

FINITE ELEMENT ANALYSIS OF A TWO DIMENSIONAL COMPOSITE BODY FOR DIFFERENT BOUNDARY CONDITIONS

Zannatul Ferdous¹, Jayanti Das² and M. Ashiqur Rahman³

¹University of Manitoba, Winnipeg, Canada,

²Seoul National University, South Korea.

³Bangladesh University of Engineering and Technology, Dhaka, Bangladesh.

ABSTRACT

A self derived finite element code is established here to analyze the stress distribution over a two-dimensional rectangular composite body. Finite Element Analysis is a versatile numerical method for analyzing different engineering problems. Composites offer several advantages over conventional materials, because monolithic metals and their alloys cannot always meet the demands of today's advanced technologies. Composite materials may include improved strength, stiffness, fatigue and impact resistance, thermal conductivity, corrosion resistance etc. In many cases, using composites is more efficient. But in case of unidirectional laminae composite body shows low stiffness and strength properties in the transverse direction. Therefore, in most cases angle lamina offers better properties. The present study also finds responses of a rectangular composite body for different boundary conditions.

Keywords: Finite Element Method, Two Dimensional Elasticity Problem, Unidirectional Composite, Angle Lamina Composite.

1. INTRODUCTION

A composite is a structural material that consists of two or more combined constituents that are combined at a macroscopic level and not soluble in each other. Traditional monolithic metals and their alloys cannot always meet the demands of today's advanced technologies. In many cases, using composites is more efficient. Composites offers several advantages over conventional materials, such as, improved strength, stiffness, fatigue and impact resistance, thermal conductivity, corrosion resistance etc. Again a laminate of a composite material does not consist only of unidirectional laminae(UD) because of their low stiffness and strength properties in transverse direction. Therefore, in most laminates, some laminae are placed at an angle i.e. angle laminae are used [1].

In this study, a rectangular elastic body is made of composite material, loaded in tension is assumed for analysis. The unidirectional lamina and angle lamina both are considered for this study. A two dimensional (2D) rectangular body is selected because 2D rectangular bodies are the most commonly used structures. Different boundary conditions are applied for UD composite and angle lamina composite (Figs 2, 3, 4, 5). Finite Element Method (FEM), a versatile numerical method, is chosen here to analyze the stresses.

2. MATHEMATICAL MODELING

Some properties are assumed to analyze the problem, such as, the body is in plane stress condition; fibers are uniformly distributed; perfect bonding exists between fiber & matrix; no residual stress; matrix is free of void etc. [1]. Consider a rectangular body made of composite material, having a length L , width W and thickness t . Thickness is very small compared to its length and width. For angle lamina angular orientation of fiber is considered to vary from 15° to 90° at an interval of 15° . For boundary condition 1, -15° to -90° angular orientation are also considered. To model this problem the tensile stress is transformed into concentrated load at two nodes (figure 1). The horizontal and vertical displacements are denoted by u & v respectively. The developed stresses are analyzed by finite element analysis using rectangular element and compare the stresses for different boundary conditions.

In order to get some numerical results, some specific values are considered for the dimensions of the body and the properties of Graphite/epoxy composite are used. Here, we considered, length, $L=254$ mm, width, $W=152.4$ mm, thickness, $t=5.08$ mm, applied uniform surface tensile load, $p_r=68.88$ MPa. For graphite/epoxy composite, longitudinal Young's modulus, $E_1=181$ GPa; transverse Young's modulus, $E_2=10.3$ GPa; shear modulus, $G=7.17$ GPa; major Poisson's ratio, $\nu_{12}=0.28$; minor Poisson's ratio, $\nu_{21}=0.01589$. Four different

boundary conditions are used. The first boundary condition for UD composite and also for homogeneous material is also known as "Constant Stress" problem. A self developed computer code in C language has been used to formulate local and global stiffness matrices and for incorporation of boundary conditions. Then matlab has been used to solve simultaneous linear equations and nodal displacements were found. By using the nodal displacements, the parameters, like stress, strain etc were calculated. C programming language was used for calculating these parameters.

3. SOLUTION BY FEM

3.1 Stress-strain Relations

Since the thickness of the body is almost negligible compared with other two dimensions, the analysis of thin body loaded in the plane of the body can be made using plane stress assumption.

