# Development of Higher Order Shear element for the static analysis of composite lamina using mat lab

A.Venkateswarlu, Dr. V.K.Sharma, N.Ranganayakulu

1. and 2 ,Bhagawanth University, Ajmeer
3. VITS, Hyderabad

## ABSTRACT

In this paper a higher order shear element, which is a 4 nodded quadrilateral element, is developed.  This element is used for the static analysis of composite lamina. Using the Higher order shear deformation plate theory for basic displacement field equations, the B matrix, (Strain-Displacement relationship matrix) the D matrix (material matrix) and the resultant K (global stiffness matrix) are formulated. These B, D & K matrices were used in formulating the 4 nodded element with 3 degree of freedom at each node. This element was also formulated to incorporate 5 degrees of freedom per node to obtain increased accuracy in the analysis. MATLAB, a numerical analysis package, was used for the calculation of nodal displacements taking A. Ferreira [7] as basis for coding principles. The modeling was initially done for an isotropic material given by Chandraputla [5] then extending it for the composite lamina with different angles of orientations. Knowing the position of the quadrilateral in the coordinate system, geometrical dimensions, its material properties and the loads which it is being subjected to, the above said entity can be calculated. Developing a MATLAB code was meticulously done to make sure that all the formulations are rightly interpreted in the code. Eliminating rows and columns after removing corresponding constraints is one such example. This code is helpful in eliminating all the paper work that is needed to solve problems in FEM based on theory used and can obtain results as accurate and precise as possible. Complete terminology, sample problem included in this report ensures that any person novice to FEM and its formulation in MATLAB will understand the code and can successfully implement it for future purposes.

Keywords: MAT LAB, Static analysis, Shear element

## 1. INTRODUCTION

### 1.1  COMPOSITES

Composite materials for construction, engineering, and other similar applications are formed by combining two or more materials in such a way that the constituents of the composite materials are still distinguishable, and not fully blended. One example of a composite material is concrete, which uses cement as a binding material in combination with gravel as a reinforcement. Composite materials take advantage of the different strengths and abilities of different materials. In the case of mud and straw bricks, for example, mud is an excellent binding material, but it cannot stand up to compression and force well. Straw, on the other hand, is well able to withstand compression without crumbling or breaking, and so it serves to reinforce the binding action of the mud.

### 1.2 COMPOSITE LAMINA

A lamina is a flat (or sometimes curved) arrangement of unidirectional (Or woven) fibers suspended in a matrix material. A lamina is generally assumed to be orthotropic, and its thickness depends on the material from which it is made. For example, a graphite/epoxy (graphite fibers suspended in an epoxy matrix) lamina may be on the order of 0.005 in (0.127 mm) thick. For the purpose of analysis, a lamina is typically modeled as having one layer of fibers through the thickness. This is only a model and not a true representation of fiber arrangement.

## 2. RESEARCH METHODOLOGY

### 2.1 DEVELOPMENT OF THE FOUR NODED QUADRILATERAL ELEMENT WITH THREE DEGREES OF FREEDOM PER NODE

For a 4 noded quadrilateral plane stress element, we have,

$(x,y,t)= N_1 + N_2 + N_3 + N_4$

$(x,y,t)= N_1 + N_2 + N_3 + N_4$        Where,    $N_i = (1+\zeta\zeta_i)(1+\eta\eta_i)$    $i=1,2,3,4$    i.e.

$N_1 = (1-\zeta)(1-\eta)$;
$N_2 = (1+\zeta)(1-\eta)$;
$N_3 = (1+\zeta)(1+\eta)$;
$N_4 = (1-\zeta)(1+\eta)$;

The coordinates are called as natural coordinates & $N_i$ are the Shape functions.

The Jacobian Matrix is given by;

$[ J ] =$

Where,    $J_{11} = = -(1-\eta) x_1 + (1-\eta) x_2 + (1+\eta) x_3 - (1+\eta) x_4$

$J_{12} == -(1-\eta) y_1 + (1-\eta) y_2 + (1+\eta) y_3 - (1+\eta) y_4$

$J_{21} = = -(1-\zeta) x_1 - (1+\zeta) x_2 + (1+\zeta) x_3 + (1-\zeta) x_4$

$$J_{22} = = -(1-\zeta) y_1 - (1+\zeta) y_2 + (1+\zeta) y_3 + (1-\zeta) y_4$$

Where
$$x = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4$$
$$y = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4$$

## 2.2 DEVELOPMENT OF THE FOUR NODED QUADRILATERAL ELEMENT WITH FIVE DEGREES OF FREEDOM PER NODE

For a 4 noded quadrilateral plane stress element, we have,

$(x,y,t) = N_1 + N_2 + N_3 + N_4$
$(x,y,t) = N_1 + N_2 + N_3 + N_4$
$(x,y,t) = N_1 + N_2 + N_3 + N_4$
$(x,y,t) = N_1 + N_2 + N_3 + N_4$

Where,

$N_i = (1+\zeta\zeta_i)(1+\eta\eta_i)$     $i = 1,2,3,4$

i.e.   $N_1 = (1-\zeta)(1-\eta)$;
    $N_2 = (1+\zeta)(1-\eta)$;
    $N_3 = (1+\zeta)(1+\eta)$;
    $N_4 = (1-\zeta)(1+\eta)$;

## 3. RESULTS AND DISCUSSIONS

Comparison of nodal displacements obtained from the MAT LAB programs for 1, 2 and 4 elements with the exact values:

For isotropic materials (3 DOF)

|            |   | 1 element | 2 elements | 4 elements | Exact Values |
|------------|---|-----------|------------|------------|--------------|
| At (3,0)   | u | 0.0005    | 0.0011     | 0.0014     | 0.0016       |
|            | w | 0         | 0          | 0          | 0            |
| At (3,2)   | u | 0.0019    | 0.0019     | 0.007      | 0.0075       |
|            | v | -0.0036   | -0.0142    | -0.0126    | -0.0111      |
|            | w | 0         | 0          | 0          | 0            |

For isotropic materials (5 DOF)

|            |        | 1 element | 2 elements | 4 elements | Exact Values |
|------------|--------|-----------|------------|------------|--------------|
| At (3,0)   | u      | -0.0011   | -0.0007    | -0.0005    | -0.0006      |
|            | W      | 0.2545    | -0.0233    | -0.4723    | -0.4627      |
|            | (in rad) | -0.0990 | -0.1011    | -0.5082    | -0.602       |
| At (3,2)   | u      | 0.0038    | -0.0007    | 0.0101     | 0.0099       |
|            | V      | -0.0073   | -0.0029    | -0.0213    | -0.0270      |
|            | W      | 0.0000    | 0.0000     | -0.5012    | -0.652       |

| (in rad) | -0.1697 | -0.0021 | -0.2218 | -0.257 |
|----------|---------|---------|---------|--------|
| (in rad) | 0.3535  | 0.0799  | 0.6554  | 0.676  |

For composite materials (3 DOF)

|          |   | 1 element | 2 elements | 4 elements | Exact Values |
|----------|---|-----------|------------|------------|--------------|
| At (3,0) | u | -0.0003   | 0.0045     | 0.0009     | 0.0012       |
|          | w | 0         | 0          | 0          | 0            |
| At (3,2) | u | -0.0017   | -0.0045    | -0.0046    | -0.0049      |
|          | v | -0.0030   | -0.0122    | -0.0074    | -0.0097      |
|          | w | 0         | 0          | 0          | 0            |

For composite materials (5 DOF)

|          |        | 1 element | 2 elements | 4 elements | Exact Values |
|----------|--------|-----------|------------|------------|--------------|
| At (3,0) | u      | -0.0007   | -0.0069    | -0.0010    | -0.0012      |
|          | W      | 0.2540    | -0.1528    | 0.0936     | 0,099        |
|          | (in rad) | -0.0665 | -0.7998    | -0.4236    | -0.555       |
| At (3,2) | u      | 0.0035    | 0.0069     | 0.0099     | 0.0116       |
|          | V      | -0.0063   | -0.0251    | -0.0180    | -0.0176      |
|          | W      | 0.0000    | -0.1012    | 0.0892     | 0.077        |
|          | (in rad) | -0.1693 | 0.0000     | -0.2642    | -0.3272      |
|          | (in rad) | 0.3205  | 0.7482     | 0.5373     | 0.6143       |

From the above comparisons, it can be observed that, as the number of elements in the program is increased, the nodal displacement values obtained are closer to the exact values and convergence is achieved in the nodal displacement values.

## 4. CONCLUSION

- The finite element formulation presented in this study, based on higher-order shear deformation theory, gives reasonably good results for both 3 and 5 degrees of freedom with different materials and fiber orientations.
- A single lamina has been taken to analyze displacements then a lamina with fibers of different orientations was considered.

- A brief introduction of each topic has been included so that reader will have a clear picture of what is being done and what will be done in the report step by step.

- These MAT LAB programs can be used for applications where ever user needs to calculate resultant displacements of an element when point load is applied on it.

- Advantage of this MAT LAB program over other programming languages is that the user does not only get the direct result but, will also have access to all other entities that are required to be calculated while solving the problem to find out displacements.

- Working on this project has placed us at a vantage of getting to know the intricacies that go into dealing with FEM, working with Mat lab and also have the real time hands-on experience.

- Lucid explanation of program, terminology behind programming, logic used for application in project projects our sincere effort and appreciable understanding of the project.

- We have made all efforts to make this project and the report as inerrant as possible. Any aberrance that might have crept in is regretted.

- This program has been a collective and honest effort in order to ensure that the developed program meets all needs required.

- Using the generated MAT LAB codes, only nodal displacement have been found, which can be further modified to be able to find out the stresses in the problem as well.

- In the sample problem point load was applied on the element even uniformly distributed load (UDL) and uniformly varying load (UVL) can also be applied in future applications.

- Only a single lamina has been considered we can extend study to analyze behavior of a laminate under different loads.

- Limited to 2D elements can further be developed for 3D elements.

- This program was written for static analysis which can be further extended to dynamic analysis.

- During MAT LAB program execution a warning explaining that the matrix may be badly scaled is obtained. This is due to the complexity in coding and the size of the matrix given as an input. In future this can be avoided by using suitable algorithms.

## REFERENCES

[1]. *Mechanics of laminated composite plates and shells by J.N.Reddy*
[2]. *Finite element analysis by S.S Bhavikatti.*
[3]. *Mechanics of composite materials by Robert M. Jones.*
[4]. *Finite element analysis of composite laminates by O.O.Ochoa and J.N.Reddy.*
[5]. *Introduction to finite elements in engineering by Tirupathi R.Chandraputla and Ashok D.Belegundu.*
[6]. *Programming the Finite Element Method with Matlab by Jack Chessa.*
[7]. *Matlab codes for Finite Element Analysis by A. J. M Ferreira.*
[8].*The Finite Element Method R.L Taylor and O.C Zienkiewicz.*

## 5. SCOPE FOR FUTURE WORK