International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 03 Issue: 12 | Dec -2016 www.irjet.net p-ISSN: 2395-0072

ANALYTICAL AND NUMERICAL STUDY OF COMPOSITE PLATES

Pallavi kokitakar[1], Prof. C. S. Wadageri[2], Mr. Yogesh Patil[3], Prof. Ratan Patil[4]

[1]*Student, Dept. of Mechanical Engineering, MMEC Belagavi, Karnataka, India*
[2]*Professor, Dept. of Mechanical Engineering, MMEC Belagavi, Karnataka, India*
[3]*Managing Director of Future technology's, Belagavi, Karnataka, India*
[4]*Professor, Dept. Of Mechanical Engineering, JGI Belagavi, Karnataka, India*

Abstract - *In past few decades considerable work has been carried out on composites as they are suitable for most of the engineering applications, where specific strength is one of the most important criteria (like automotive and aerospace applications). In this paper the symmetrical and unsymmetrical glass/epoxy, boron/epoxy and graphite/epoxy laminated composite plates are analyzed for different in-plane loading conditions. Thermal loading conditions are also included along with in-plane mechanical loads and the simulations are carried out using ANSYS 14.5 FEA tool. The results are compared with the classical laminate theory (CLT) and it was observed that the results are in unison with each other. Further the mathematical model is generated for CLT by using MATLAB simulation tool for investigating different configurations of composite plate.*

Key Words: Finite element analysis (FEA), Classical laminate theory (CLT), Laminate, ply, Composite plate.

1. INTRODUCTION

Composite materials possess enhanced properties compared to other material. Since composite materials have advantages such as light weight, low density, elevated ratio of stiffness and strength, it gives long life to the components. Composite materials wide range of applications in aerospace industry, automotive industry, transport, building and construction, marine & also in medical equipment etc. A laminated composite material consists of some layers of composite combination consisting of matrix and fibers. All layers may have similar or dissimilar material properties with different fiber orientations under varying stacking progression. There are many open issues connecting to design of these laminated composites. Each lamina is represented by the angle of ply and separated from other plies by slash sign. In this work the composite plate are analyzed using both FEA and analytical method considering in-plane loading condition. The study is also extended to understand the combined effect of mechanical and thermal loading on composite plate.

1.1 Classical lamination theory

Relationship between stress, strain and displacement are established for a composite plate under in-plane loads such as shear and axial forces. The classical laminate theory is used to develop these relationships. Some assumptions are made to develop the relationship in the classical lamination theory.

- Every lamina is orthotropic.
- Every lamina is homogeneous.
- During deformation the line which is straight and perpendicular to the middle surface remains straight and perpendicular to the middle surface($M_{xz} = M_{yz} = 0$)
- Every lamina is elastic.
- Throughout the laminate displacements are continuous and small. ($|p|, |q|, |r| << |h|$), where h is the laminate thickness.
- Between the lamina interfaces there is no occurrence of slip.
- The thin laminate is loaded only in its plane ($S_z = N_{xy} = N_{yz} = 0$)

2. Finite element analysis

Finite element analysis has now become integral part of Computer Aided Engineering (CAE) and is been extensively used in the analysis and design of many complex real life systems. It is a numerical method for solving problems of engineering & mathematical physics for complex geometries, materials, loading & boundary conditions.

The model and simulating tool ANSYS 14.5 is used for FEA analysis. The modeling environment is shown in fig. 2.1

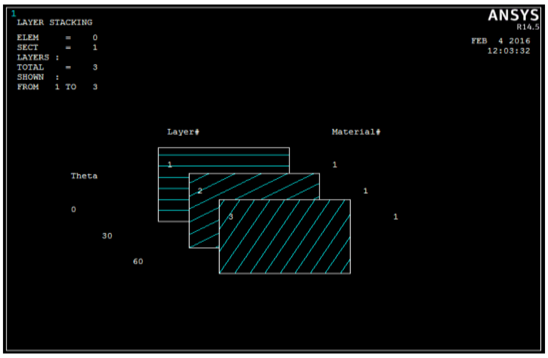


Fig-2.1 Orientation & stacking sequence for [0/30/60]

In this work 2D rectangular composite plate is used for analysis. The mesh size is 0.5 mm and the thickness of each lamina is 5mm.

