Mechanics of Composites

Project report

on

Finite Element Analysis of Uniform and Non-Uniform Unidirectional Continuous Fibre Composites using Abaqus

Department of Mechanical Engineering

Indian Institute of Technology, Kharagpur

Submitted to: -

Prof. Atul Jain

Submitted by

Pankaj Suhas Pandit (21ME63R08)

Dhiraj Dharmadip Raut (21ME63R31)

ACKNOWLEDGEMENT

We would like to thank sincerely our professor **Dr. Atul Jain**, Dept. of Mechanical Engineering, Indian Institute of Technology, Kharagpur for providing us this opportunity and motivate us to make best use of our knowledge and solve a problem in a professional manner. We are obliged for his cooperation and support during the process and is one of the main reasons we could complete this report with a satisfactory conclusion

Finally, we are also thankful to each other for our continued moral and material support throughout the preparation of this project.

Table of Contents

Abstract

This project aims at studying and verifying various formulations and assumptions taken for a unidirectional reinforced fibre composite. The model is created in the CAE software Abaqus. The model has uniformly and randomly(nonuniformly) arranged fibre inside the matrix to simulate an actual unidirectional composite. The material selected for the matrix is **BSL914c Epoxy** and for the fibre is **E-Glass 1200 tex**. The model first checks for the iso-strain condition and iso-stress conditions and verifies the results with the formulations called Linear rules of mixture and inverse rule of mixture respectively. The iso-stress conditions are further verified in two different directions which when compared resulted in the transverse isotropy of the composite material.

Problem Statement

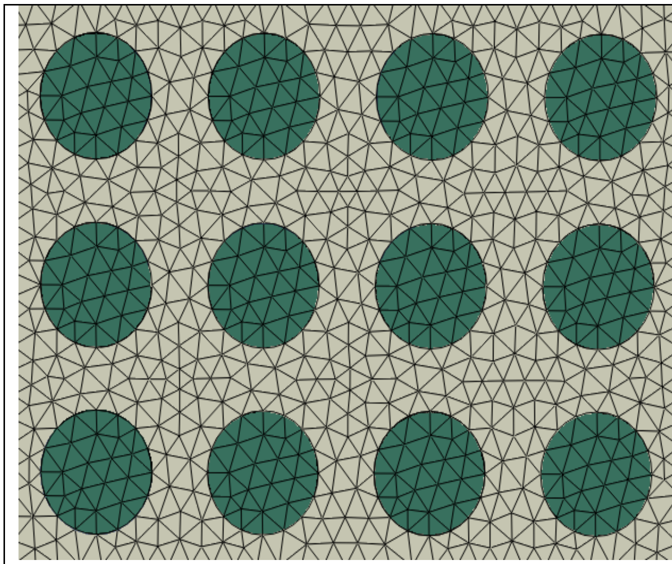
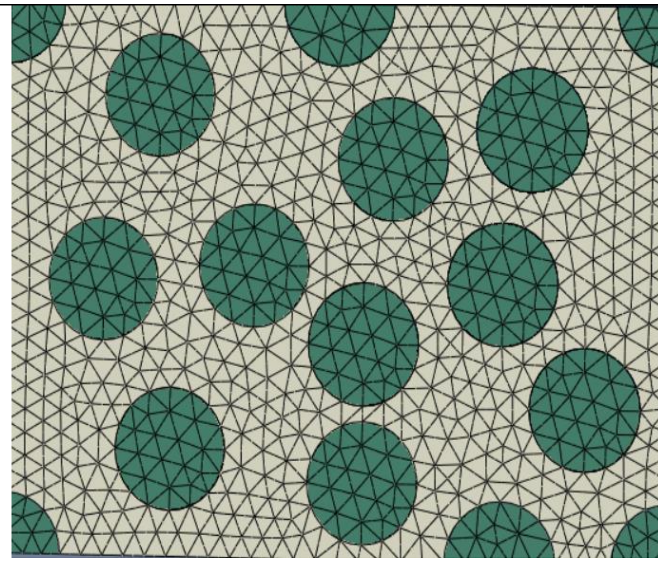
	

Fig. Uniform Fibre composite Case	Fig. Non-Uniform Fibre composite Case

We considered matrix and fibre material same for both Uniform and Non-Uniform models.

We know the following material properties: -

- Young's modulus of matrix and fibre.
- Poisson's ratio of matrix and fibre.
- Considered 12 number of fibres with each of diameter 10mm.

From the above information we have calculated the Volume Fraction of Fibre and Matrix.

Our objective is to find out the following results and compare the results of uniform-nonuniform models: -

1. Calculate the Effective Young's Modulus of uniform and nonuniform composite considering iso-strain condition and iso-stress condition
2. Compare the calculated Effective Young's Modulus with micromechanics formulae discussed in class.
3. Compare the results of stress, strain and displacement of uniform and nonuniform composites through graphical images.
4. Proving the transverse isotropy of the designed composite
5. Calculate the Effective Shear Modulus of uniform and nonuniform composite considering Shear-Isostress Loading condition.

Model Properties

- Unidirectional Fibre Composite
- **Size of 3D Model :**

Length (L)	60 mm
Width (W)	45 mm
Depth (H)	100 mm

- **Fibre Properties :**

Fibre Material	E-Glass 1200 tex
Young's Modulus €	80000 MPa
Poisson's Ratio (ν)	0.2
Fibre Diameter	10 mm
Fibre Length	100 mm

- **Matrix Properties :**

Matrix Material	BSL914c Epoxy
Young's Modulus (E)	4000 MPa
Poisson's Ratio (ν)	0.35

- **Estimation of Volume Fraction of Fibre and Matrix :**

Volume Fraction of Fibre :

$$v_f = \frac{N\pi D^2}{4LW} = \frac{12 * \pi * 10^2}{4 * 60 * 45} = 0.3488$$

Volume Fraction of Matrix :

$$v_m = 1 - v_f = 0.6512$$

Model Development

The development of the model is the most important step in the project. We have total 6 models, 3 from Uniform fibre case and 3 from Non-Uniform fibre case. First we need to create one model by following the steps of Model Development mentioned below and just by changing Loading and boundary conditions we will obtain remaining models.

