

# Finite Element Analysis of Laminated Composite Plates Using ANSYS

Mahaboobali Nadaf and Dr.R.J. Fernandes

**Abstract---** Laminated composite plates are used in various types of structure, viz., aircraft fuselages, vehicle chassis, ship hulls, etc., due to their low specific weight (mass), high specific strength and stiffness. But, they are highly susceptible to the external loadings which can affect its gross performance. Therefore it is necessary to study the structural behaviour of these plates such as deflections, stresses and strains. This study will help in preparing strong, efficient laminated plates. The main objective of the present work is to analyze the laminated composite plate using FEM software (ANSYS) and to conduct the parametric study on response of laminated plates for various side to thickness ratios, aspect ratios, different ply orientation and varying number of laminas for Static loading condition..

**Keywords---** Finite Element Analysis, Laminated Composite Plates, Uniformly Distributed Load, Orthotropic

## I. INTRODUCTION

COMPOSITE material is formed when two or more materials are combined on a macroscopic scale. If the materials are combined on a macroscopic scale, the naked eye can identify the components of composite materials. If we design it properly, the properties of composite material are far better compared with the properties of its components or constituents. The properties which can be improved in composite materials are strength, stiffness, corrosion resistance, weight, fatigue life, attractiveness, temperature dependent behaviour, thermal insulation etc.

A lamina or a Ply is formed by a combination of a large number of fibres in a thin layer of matrix. Fibres in the lamina may be continuous or discontinuous, arranged in a specific direction or in random orientation. A laminate is formed by stocking or bonding two or more laminas, it is the most common form of fibre reinforced composite, it is made of a desired thickness, so as to enable it to support a given load and maintain a given deflection. Fibre orientation of each lamina and stocking sequence of various layers can be varied to obtain a wide range of physical and mechanical properties of composite.

Laminated composite plates are used in various types of structure, viz., aircraft fuselages, vehicle chassis, ship hulls, etc., due to their low specific weight (mass), high specific

strength and stiffness. But, they are highly susceptible to the external loadings which can affect its gross performance. Therefore it is necessary to study the structural behaviour of these plates such as deflections, stresses and strains. This study will help in preparing strong, efficient laminated plates. The main objective of the present work is to analyze simply supported laminated composite plate for uniformly distributed load using FEM and to conduct a parametric study on response of laminated plates for various sides to thickness ratios, aspect ratio, different ply orientation and varying number of laminas.

## Methodology

- The study on displacement parameter is done using ANSYS software for simply supported orthotropic laminated composite plate for various sides to thickness ratios, aspect ratio, different ply orientation and varying number of laminas for Static loading condition.
- For validation purpose, these results obtained from ANSYS software are compared with FSDT results from J.N.REDDY[4].
- Once the validation is done, the parametric study is made by changing the properties and dimensions of materials.
- Based on above observations, the results, discussions and conclusions will be presented.

## II. LITERATURE REVIEW

The finite element method is widely used in the analysis of laminated composite plates to understand the structural behaviour.

Choudary and Tungikar [1] have used the finite element method to analyse the geometrically nonlinear behaviour of laminated composite plates. In their work, the effect of different fibreorientation, effect of degree of orthotropic properties both symmetric and anti-symmetric, effect of number of layers on central deflections.

Ganapathi et al. [2] studied the buckling analysis, membrane-plate coupling effects, static and dynamic problems of moderately thick laminates, moderately large deflections. For this he has used an eight node Co membrane plate quadrilateral finite element. This is based on Reissner-Mindlin plate theory.

Mahaboobali Nadaf, PG Student, Civil, SDM College of Engineering and Technology, Dharwad, India. E-mail:mehboob\_ali679@yahoo.com

Dr.R.J. Fernandes, Asst. Professor, Civil, SDM College of Engineering and Technology, Dharwad, India. E-mail:fernandes.regi@gmail.com

DOI:10.9756/BJJMML8171

Vanam B.C.L., Rajyalakshmi M. and Inala R. [3]. They have analysed an isotropic rectangular plate with different boundary conditions and load applications for static conditions. For this Finite element approach, they have used a four noded quadrilateral element. Using these FE formulations, a MATLAB programme is developed to analyse the plate. After analysis the results from MATLAB are compared with classical method-exact solutions for validation.

J.N.Reddy [4] has explained the various concepts of mechanics of laminated composite plates in his book, such as equations of anisotropic elasticity, virtual work principles, variational methods, basic concepts of composite materials, classical, first order, higher order theories for laminated plates, one dimensional analysis of laminated composite plate, analysis of specially orthotropic plates,, analytical solutions for rectangular laminated plates using CLPT, FSDT, linear and non linear analysis of plates. This book is referred for validation of our ANSYS results.