The materials used for rectangular elements are glass/epoxy, graphite/epoxy and boron/epoxy. Typical mechanical properties for unidirectional lamina are given:

Table -2.1: Mechanical properties of unidirectional lamina

Properties	Symbol	units	Glass/ epoxy	Graphite/ epoxy	Boron/ epoxy
Longitudinal elastic modulus	Y_1	GPa	38.6	204	181
Transverse elas2tic modulus	Y_2	GPa	8.27	18.50	10.30
Major Poisson's ratio	V_{12}		0.26	0.23	0.28
Shear modulus	G_{12}	GPa	4.14	5.59	7.17
Longitudinal Coefficient of thermal expansion	α_1	µm/m/⁰C	8.6	6.1	0.02
Transverse Coefficient of thermal expansion	α_2	µm/m/⁰C	22.1	30.3	25.5

2.1 Material under loading without temperature

The composite plate is analyzed by using analytical method that is classical laminate theory. The same material for the same fiber orientation is analyzed using MATLAB stimulation tool and it is observed that both the results are in good agreement with each other.

The three ply laminate with orientation of 0/30/-45. The graphite/epoxy material is considered for analysis.

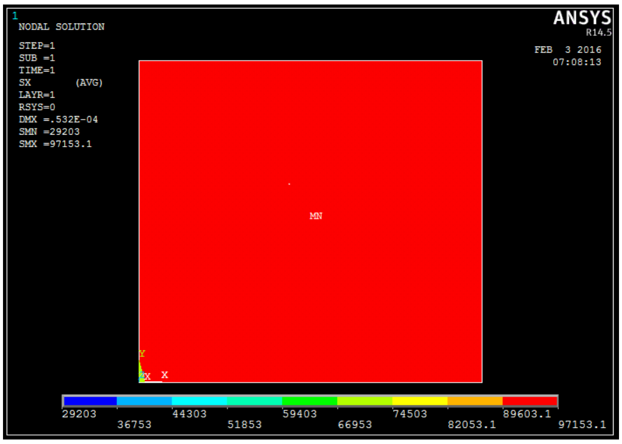
Table -2.1.1: Comparistion between MATLAB and Analytical solution for Graphite/epoxy

	Stress in x direction		Stress in Y direction		Stress in XY direction	
	MATLAB	Analytical solution	MATLAB	Analytical solution	MATLAB	Analytical solution
0^0 top	33513	33510	61875	61880	-27504	-27500
0^0 bottom	55767	55770	45312	45310	-12800	-12800
30^0 top	69297	69300	73914	73910	33808	33810
30^0 bottom	143360	143400	81022	81020	84256	84260
-45^0 top	123530	123500	156280	156300	-118670	-118700
-45^0 bottom	-25469	-25470	-18402	-18400	40913	40910

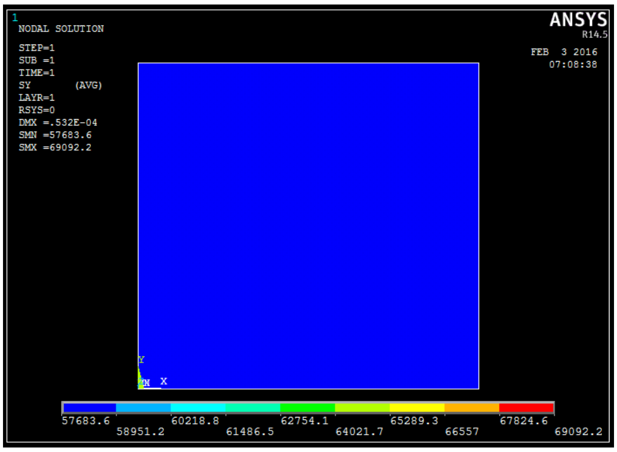
Further the MATLAB results are compared with ANSYS 14.5 analysis tool for different ply orientation of different material.

Glass/Epoxy material is used for the analysis and the ply angle are (0/30/60). The above fig. 2.1 shows the orientation & stacking sequence for [0/30/60] for FEA model.