- ## Step 1 - Part Module :

The first step in Abaqus is to design the part. In the beginning, We have created a square with (0,0) and (60,45) coordinates and named it as matrix. Similarly created another three parts that is complete, half and quarter cylinder which are named as fibre1, fibre2 , fibre3 respectively.

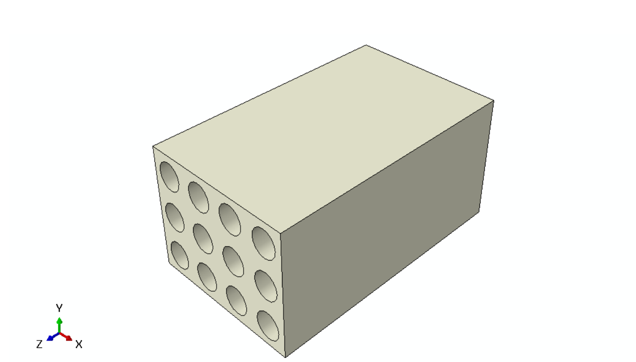
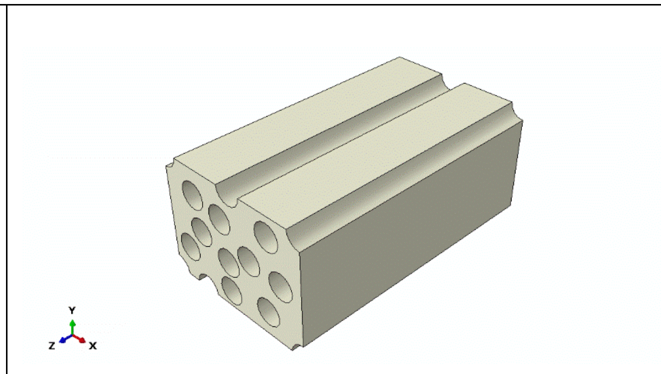
	

Fig. Matrix for Uniform Case	Fig. Matrix for Non-Uniform Case

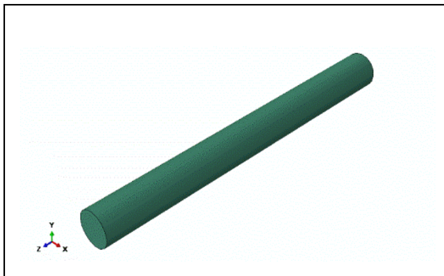
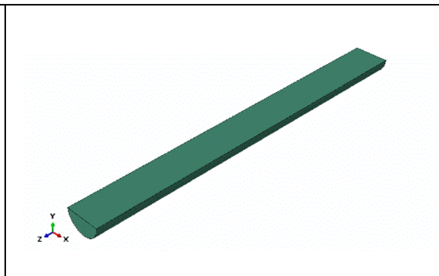
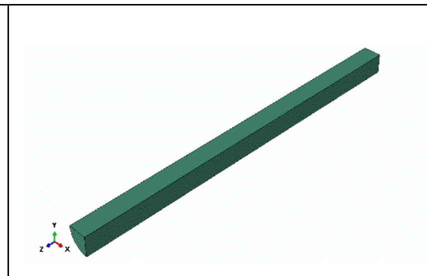
		

Fig. Fibre1	Fig. Fibre2	Fig. Fibre3

- ## Step 2 - Property Module :

We have used the required Elastic properties of Fibre and Matrix and created two materials naming FibreMaterial and MatrixMaterial. The next step is section generation. Since there have to be 2 sections - one for the fibre and the other for the matrix, so I have made 2 sections named 'Fibre' and 'Matrix' and applied material properties accordingly. The next step is assigning of section. For this, Expand each part created, select appropriate section under section assignment for each part carefully. This can be confirmed by observing change in colour after section assignment.

- **Step 3 - Assembly Module :**

We have to create instances and make sure to check on Auto-offset and keep independent (mesh on instances). Now, selected all the parts to be assembled and have used translate instances feature to assemble final model.

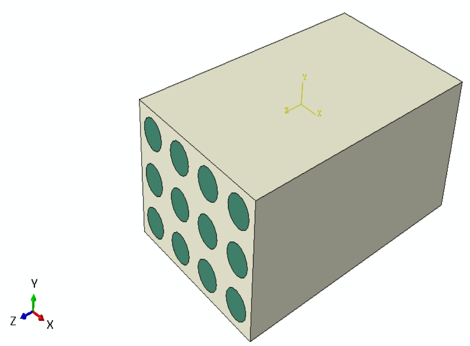
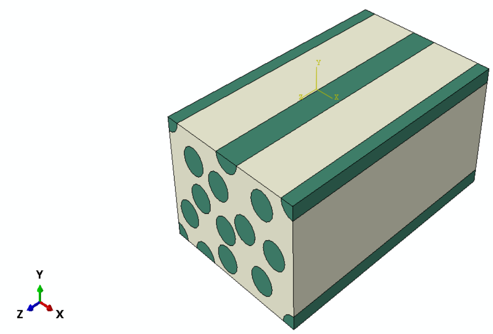
	

Fig. Assembly Model for Uniform Case	Fig. Assembly Model for Non-Uniform Case

- **Step 4 - Step Module :**

This module is to define the problem that is being solved. We set it to a Static General Problem and continued.

