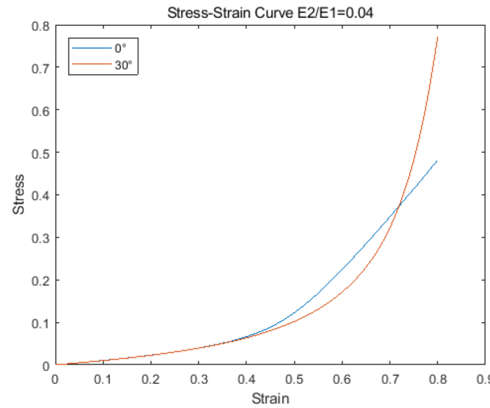


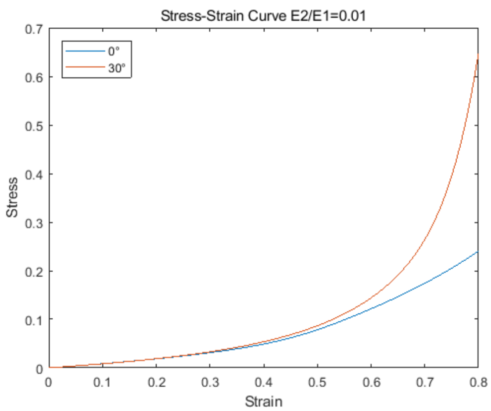


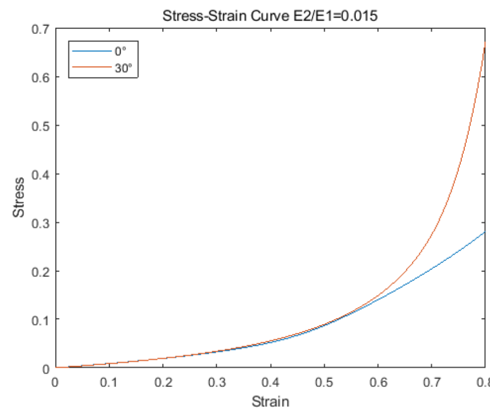


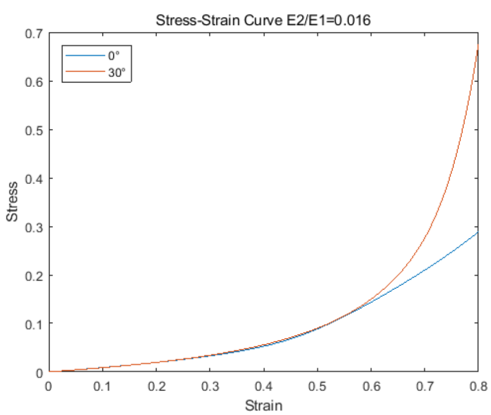


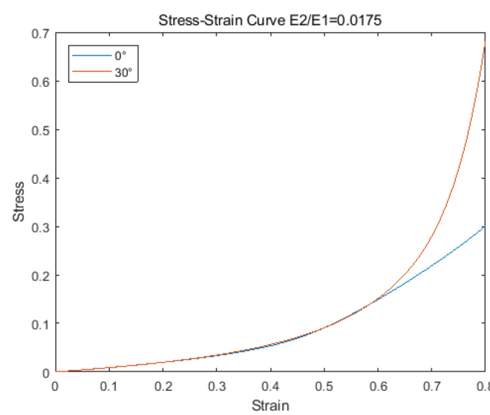




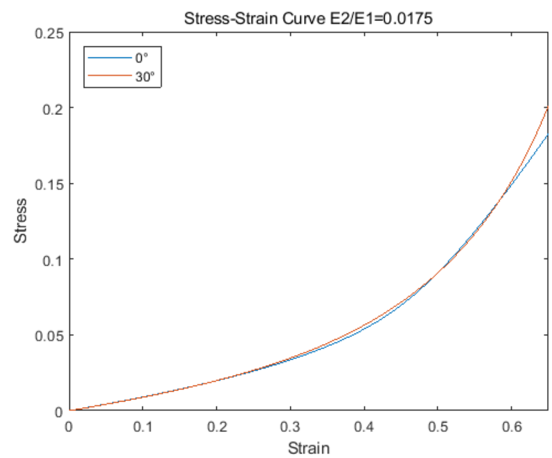








From the stress-strain curves above, we can know that high isotropy is obtained with $\beta \approx 0.0175$, where the stress-strain response in 0°, 30° are almost the same for strain under 60%. In comparison, network with only one material in simple honeycomb ($\beta = 0.001 \approx 0$) or triangular ($\beta = 1$) structure is approximately isotropic for strain under ~30% because of strict isotropy at small deformation for structure with 6-fold symmetry. The network material combining two materials with different Young's modulus can greatly reduce the anisotropy, and the strain limit for approximate isotropy can be doubled to ~60% by choosing proper material property. With $\beta = 0.0175$, the maximum stress difference under the same strain (<60%) is less than 5%. To better visualize the extent of isotropy, an enlarged figure of stress-strain curve of $\beta = 0.0175$ is shown as follows



For practical application, it may be difficult to fabricate such network with wavy structure by two different materials. However, similar effect can be achived in other methods. For example, we can use only one material in the network, but choose 2 different values cross section areas to substitute the "material I" and "material II" in this project. We can estimate that by changing the ratio of cross section area $\beta = A_{II} / A_I$, similar improvement on the isotropy of mechanical properties can be obtained.

5. Conclusion

In this project, I solved a problem about optimizing isotropy of mechanical properties of 2D network materials by using ABAQUS and writing MATLAB and Python code. To simplify this problem about periodic structure, a representative unit cell was modeled. Corresponding periodic boundary conditions, displacement loading, and other geometric constraints were applied by coding in MATLAB and Python in order to make the behavior of unit cell resemble that of the full-scale network. A simple verification of the correctness of this model was shown. The stress-strain curves were be obtained with computation of ABAQUS and post-processing with MATLAB. By adjusting the modulus ratio of two materials to a proper value, the stress-strain responses in the loading direction of two principal axes were made almost the same, which means that apprximate isotropy of stress-strain behavior was achieved. It was found that the network we studied reached the maximum isotropy of stress-strain response when the ratio of Young's modulus of the two materials is around 0.0175. With this parameter, approximate isotropy of stress-strain behavior can be ensured for the network material I designed with applied strain up to 60%.

Reference

[1] Enrui Zhang, Yuan Liu, and Yihui Zhang. *A computational model of bio-inspired soft network materials for analyzing their anisotropic mechanical properties*, 2017, submitted.