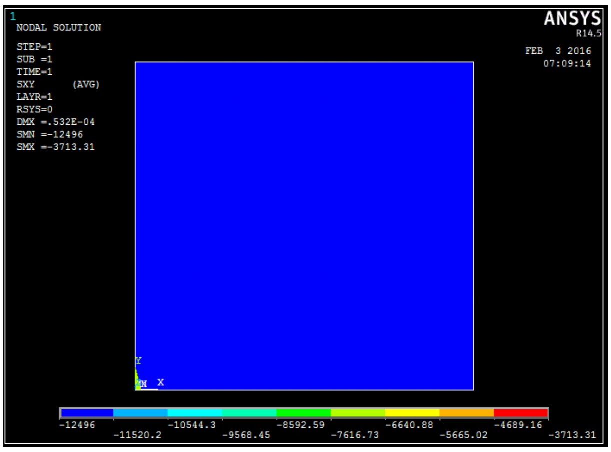
The same materials are simulated using ANSYS 14.5 analysis tool. The contour for stresses is plotted in fig and it observed that both FEA and MATLAB results are in good union with each other.



(a)



(b)



(c)

Fig-2.1.1: Stress distribution for 0^0 lamina in (a) X (b) Y and (c) XY direction

Fig-2.1.2 shows the lamina stresses X,Y and XY direction respectivity. These resuls are compared with MATLAB simulation tool as per given in Table no.-2.1.2

Table -2.1.2: Comparistion between MATLAB and ANSYS result for Glass/epoxy.

	Stress in x direction		Stress in Y direction		Stress in XY direction	
	MATLAB	ANSYS	MATLAB	ANSYS	MATLAB	ANSYS
0^0 top	29203	29203	69092	69092.2	-3713.3	-3713.31
0^0 bottom	97153	97153.1	57684	57683.6	-12496	-12496
30^0 top	72899	72899.3	73969	73968.8	16639	16639.2
30^0 bottom	88111	88111.4	58757	58756.7	7856.5	7856.52
60^0 top	56729	56729.1	98108	98107.6	10843	10842.8
60^0 bottom	55904	55904.1	42391	42391.1	-19129	-19129.2

The above table shows the result of comparisons of MAT LAB simulation tool and ANSYS analyses tool.

The MATLAB results are given in fig. 2.1.1

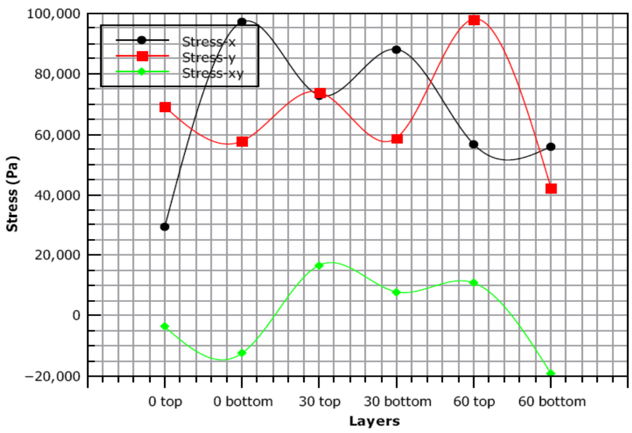


Chart -2.1.1: MATLAB results for glass/epoxy composite plate

Boron/Epoxy material is used for the anylses and the ply angle are (0/30/60)

Table -2.1.3: Comparistion between MATLAB and ANSYS result for Boron/epoxy.

	Stress in x direction		Stress in Y direction		Stress in XY direction	
	MATLAB	ANSYS	MATLAB	ANSYS	MATLAB	ANSYS
0^0 top	-11914	-11914.3	66823	66823	-5977.5	-5977.53
0^0 bottom	145730	145734	55580	55580.1	-11048	-11047.5
30^0 top	73443	73442.3	70334	70334.2	12832	12831.9
30^0 bottom	82224	82224.5	61553	61552.8	7762	7761.95
60^0 top	68551	68551.2	132760	132763	33514	33514.2
60^0 bottom	41962	41962	12947	12946.7	-37083	-37083

The above table shows the result of comparisons of MAT LAB situation tool and ANSYS analyses tool.