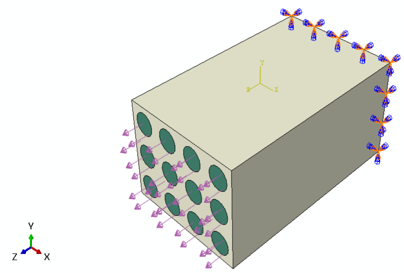
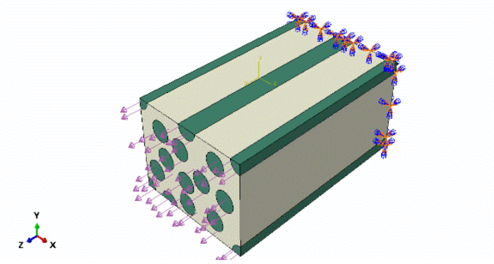
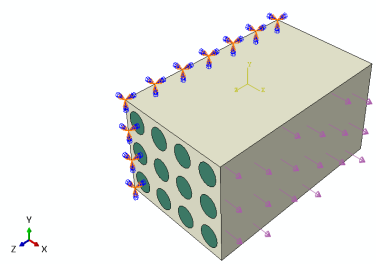
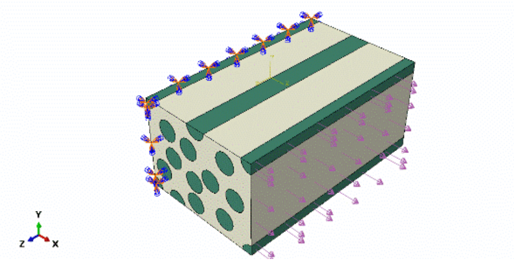
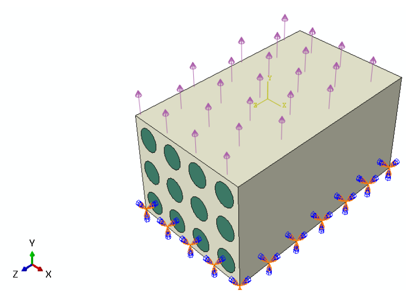
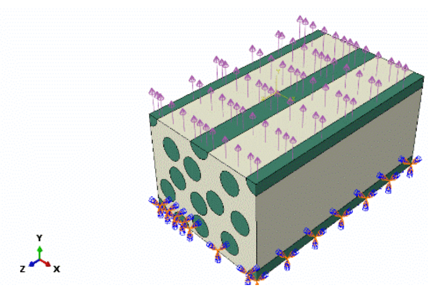
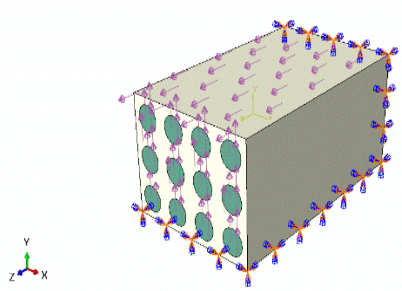
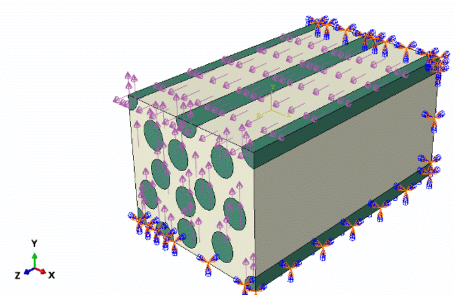
- **Step 5 - Interaction Module :**

Here, we have to define Master surface and Slave surface. This module is to create proper constraint between fibre and matrix. For this ‘Tie’ option is used. Select outside of Fibre cylinders as master i.e., independent surface and select inner surfaces of matrix cuboid as slave i.e., dependent surface. Once it is done, we can observe yellow rings which ensures tying is done properly between them.

- **Step 6 - Load Module:**

In this module loads have to be applied and Boundary conditions have to be defined. Since there was no stress option under mechanical load in Abaqus ,We have applied a pressure of magnitude 10MPa in the negative sense along the z-direction. This is a good work around of the problem for considering stress. The load is applied to both the faces of matrix and fibre facing along the z-direction. Once done, it will show direction of load with pink arrows.

Now, we need to defined Boundary conditions. For this, we have Encastrated the Z face on the back side for the Model 1 of both Uniform and Non-Uniform case. Similar way it is done for other models on different faces.

Cases	Uniform Fibre composite case	Non-Uniform Fibre Composite Case
Case 1: Isostrain Loading along Z direction		
Case 2: Isostress Loading along X direction		
Case 3: Isostress loading along Y direction		
Case 4: Shear-Isostress loading on Z face and Y face		

- **Step 7 - Mesh Module :**

The meshing is a crucial step in solving the problem. In this module meshing is done
Since the domain is a pretty simple one, I have selected linear meshing using CPS4R and CPS3 elements. For this, use seed part instances and set Approximate mesh global size as 2. White rings will appear on the model assembly. Under assign mesh control, Tet has been selected as Element shape and Technique kept as free. Mesh needs to be verified for proper results. We obtained Analysis warning as 0.015444% and 0% analysis errors.

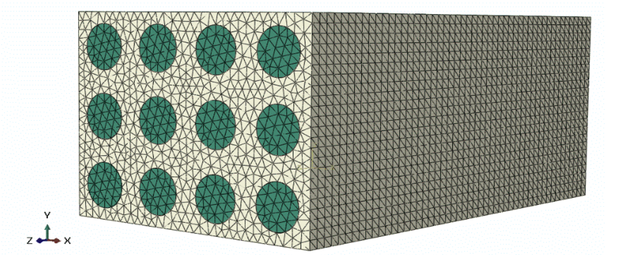
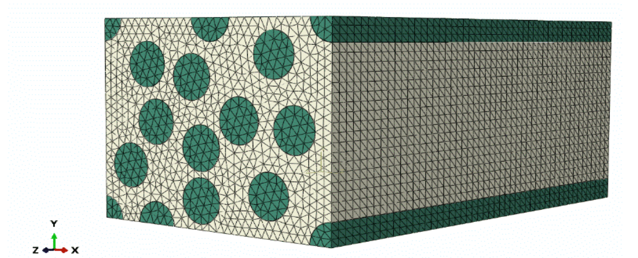
	

Fig. Uniform Fibre Mesh **(No. of Elements : 231432)**	Fig. Non-Uniform Fibre Mesh **(No. of Elements : 233478)**

- **Step 8 - Job Module :**

We have created a job for this particular model and Finally the model is sent for analysis by submitting it. This will create a job.inp file which will give all details like nodes, BCs, etc. This .inp file can be used further for creating other models.