In plane stress distribution, it is assumed that

$$\sigma_{yy} = \tau_{zx} = \tau_{zy} = 0$$

The strain and stress vectors are expressed as

$$\{\epsilon\} = \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{Bmatrix}, \{\sigma\} = \begin{Bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \tau_{xy} \end{Bmatrix}$$

Strains are defined as, $\epsilon_{xx} = \delta u / \delta x$, $\epsilon_{yy} = \delta v / \delta y$, $\gamma_{xy} = \delta u / \delta x + \delta v / \delta y$, where, u and v are the nodal displacements along horizontal and vertical direction respectively.

The strain vector can be calculated as,

$$\{\epsilon\} = [B] * \{u\} \quad (1)$$

Here $[B]$ is a 3×8 matrix for rectangular composite element (unidirectional composite and angle lamina) [2, 3]. For unidirectional composite, the material property matrix $[D]$ is expressed as follows, [1], which is similar that for isotropic materials.

$$[D] = \begin{bmatrix} E_1 / (1 - \nu_{12} \nu_{21}) & \nu_{12} \times E_2 / (1 - \nu_{21} \nu_{12}) & 0 \\ \nu_{12} \times E_2 / (1 - \nu_{21} \nu_{12}) & E_2 / (1 - \nu_{21} \nu_{12}) & 0 \\ 0 & 0 & G_1 \end{bmatrix} \quad (2)$$

Here,

E_1 = Longitudinal Young's modulus of elasticity

E_2 = Transverse Young's modulus of elasticity

ν_{12} = Major Poisson's ratio

G_1 = Shear modulus of rigidity.

But the material property matrix $[D]$ is completely different for angle lamina composites. For a 2-dimensional angle lamina the material property matrix $[D]$ can be expressed as follows [1] (applying plane stress assumption):

$$[D] = \begin{bmatrix} \overline{Q}_{11} & \overline{Q}_{12} & \overline{Q}_{16} \\ \overline{Q}_{12} & \overline{Q}_{22} & \overline{Q}_{26} \\ \overline{Q}_{16} & \overline{Q}_{26} & \overline{Q}_{66} \end{bmatrix} \quad (3)$$

Here,

$$\overline{Q}_{ij} = \frac{Q_{ij}}{1 - \nu_{12} \nu_{21}}$$

$[\overline{Q}]$ is called transformed reduced material property matrix.

$$[T] = \begin{bmatrix} c^2 & s^2 & 2cs \\ s^2 & c^2 & -2cs \\ -sc & sc & c^2 - s^2 \end{bmatrix}, c = \cos\theta \text{ \& } s = \sin\theta$$

$$[R] = \begin{bmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 2 \end{bmatrix}, [R] \text{ is called Reuter matrix}$$

The stress is calculated as,

$$\{\sigma\} = [D] * \{\epsilon\} \quad (4)$$

3.2 Step by Step Solution

The rectangular element is multiplex elements because its boundaries are parallel to the coordinate axes to achieve inter elemental continuity. Here 4 rectangular elements were used in 2 layers for the 2D composite body (Figures.1-4). The elements were of equal size. The number of nodes was 9. The FEM provides better approximation to the exact value, if the aspect ratio (the ratio of the largest dimension of the element to the smallest dimension) is nearly unity [5].

The next step was to derive the element equations that is, to formulate the element stiffness matrix and load vectors. These equations can be obtained using the principle of minimum potential energy. The following formulae were used,

Element stiffness matrix,

$$[K]^{(e)} = \iiint_{V^{(e)}} [B]^T [D] [B]. dv \quad (5)$$

Element load vector due to tensile stress,

$$F_3 = F_9 = (pr * W *) / 2 \quad (6)$$

Once the element stiffness matrices and element vectors are found in a common global coordinate system, then the element stiffness matrices and element load vectors were assembled to form the overall system of equations of the form

$$[K] * \{u\} = \{F\}$$

where, $[k]$ is the assembled stiffness matrix, $\{u\}$ is the vector of nodal displacements and $\{F\}$ is the vector of nodal forces for the complete structure. The next step was the incorporation of boundary conditions. Here, we had four different boundary conditions at the different nodes of the body (Figures 1-4).

Then the system of the equations was solved by matlab to obtain the nodal displacements $\{u\}$. From these known nodal displacements, element strains and stresses were computed using the relations of plane stress conditions by using C language. Stresses are calculated at point 1-10, according to figure 6.