Graphite/Epoxy material is used for the anylses and the ply angle are (0/30/60)

Table -2.1.4: comparistion between MATLAB and ANSYS result for Graphite/epoxy

	Stress in x direction		Stress in Y direction		Stress in XY direction	
	MATLAB	ANSYS	MATLAB	ANSYS	MATLAB	ANSYS
0^0 top	-64138	-64137.9	61791	61791	-11180	-11180
0^0 bottom	188490	188491	51273	51272.6	-22870	-22870.3
30^0 top	84501	84500.6	78083	78082.8	28939	28938.6
30^0 bottom	104750	104749	57835	57834.6	17248	17248.3
60^0 top	65771	65770.9	173990	173993	47982	47982.1
60^0 bottom	20627	20626.7	-22974	-22973.5	-60119	-60118.8

The above table shows the result of comparisons of MAT LAB situation tool and ANSYS analyses tool. Now as it is established that results obtain from both MATLAB coding and ANSYS simulation have sink with the analytical results.

Glass/Epoxy material is used for the analysis and the ply angle are (30/60/30/60). The fig. 2.1.2 shows the orientation & stacking sequence for [30/60/30/60] for FEA model.

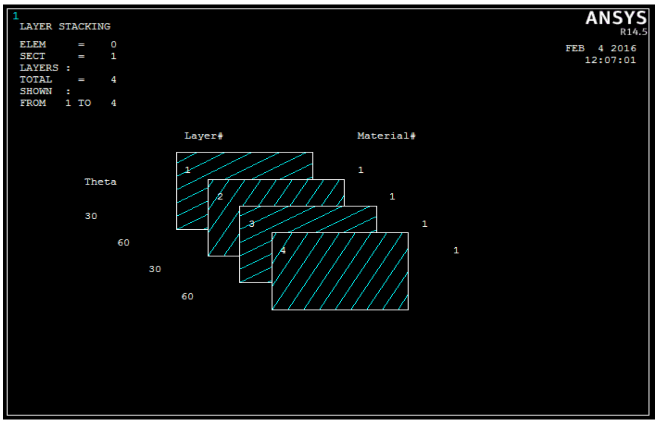
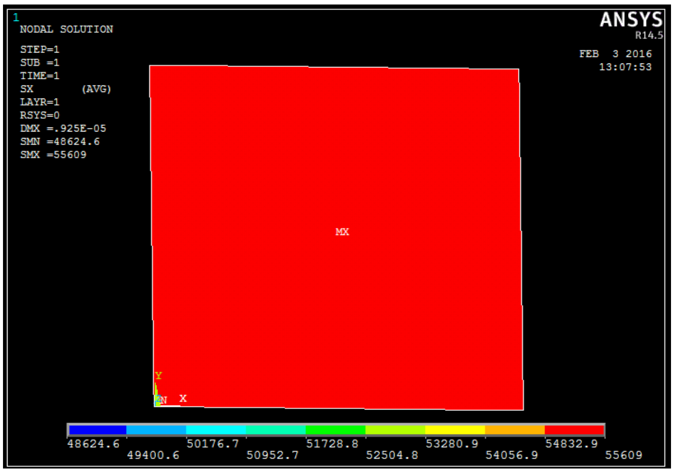


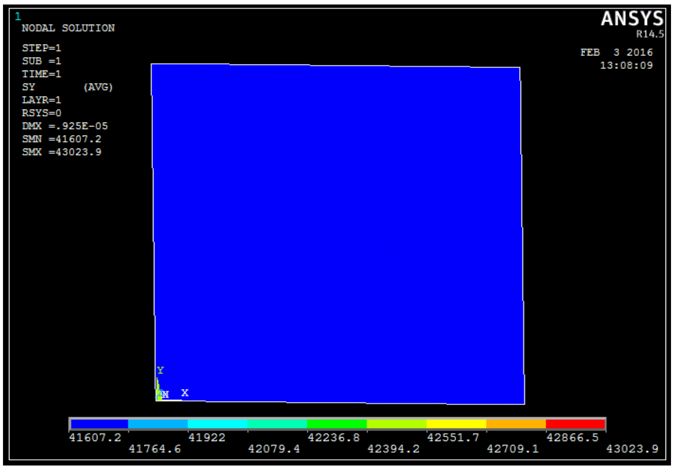
Fig-2.1.2: Orientation & stacking sequence for [30/60/30/60]

Fig-2.1.3 shows the lamina stresses X and Y direction respectivity. These resuls are compared with MATLAB simulation tool as per given in Table no.-2.1.5

The table-2.1.5 shows the result of comparisons of MAT LAB situation tool and ANSYS analyses tool.