- **Step 9 - Visualization Module :**

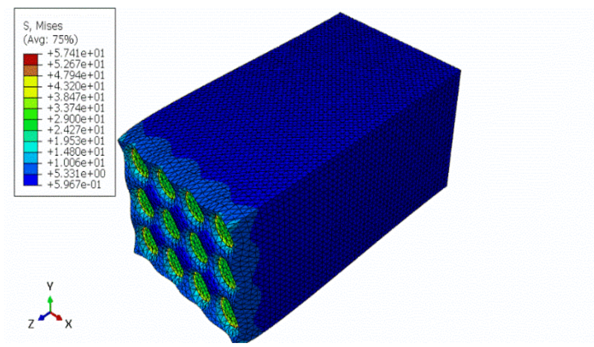
Finally after analysis, the results has been checked using this module by clicking on particular .odb file. Results are visualized through this and necessary data is extracted through report.rpt file. The results for each model are discussed ahead in the document.

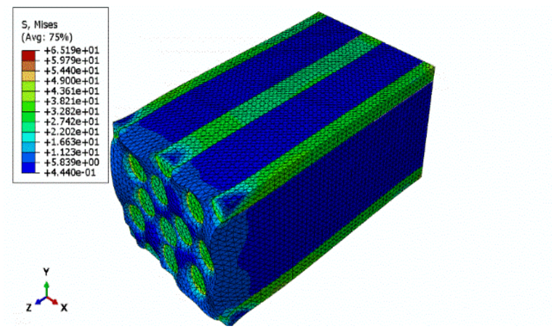
- **Step 10 :**

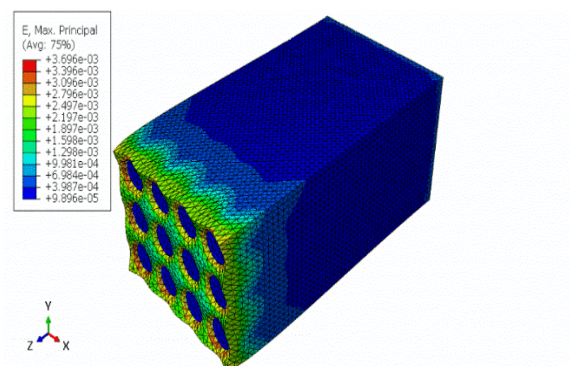
We have created other models from Previous Model by changing Loading conditions and boundary condition.

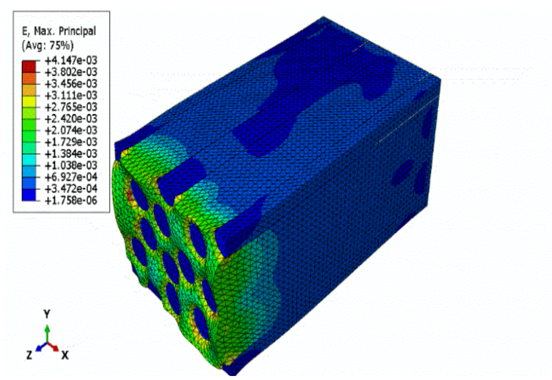
Plot Contours (Graphical Results)

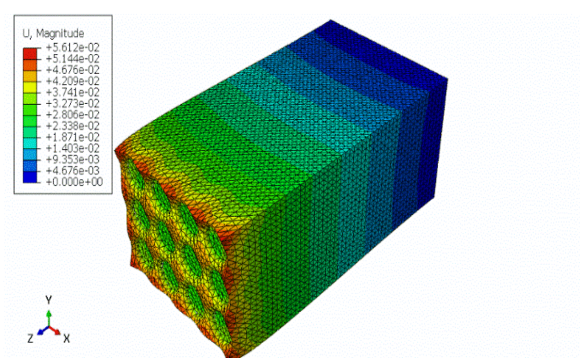
Case 1 - Isostrain condition (load applied along Z direction):

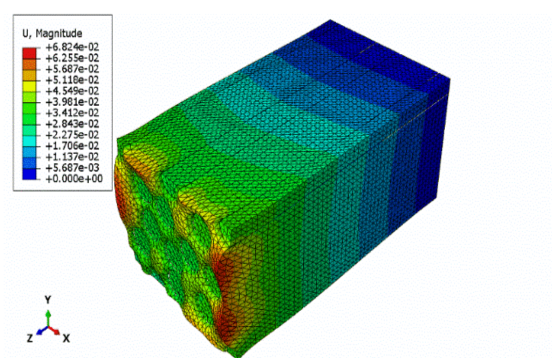
<table>
<tr><th></th><th>Uniform Fibre Continuous Composite</th><th>Non-Uniform Fibre Continuous Composite</th></tr>
<tr><td>Stress</td><td>