4. RESULT AND DISCUSSIONS

Since analytical solutions are not available for mixed boundary problems, the validity of the FEM modeling of the problem was checked by considering an isotropic, homogeneous elastic body of the same size under constant stress condition. An exact result was obtained by the same code used later for the problems.

4.1 Unidirectional Composite

4.1.1 Boundary Condition 1

For unidirectional composite, boundary condition 1 is a constant stress problem. Only longitudinal stress should present in the all elements. Though because of numerical errors, transverse and shear stresses are also present. The value of longitudinal stresses for elements 1, 2, 3 and 4 vary from 66 to 71 MPa. The variations in longitudinal, transverse and shear stresses are shown in Figs 7, 8, 9. The value of transverse stress varies from -9.31 MPa to +9.31 MPa. For boundary condition 1, the shear stress varies from -16 MPa to +16 MPa. At the middle of the element it is nearly zero.

4.1.2 Boundary Condition 2

The stresses for boundary conditions 2 are quite similar to boundary condition 1. For boundary condition 2, the highest longitudinal stress is 72.047 MPa at element 4 whereas the lowest value is 66.828 MPa at element 2. The transverse stresses are also higher at element 3 and 4 than element 1 and 2. The highest value of transverse stress is -7.338 MPa at element 4 and shear stress is 19.73 MPa at element 1.

4.1.3 Boundary Condition 3

For boundary condition 3, element 3 shows the lower value of longitudinal stress than element 1, 2 and 4. For boundary condition 3 the highest value of longitudinal, transverse and shear stresses are 75.532 MPa, -6.886 MPa and -20.354 MPa respectively.

4.1.4 Boundary Condition 4

For boundary condition 4, both the element 1 and 3 show the lower value of longitudinal stress. For boundary condition 4 the values are 70.982 MPa, -4.007 MPa and -16.34 MPa.

The stress variations for boundary conditions 2, 3 & 4 are also shown graphically in figures 10-18. From these graphs it can be concluded that boundary conditions are the dominant factors for developing stresses in the elements.

4.2 Angle Lamina

The same boundary conditions were imposed for angle lamina and then the stresses are now analyzed. From the analysis it is seen that when the boundary conditions are changed the stresses are changed. For boundary condition 1, maximum stress developed when the angle is 60°. But in case of boundary condition 2, 3 and 4 it is 45, 30° and 30°.

4.2.1 Boundary Condition 1 ($\theta=60^\circ$)

Here longitudinal, transverse and shear stresses are maximum at element 1. In case of longitudinal stress it is 187.967 MPa. the values of transverse and shear stresses are 362.856 MPa and 230.933 MPa respectively.

4.2.2 Boundary Condition 2 ($\theta=45^\circ$)

For boundary condition 2 longitudinal, transverse and shear stresses are very much higher than other

boundary conditions. The maximum stresses developed in the body are 18130.11MPa (longitudinal), 18875.39 MPa (transverse) and 22454.95 MPa (shear).

4.2.3 Boundary Condition 3 ($\theta=30^\circ$)

For boundary condition 3, the highest value of longitudinal, transverse and shear stresses developed in the body are 884.32 MPa, -987.697 MPa and -708.792 MPa respectively.

4.2.4 Boundary Condition 4 ($\theta=30^\circ$)

In case of boundary condition 4, the highest longitudinal stress is -814.362 MPa, the highest transverse stress is -662.939 MPa, and the highest shear stress is -422.1 MPa.

The variations are also shown in graphs in Figs 19-27.

5. FIGURES

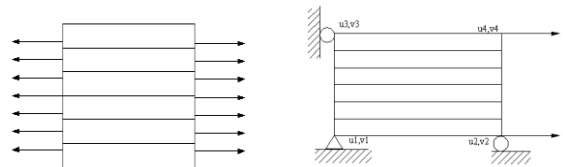
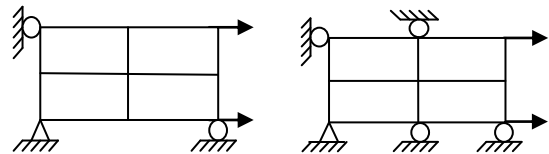
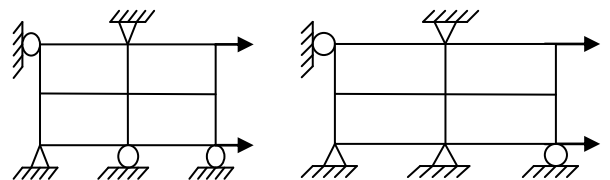


Fig 1. Transformation from applied stress into concentrated load



Figs 2,3. Boundary conditions 1 and 2



Figs 4,5. Boundary conditions 3 and 4.