III. FINITE ELEMENT FORMULATION

In FEM, the physical domain is divided into number of sub-domains, which are termed as finite elements. This process of dividing is called discretisation which represents the complex domain in a proper representation. The number of these finite elements decides the accuracy of the solution. Increase in finite element numbers increases the accuracy but requires more computational effort and time. So it has to be appropriately selected by considering the past experience of the analyst and after making the convergence study by varying mesh size.

Nodes: These are the discrete points which connect the two adjacent finite elements.

Finite element Mesh: It is the particular arrangement of finite elements.

Derivation of suitable approximation functions for each element is obtained.

On the basis of continuity of solution and balance of internal forces, element stiffness and mass matrices are assembled.

A. Steps in FE Formulation Using iso-Parametric Element

- From the element library, finite element is chosen for given problem.
- Using interpolation functions and their derivatives, stress-strain and strain-displacement relationships are formulated.
- In the elemental level, stiffness and mass matrices are formulated.
- Global equations are derived using elemental level equations.
- From these resulting equations, solutions are obtained.

B. Geometry of Element is given by Expressions

$$x = \sum_{i=1}^n N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3 + \dots + N_n x_n = [N_1 \ N_2 \ N_3 \ \dots \ N_n] \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \\ \vdots \\ x_n \end{Bmatrix} = [N]\{x\}$$

$$y = \sum_{i=1}^n N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_3 + \dots + N_n y_n = [N_1 \ N_2 \ N_3 \ \dots \ N_n] \begin{Bmatrix} y_1 \\ y_2 \\ y_3 \\ \vdots \\ y_n \end{Bmatrix} = [N]\{y\}$$

Where, x<sub>i</sub> and y<sub>i</sub> are nodal coordinates of element.

N represents the number of nodes per element (4, 8 or 9)

N<sub>i</sub> represents shape functions are expressed as below interms of natural coordinate system.

For four noded element:

$$N_1 = \frac{1}{4}(1-\xi)(1-\eta) \quad N_2 = \frac{1}{4}(1+\xi)(1-\eta)$$

$$N_3 = \frac{1}{4}(1+\xi)(1+\eta) \quad N_4 = \frac{1}{4}(1-\xi)(1+\eta)$$

IV. ANALYSIS, RESULTS AND DISCUSSIONS

For the validation and analysis work, the material used has following properties.

Table1: Material Properties

Properties	Values	Units
G <sub>12</sub> =	0.5	Gpa
G <sub>13</sub> =	0.5	Gpa
G <sub>23</sub> =	0.2	Gpa
E <sub>1</sub> =	25	Gpa
E <sub>2</sub> =	1	Gpa
E <sub>3</sub> =	1	Gpa
μ <sub>12</sub> =	0.25	-
μ <sub>23</sub> =	0.25	-
μ <sub>13</sub> =	0.25	-
q <sub>0</sub> =	1	N/sqm
Supports	Simply supported	

All the output values for displacement are converted into non-dimensional form by multiplying with term m1.

w	m1	w = w x m1
Max. displacement	(100 x E <sub>2</sub> x h <sup>3</sup> ) / (q x a <sup>4</sup> )	non dimensional

Where h=thickness of plate, q=load intensity

a=side of plate, E2=young's modulus in direction-2.

A. Validation

For validation, the non dimensional displacement values from ANSYS software are compared with FSDT results from J.N.REDDY[4] and percentage variation is calculated. From the comparison, we can see that errors are negligible and values are almost same. Hence we can go for parametric study by changing a/b ratios, a/h ratios for symmetric orthotropic plate.

Table 2: Validation of Non Dimensional Displacement from J.N. Reddy[4]

Validation for non dimensional displacement, w				
No.of layers	a/h	J.N. Reddy	ANSYS	% error
		W	w	
1	100	0.6528	0.6552	0.37%
1	20	0.7262	0.7327	0.90%
1	10	0.9519	0.9686	1.75%
3	100	0.6697	0.676	0.94%
3	20	0.7572	0.8033	6.09%
3	10	1.0219	1.1881	16.2%
4	100	0.6833	0.6868	0.51%
4	20	0.7694	0.8106	5.35%
4	10	1.025	1.1665	13.8%
5	100	0.6874	0.69	0.38%
5	20	0.7581	0.7877	3.90%
5	10	0.9727	1.0815	11.1%