</td><td>


</td></tr>
<tr><td>Strain</td><td>


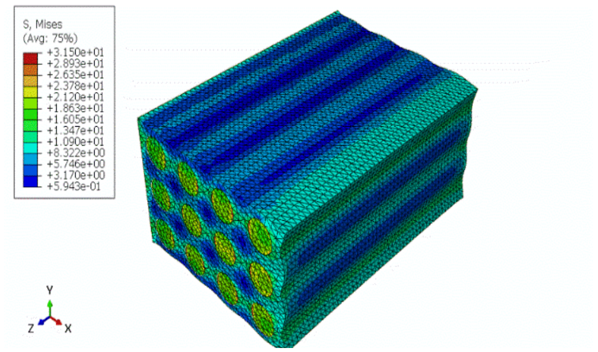
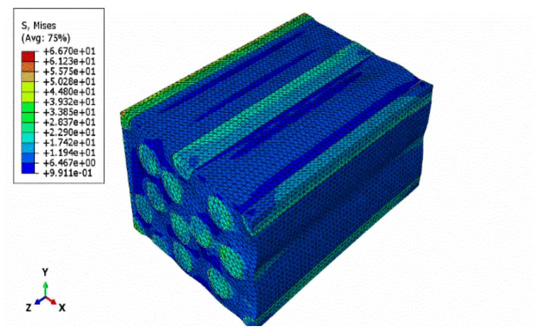
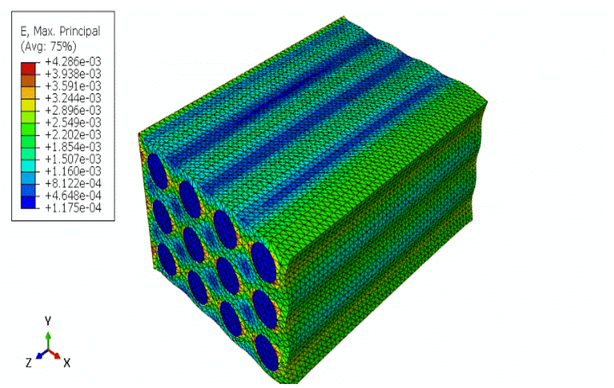
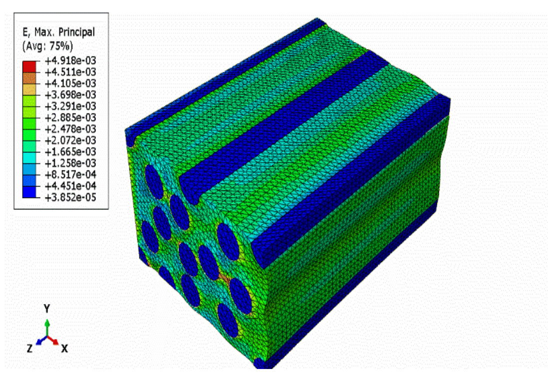
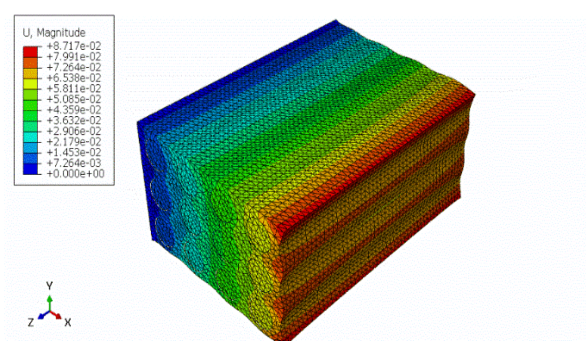
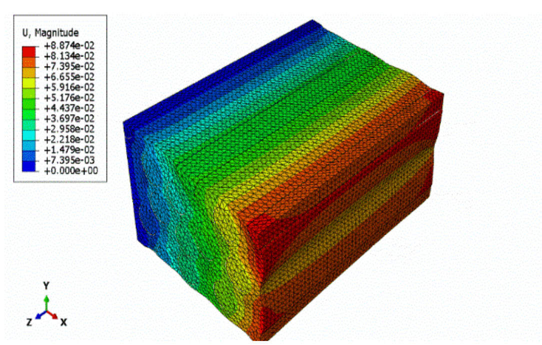
</td><td>