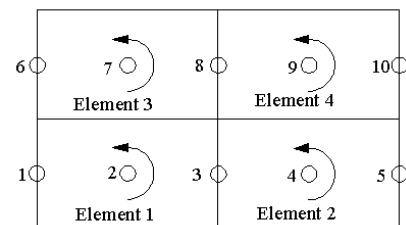


Fig 6. Node numbering for rectangular element (where stresses are calculated)

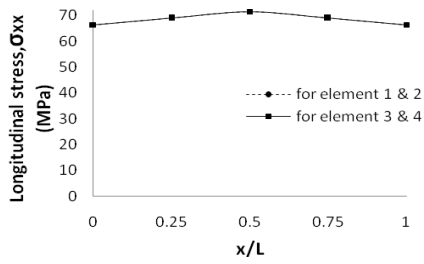


Fig 7. Longitudinal stress variation (UD composite, Boundary condition 1)

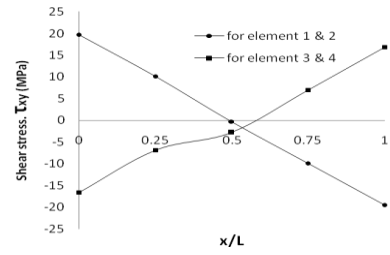


Fig 12. Shear stress variation (UD composite, Boundary condition 2)

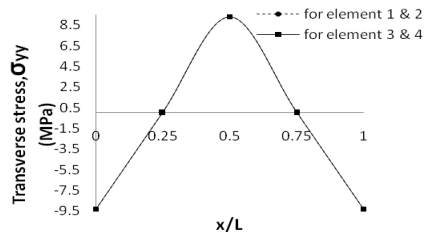


Fig 8. Transverse stress variation (UD composite, Boundary condition 1)

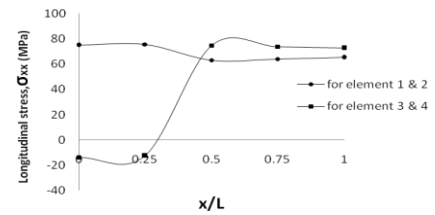


Fig 13. Longitudinal stress variation (UD composite, Boundary condition 3)

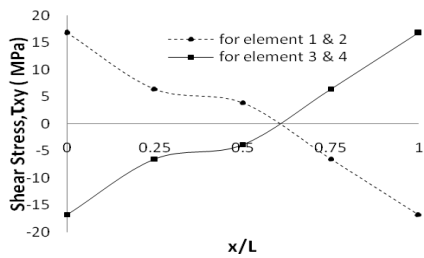


Fig 9. Shear stress variation (UD composite, Boundary condition 1)

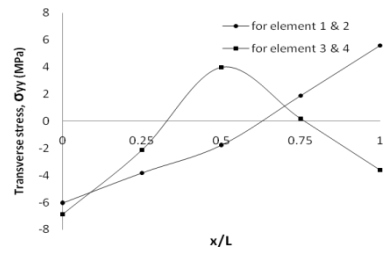


Fig 14. Transverse stress variation (UD composite, Boundary condition 3)

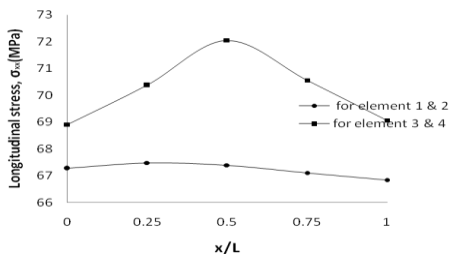


Fig 10. Longitudinal stress variation (UD composite, Boundary condition 2)

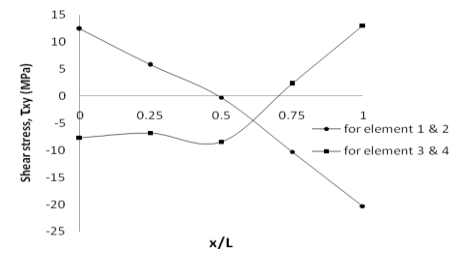


Fig 15. Shear stress variation (UD composite, Boundary condition 3)