(a)



(b)

Fig-2.1.3: Stress distribution for 0^0 lamina in (a) X (b) Y direction

Table -2.1.5: comparistion between MATLAB and ANSYS result for Glass/epoxy

	Stress in x direction		**Stress in Y direction**	
	MATLAB	**ANSYS**	**MATLAB**	**ANSYS**
30^0 top	486256	48624.6	43024	43023.9
30^0 bottom	55609	55609	41607	41607.2
60^0 top	38774	38773.8	69578	69577.7
60^0 bottom	40190	40190.5	62593	62593.3
30^0 top	62593	62593.3	40190	40190.5
30^0 bottom	69578	69577.7	38774	38773.8
60^0 top	41607	41607.2	55609	55609
60^0 bottom	43024	43023.9	48625	48624.6

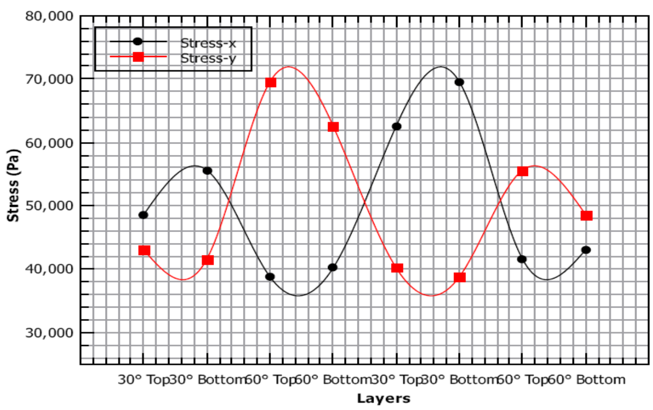


Chart -2.1.2: MATLAB results for glass/epoxy composite plate

Now as it is established that results obtain from both MATLAB coding and ANSYS simulation have sink with the analytical results.

Glass/Epoxy material is used for the analysis and the ply angle are (30/60/60/30). The above fig. 2.1.1 shows the orientation & stacking sequence for [30/60/60/30] for FEA model.

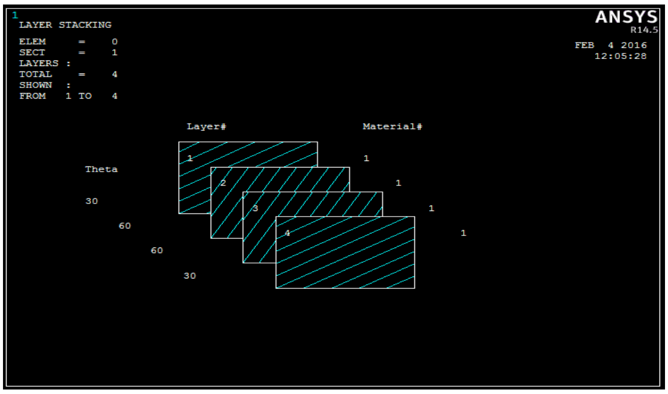
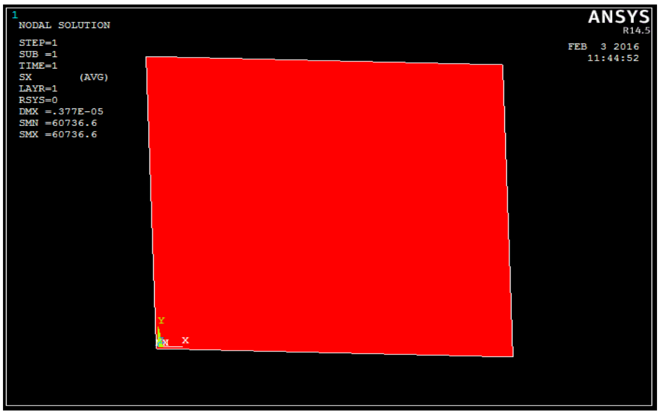
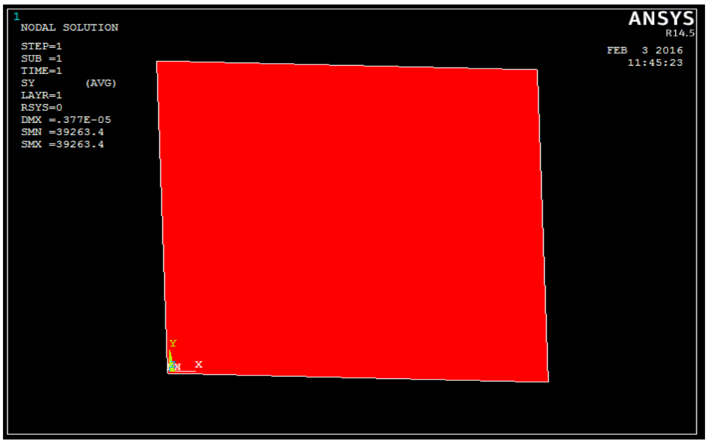