</td></tr>
<tr><td>Deformatio
n</td><td>


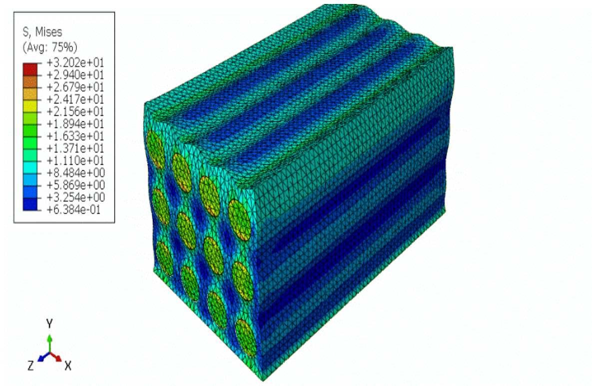
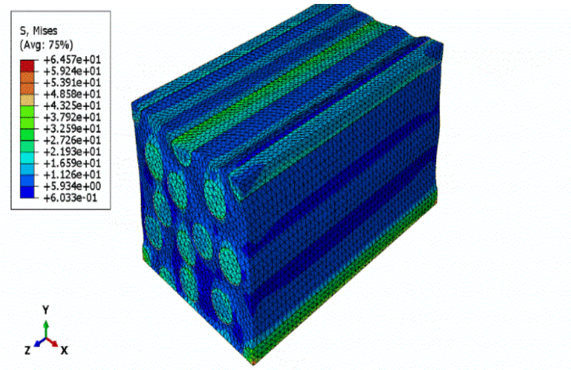
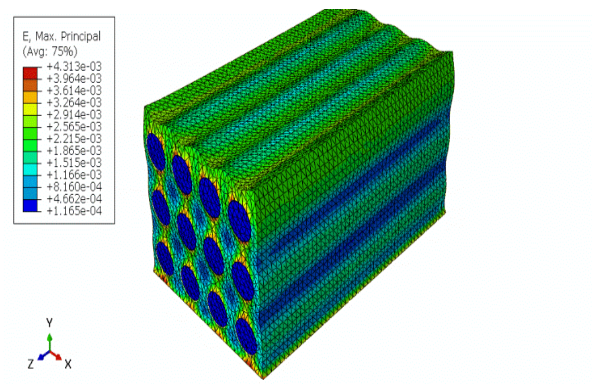
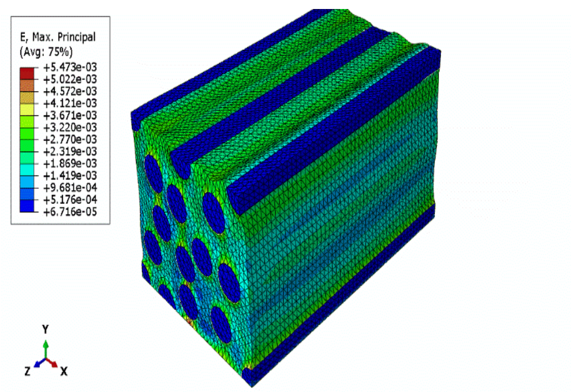
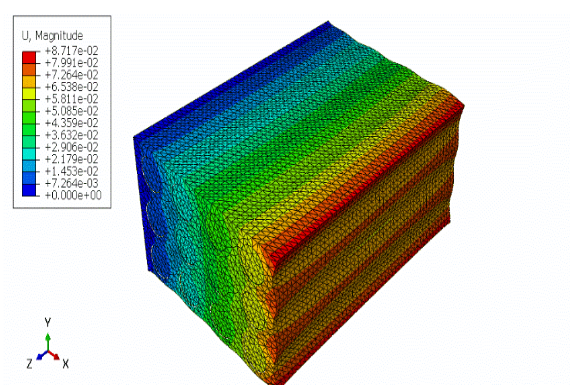
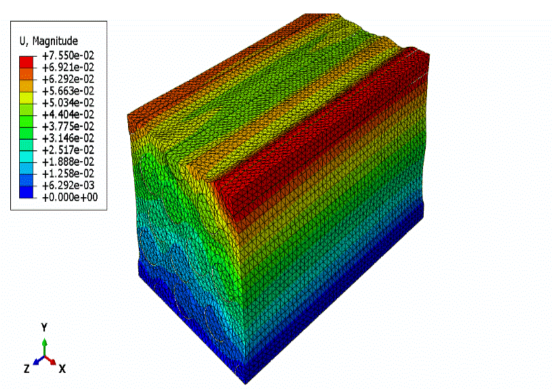
</td><td>


</td></tr>
</table>

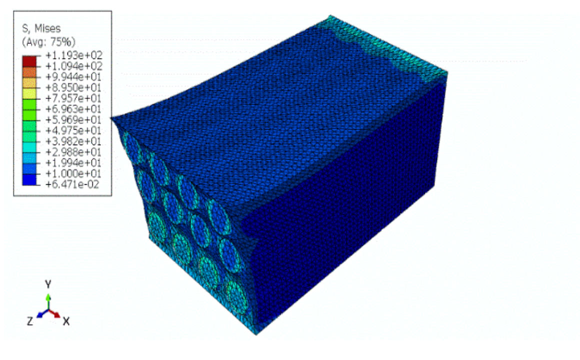
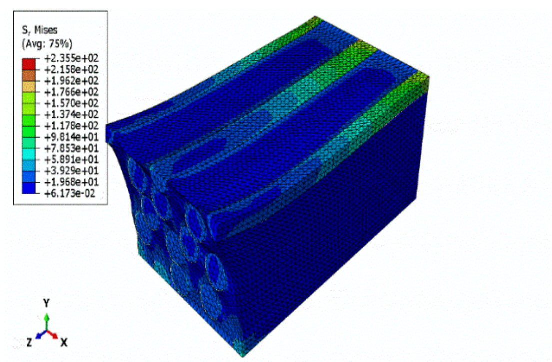
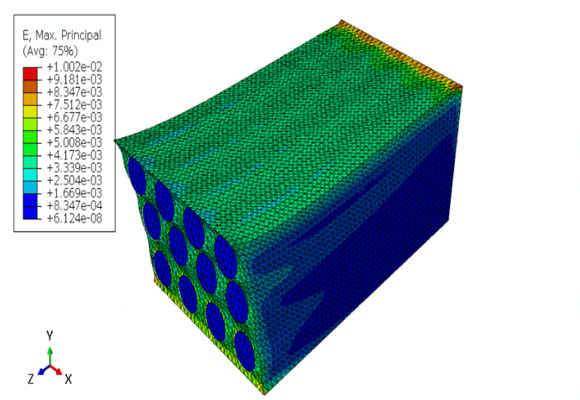
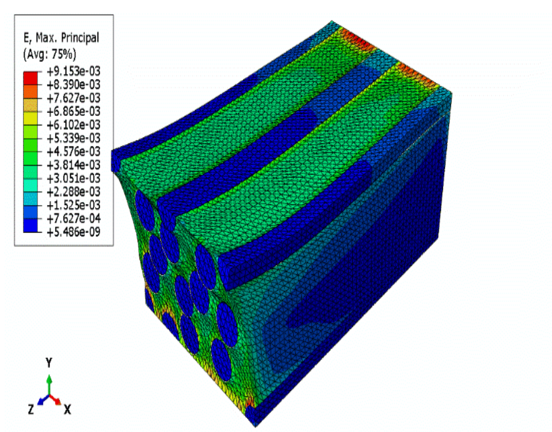
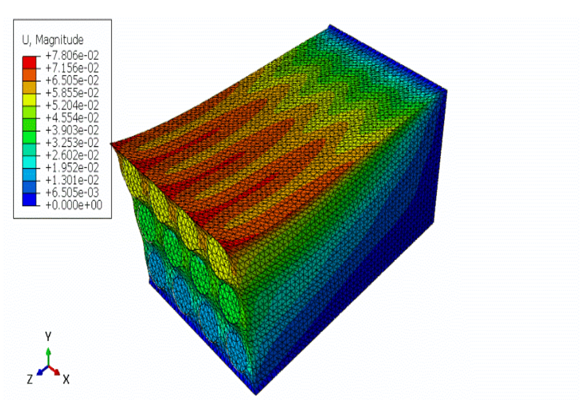
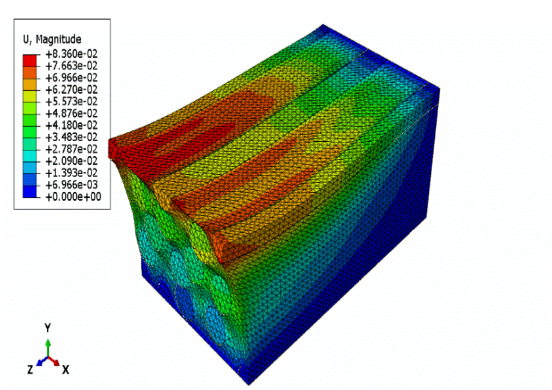
Case2: Isostress condition 1 (load applied along X direction)