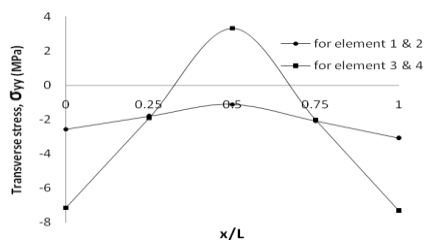


Fig 11. Transverse stress variation (UD composite, Boundary condition 2)

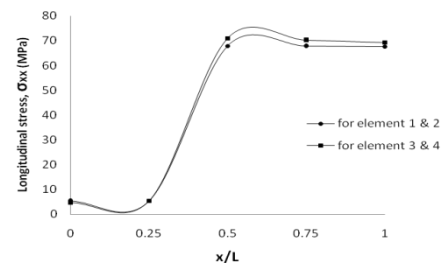


Fig 16. Longitudinal stress variation (UD composite, Boundary condition 4)

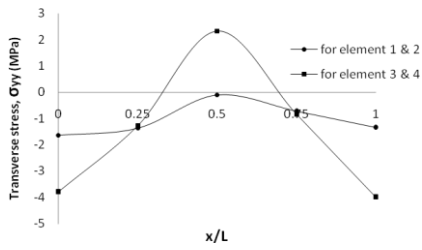


Fig 17. Transverse stress variation (UD composite, Boundary condition 4)

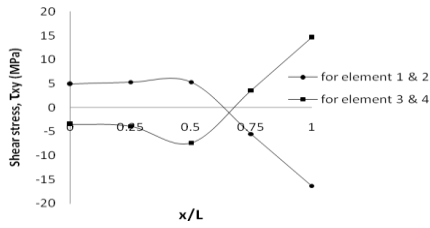


Fig 18. Shear stress variation (UD composite, Boundary condition 4)

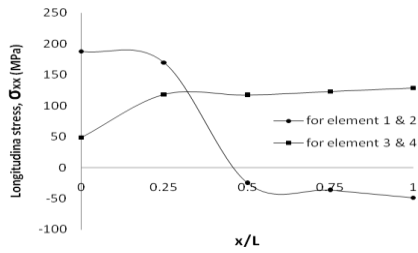


Fig 19. Longitudinal stress variation for 1st boundary condition (Angle lamina, $\theta=60^\circ$)

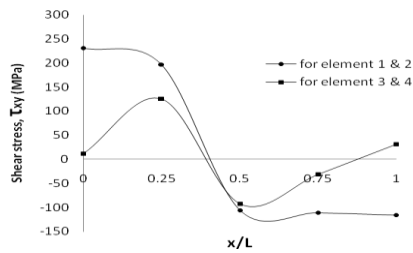


Fig 20. Shear stress variation for 1st boundary condition (Angle lamina, $\theta=60^\circ$)

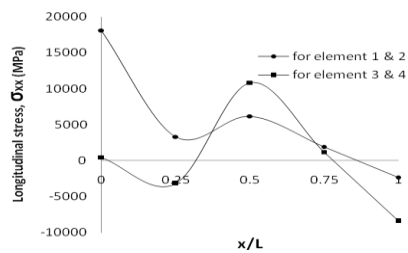


Fig 21. Longitudinal stress variation for 2nd boundary condition (angle lamina, $\theta=45^\circ$)

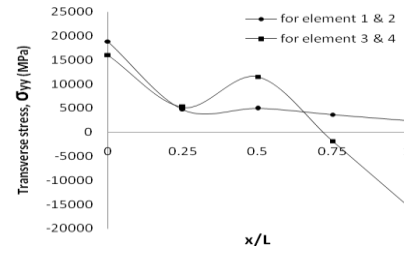


Fig 22. Transverse stress variation for 2nd boundary condition (Angle lamina, $\theta=45^\circ$)

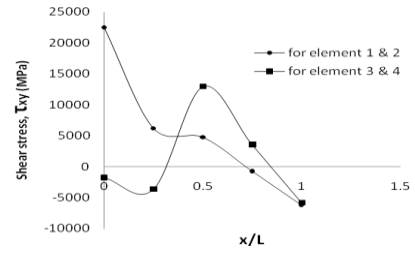


Fig 23. shear stress variation for 2nd boundary condition (Angle lamina, $\theta=45^\circ$)