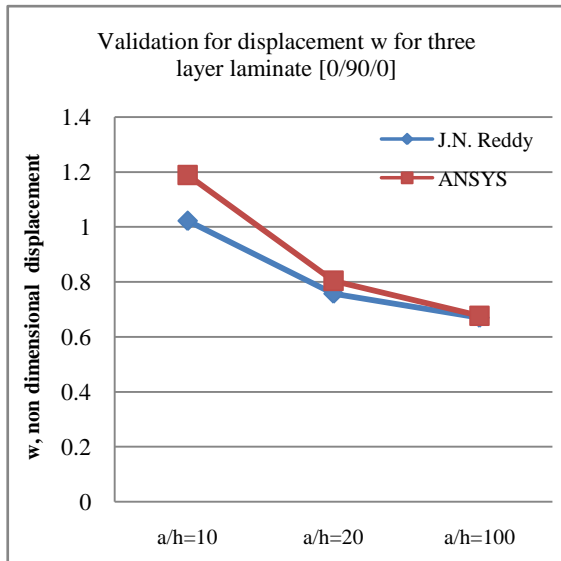


Figure 1: Plot for Validation of Non Dimensional Displacement from J.N. Reddy[4]

B. Parametric Study for Symmetric Orthotropic Plate

For parametric study, the material properties mentioned in Table 1 is adopted.

Laminate plates having one layer [0], three layers [0/90/0], four layers [0/90/90/0], five layers [0/90/0/90/0] are analysed.

Analysis is done for aspect ratios (a/b ratios) for 1.0, 1.5, 2.0, 2.5 and 3.0. And for side to thickness ratio (a/h ratio) for 4, 5, 10, 20, 40, 50 and 100.

Table 3: ANSYS Values for w, Non Dimensional Displacement [0/90/0]

ANSYS values for w, non dimensional displacement for different a/b and a/h ratios, for three layered laminate [0/90/0]					
a/h	a/b =1	a/b =1.5	a/b =2.0	a/b =2.5	a/b =3.0
100	0.67597	0.50188	0.30717	0.17334	0.09771
50	0.6894	0.51292	0.31339	0.17674	0.09975
40	0.70353	0.52106	0.3178	0.17918	0.10121
20	0.80333	0.58571	0.35208	0.19773	0.11255
10	1.18806	0.81195	0.4681	0.26376	0.15569
5	2.53082	1.52975	0.85991	0.50992	0.32654
4	3.42625	2.0032	1.13696	0.69169	0.4546

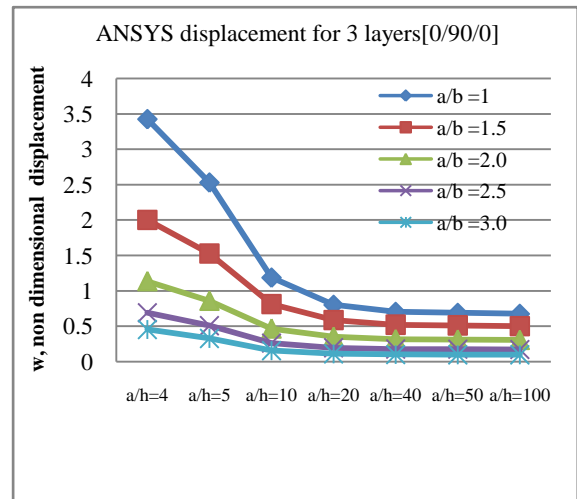


Figure 2: Plot for ANSYS Values for w, Non Dimensional Displacement [0/90/0]

Table 4: ANSYS Values for w, Non Dimensional Displacement [0/90/90/0]

ANSYS values for w, non dimensional displacement for different a/b and a/h ratios, for four layered laminate [0/90/90/0]					
a/h	a/b =1	a/b =1.5	a/b =2.0	a/b =2.5	a/b =3.0
100	0.687	0.413	0.205	0.101	0.052
50	0.703	0.421	0.209	0.103	0.054
40	0.714	0.427	0.212	0.105	0.055
20	0.811	0.475	0.236	0.119	0.065
10	1.166	0.647	0.327	0.177	0.105
5	2.375	1.251	0.680	0.409	0.268
4	3.188	1.684	0.943	0.583	0.391