<table>
<tr><th></th><th>Uniform Fibre Continuous Composite</th><th>Non-Uniform Fibre Continuous Composite</th></tr>
<tr><td>Stress</td><td>
</td><td>
</td></tr>
<tr><td>Strain</td><td>
</td><td>
</td></tr>
<tr><td>Deformation</td><td>
</td><td>
</td></tr>
</table>

Case3: Isostress condition 2 (load applied along Y direction)

<table>
<tr><th></th><th>Uniform Fibre Continuous Composite</th><th>Non-Uniform Fibre Continuous Composite</th></tr>
<tr><td>Stress</td><td></td><td></td></tr>
<tr><td>Strain</td><td></td><td></td></tr>
<tr><td>Deformatio n</td><td></td><td></td></tr>
</table>

Case4: Shear-Isostress condition (Surface traction applied on Z face and Y face)

	Uniform Fibre Continuous Composite	Non-Uniform Fibre Continuous Composite
Stress		
Strain		
Deformation		

Property estimation using micro-mechanics theory

- Young's Modulus of Fibre,E_f = 80 GPa
- Young's Modulus of Matrix, E_m = 4GPa
- Poisson's Ratio of Fibre, v_f= 0.2
- Poisson's Ratio of Matrix, v_m = 0.35
- Volume Fraction of fibre, Vf = 0.35

Now, using **Isostrain condition :**

$$E_{1,C} = E_f V_f + E_m V_m$$

$$E_{1,C} = 80*0.35 + 4*0.65$$

$$\boldsymbol{E_{1,C} = 30.6\ \mathrm{GPa}}$$

$$v_{12} = V_m v_m + V_f v_f$$

$$v_{12} = 0.65*0.35 + 0.35*0.2$$

$$\boldsymbol{v_{12} = 0.2975}$$

Now, using **Isostress condition** :

$$E_{2,C} = \frac{E_f E_m}{E_m V_f + E_f V_m}$$

$$E_{2,C} = \frac{80*4}{4*0.35+80*0.65}$$

$$\boldsymbol{E_{2,C} = E_{3,C} = 5.99\ \mathrm{GPa}}$$

Now, using **Shear-Isostress condition** :

For an isotropic material, $G = \frac{E}{2(1+v)}$

$$G_f = \frac{80}{2(1+0.2)} = 33.33\ \mathrm{GPa}$$

$$G_m = \frac{4}{2(1+0.35)} = 1.4814\ \mathrm{GPa}$$

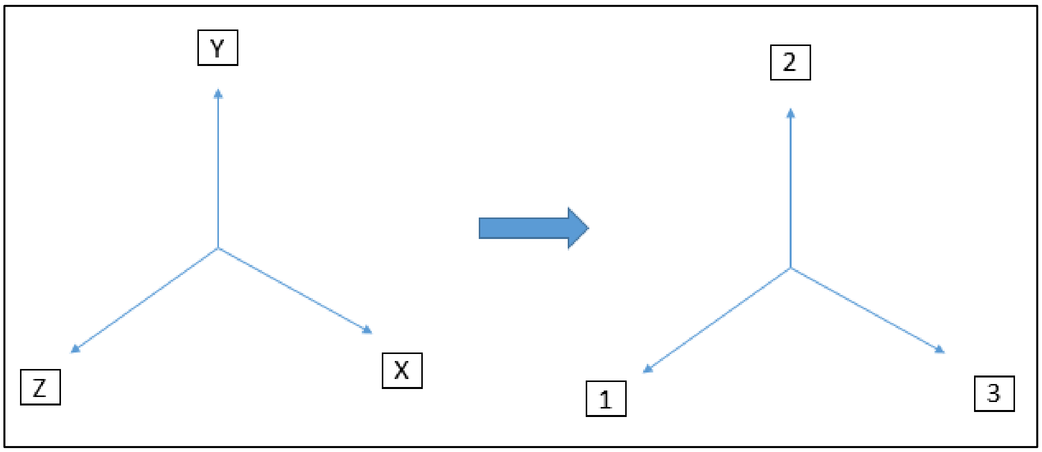
$$\boldsymbol{G_{12} = \frac{G_f G_m}{G_m V_f + G_f V_m} = 2.22\ \mathrm{GPa}}$$