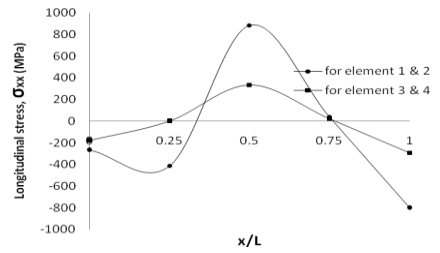


Fig 24. Longitudinal stress variation for 3rd boundary condition (Angle lamina, $\theta=30^\circ$)

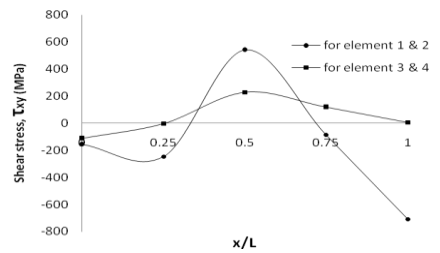


Fig 25. Shear stress variation for 3rd boundary condition (Angle lamina, $\theta=30^\circ$)

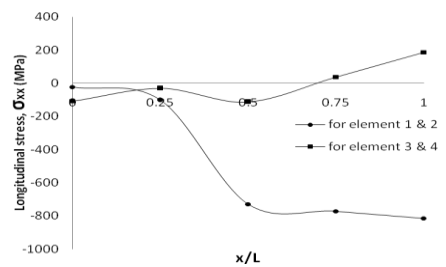


Fig 26. Longitudinal stress variation for 4th boundary condition (Angle lamina, $\theta=30^\circ$)

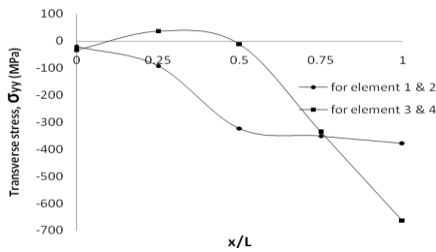


Fig 27. Transverse stress variation for 4th boundary condition (Angle lamina, $\theta=30^\circ$)

6. CONCLUSIONS

When different boundary conditions are present, then the imposed boundary conditions become the dominating factors and this can be observed by comparing the results of different boundary conditions with constant stress problem. The numerical results show that the location and type of the boundary conditions and the orientation of the lamina influence the distribution of the mechanical behavior like stresses of that body significantly.

In this study, the finite element method was used to analyze the stresses in regular shaped body. This method is even more suitable for irregular shaped bodies. So, the future researchers can focus on determining the effects of stresses considering irregular shaped body. Here, the effect of thermal stresses was not considered. It can be a more interesting problem, if thermal stresses are added in these problems.

7. REFERENCES

1. Kaw, A. K., 2006, *Mechanics of Composite Materials*, Taylor & Francis.
2. Rao, S. S., 2005, *The Finite Element Analysis*, Pergamon Press.
3. Buchanan, G. R., 1995, *Theory and Problems of Finite Element Analysis*, McGraw-Hill.
4. DebNath, S. K., 2007. "Displacement potential approach to solution of elasticity problems of orthotropic composite structures", Ph.D. Thesis,

Bangladesh University of Engineering & Technology (BUET), Bangladesh.

5. Clough, R.W., 1969, "Comparisons of Three Dimensional Finite Elements", *Proc. of the Symposium on Application of FEM in Civil Engineering*.
6. Chen, H., Bai, R. and Wang, M., 2003, "Study on failure process of delaminated stiffened composite plates under compression", *Acta Mechanica Sinica*, pp. 19:289-299.

8. NOMENCLATURE

Symbol	Meaning	Unit
K	Stiffness Matrix	N/mm
E	Young's modulus	GPa
ν	Poisson's ratio	-
τ	Shear Stress	MPa
$\sigma_{xx}, \sigma_{yy}, \tau_{xy}$	Stress components in the x-direction, y-direction and xy-plane	MPa
$\epsilon_{xx}, \epsilon_{yy}, \gamma_{xy}$	Strain components in x-direction, y-direction and xy plane	-
$\{u\}, \{v\}$	Nodal Displacement in the x and y direction	mm
θ	Angle between global and local axis	Degree
D	Material property matrix	MPa
L	Length of the elastic body	mm
W	Width of the elastic body	mm
t	Thickness of the elastic body	mm

9. MAILING ADDRESS

Zannatul Ferdous

Graduate Student
Department of Mechanical and Manufacturing Engineering,
University of Manitoba,
Winnipeg, Canada, R3T 5V6.
E-mail : umferdou@cc.umanitoba.ca