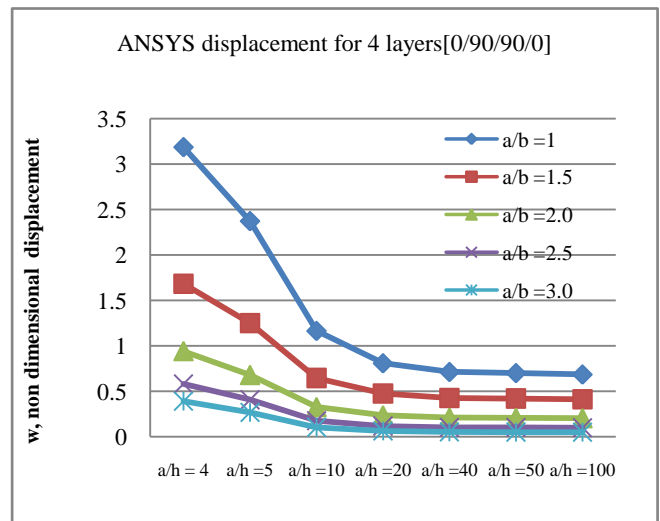


Figure 3: Plot for ANSYS Values for w, Non Dimensional Displacement [0/90/90/0]

Table 5: ANSYS Values for  $w$ , Non Dimensional Displacement [0/90/0/90/0]

ANSYS values for $w$ , non dimensional displacement for different $a/b$ and $a/h$ ratios, for five layered laminate [0/90/0/90/0]					
$a/h$	$a/b=1$	$a/b=1.5$	$a/b=2.0$	$a/b=2.5$	$a/b=3.0$
100	0.690	0.354	0.156	0.072	0.036
50	0.702	0.361	0.162	0.076	0.039
40	0.712	0.367	0.166	0.079	0.041
20	0.788	0.415	0.199	0.102	0.058
10	1.081	0.603	0.328	0.194	0.124
5	2.210	1.346	0.834	0.549	0.383
4	3.042	1.903	1.212	0.814	0.575

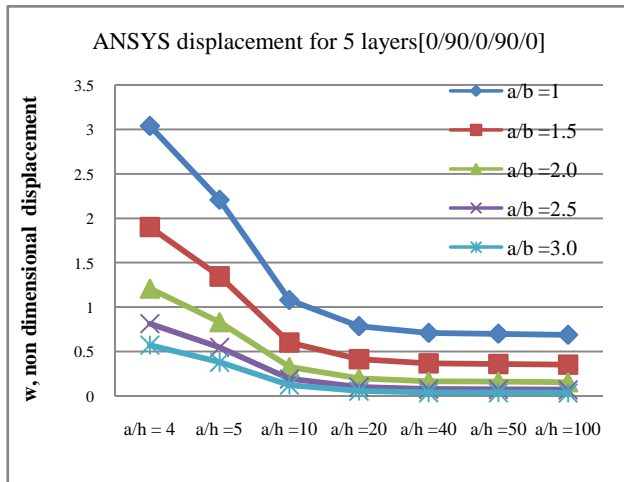


Figure 4: Plot for ANSYS Values for  $w$ , Non Dimensional Displacement [0/90/0/90/0]

## V. CONCLUSION

In the present work, analysis of laminated composite plate is done using finite element software ANSYS. Results are first validated with FSDT results from J.N.Reddy[4]. Parametric study is done for non dimensional displacement

The above results are for square plate i.e.  $a/b$  ratio =1 for uniformly distributed load, all sides are simply supported.

- When non dimensional displacement  $w$ , results from ANSYS are compared with J.N.Reddy results, error for thin plate ( $a/h=100$ ), is within 2.00% and error for thick plate ( $a/h=10$ ), is up to 16.2%.

## REFERENCES

- [1]. S.S. Choudhary and V.B. Tungikar, "A simple finite element for nonlinear analysis of composite plates", International Journal of Engineering Science and Technology, Vol. 3, No. 6, Pp. 4897-4907, 2011.
- [2]. M. Ganapathi, O. Polit, M. Touratier, "C<sub>0</sub> eight-node membrane shear-bending element for geometrically non-linear (static and dynamic) analysis of laminates", Int. Journal Number Meth Eng, Vol. 39, No. 20, Pp. 3453-3474, 1996.
- [3]. B.C.L. Vanam, M. Rajyalakshmi and R. Inala, "Static analysis of an isotropic rectangular plate using finite element analysis (fea)", Journal of Mechanical Engineering Research, Vol. 4, No. 4, Pp. 148-162, 2012.
- [4]. J.N. Reddy, "mechanics of laminated composite plates and shells, theory and analysis", second edition, CRC Press (India) private limited, 2004.
- [5]. M. Mukhopadhyay, "Mechanics of composite materials and structures", first edition, Universities Press (India) private limited, 2005.