Property estimation From Abaqus

I tried to find the strain and stress distribution for each element at centroid. Once found, the weighted average strain and stress can be determined. Dividing the weighted average stress with strain can provide me the value for Young's Modulus. (Here load is along X direction)

$$\boldsymbol{E_{11}} = \frac{\boldsymbol{\sigma_{11,}avg}}{\boldsymbol{\varepsilon_{11,}avg}}$$

Axial Notations taken :



That means, subscripts of notations: Z (33) => 1 , Y (22) => 2, X (11) => 3

Now, **for Isostrain condition (load applied along Z direction):**

$$E_{1,c} = \frac{\Sigma E_{fi} V_{fi}}{\Sigma V_i} + \frac{\Sigma E_{mi} V_{mi}}{\Sigma V}$$

$E_{1,c}$ = 27732.004 +2794.959 = **30526.963 MPa** (For ***Uniform*** Fibre Composite)

$E_{1,c}$ = 27731.394+2790.157 = **30521.550 MPa** (For ***Non-Uniform*** Fibre Composite)

$$v_{13} = \frac{\Sigma v_{13,i} V_i}{\Sigma V_i} \text{ where, } v_{13,i} = \frac{\varepsilon_{34\mathrm{i}}}{\varepsilon_{1\mathrm{i}}}$$

$v_{12} = \frac{\Sigma v_{12,i} V_i}{\Sigma V_i}$ = **0.2762** (For ***Uniform*** Fibre Composite)

$v_{12} = \frac{\Sigma v_{12,i} V_i}{\Sigma V_i}$ = **0.2819** (For ***Non-Uniform*** Fibre Composite)

Now, **for Isostress condition 1 (load applied along X direction) :**

$$E_{2,C} = \frac{\sigma_{2,C}}{\varepsilon_{2,c}} \quad \text{where, } \sigma_{2,C} = \frac{\Sigma \sigma_{2,Ci} V_i}{\Sigma V_i} \quad \text{and } \varepsilon_{2,c} = \frac{\Sigma \varepsilon_{2,fi} V_{fi}}{\Sigma V_{fi}} + \frac{\Sigma \varepsilon_{2,mi} V_{mi}}{\Sigma V_{mi}}$$

$E_{2,C} = \frac{10.0000}{0.0019}$ = **5250.00** MPa (For ***Uniform*** Fibre Composite)

$E_{2,C} = \frac{10.0000}{0.00194}$ = **5251.11** MPa (For ***Non-Uniform*** Fibre Composite)

Similarly, **for Isostress condition 2 (load applied along Y direction) :**

$$E_{3,C} = \frac{\sigma_{3,C}}{\varepsilon_{3,c}} \quad \text{where, } \sigma_{3,C} = \frac{\Sigma \sigma_{3,Ci} V_i}{\Sigma V_i} \quad \text{and } \varepsilon_{3,c} = \frac{\Sigma \varepsilon_{3,fi} V_{fi}}{\Sigma V_{fi}} + \frac{\Sigma \varepsilon_{3,mi} V_{mi}}{\Sigma V_{mi}}$$

$E_{3,C} = \frac{10.0000}{0.00176}$ = **5670** MPa (For ***Uniform*** Fibre Composite)

$E_{3,C} = \frac{10.0000}{0.00183}$ = **5461.74** MPa (For ***Non-Uniform*** Fibre Composite)

Now, **for Shear-Isostress condition (Surface traction applied on Z face and Y face) :**

$$G_{12} = \frac{\tau_{12,C}}{\Upsilon_{12,c}} \quad \text{where, } \tau_{12,C} = \frac{\Sigma \tau_{12,Ci} V_i}{\Sigma V_i} \quad \text{and } \Upsilon_{12,c} = \frac{\Sigma \Upsilon_{12,fi} V_{fi}}{\Sigma V_{fi}} + \frac{\Sigma \Upsilon_{12,mi} V_{mi}}{\Sigma V_{mi}}$$

$G_{12,C} = \frac{4.2700}{0.00228}$ = **1880.00** MPa (For ***Uniform*** Fibre Composite)

$G_{12,C} = \frac{4.22217}{0.002229}$ = **1893.90** MPa(For ***Non-Uniform*** Fibre Composite)

Comparison Table between Results

Cases		**Theoretical**	**Uniform (Abaqus)**	**Non-Uniform (Abaqus)**
Case1: isostrain condition (load applied along Z direction):	$E_{1,C}$ (MPa) ν_{12}	30508.8 0.29768	30526.963 0.2762	30521.550 0.2819
Case2: isostress condition 1 (load applied along X direction)	$E_{2,C}$ (MPa)	5982.3	5250.0	5251.11
Case3: isostress condition 2 (load applied along Y direction)	$E_{3,C}$ (MPa)	5982.3	5670	5461.74
Case4: Shear-Isostress condition (Surface traction applied on Z face and Y face)	G_{12} (MPa)	2220	1880	1893.90

Conclusion

- In Case1: By comparing the effective elastic modulus $E_{1,C}$ and ν_{12} calculated from theoretical formulation and from the Abaqus FEA, the values are close to each other. This verifies the iso-strain conditions in the designed composite.
- In Case2: By comparing the effective elastic modulus $E_{2,C}$ calculated from theoretical formulation and from the Abaqus FEA, the values are close to each other. This verifies the iso-stress conditions in the designed composite in the x-direction.
- In Case3: By comparing the effective elastic modulus $E_{3,C}$ calculated from theoretical formulation and from the Abaqus FEA, the values are close to each other. This verifies the iso-stress conditions in the designed composite in the y-direction.
- From Case2 and Case3: comparing FEA results of $E_{2,C}$ and $E_{3,C}$, it can be observed that the results are almost comparable. This verifies the transverse isotropy of the designed composite models.
- In Case4: By comparing the effective shear modulus $G_{12,C}$ calculated from theoretical formulation and from the Abaqus FEA, the values are close to each other. This verifies the Shear-Isostress condition in the designed composite.