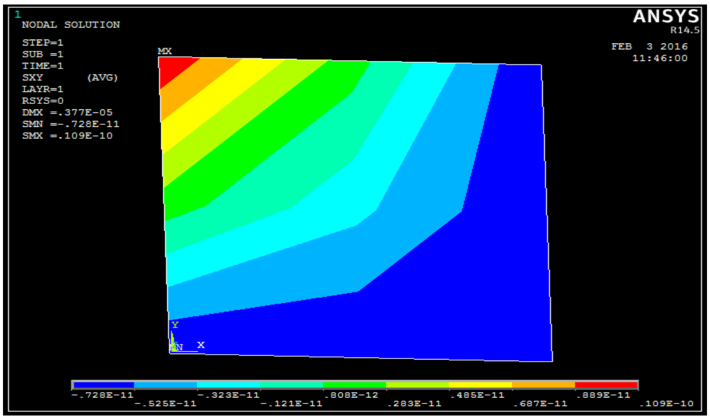
Fig-2.1.4: Orientation & stacking sequence for [30/60/60/30]



(a)



(b)



(c)

Fig-2.1.5: Stress distribution for 0^0 lamina in (a) X (b) Y and (c) XY direction

Fig 2.1.5 shows the lamina stresses X and Y direction respectivity. These resuls are compared with MATLAB simulation tool as per given in Table no.-2.1.6

Table -2.1.6: comparistion between MATLAB and ANSYS result for Glass/epoxy

	Stress in x direction		**Stress in Y direction**	
	MATLAB	**ANSYS**	**MATLAB**	**ANSYS**
30^0 top	60737	60736.6	39263	39263.4
30^0 bottom	60737	6073.6	39263	39263.4
60^0 top	39263	39263.4	60737	60736.6
60^0 bottom	39263	39263.4	60737	6073.6
60^0 top	39263	39263.4	60737	60736.6
60^0 bottom	39263	39263.4	60737	6073.6
30^0 top	60737	60736.6	39263	39263.4
30^0 bottom	60737	6073.6	39263	39263.4

The table-2.1.6 shows the result of comparisons of MAT LAB situation tool and ANSYS analyses tool.

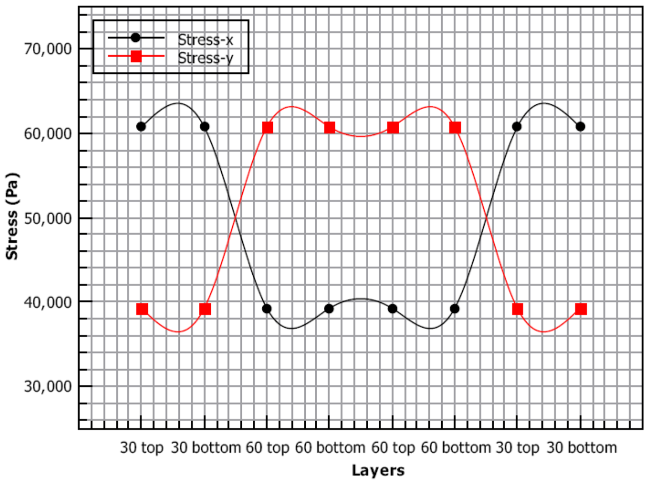


Chart -2.1.3: MATLAB results for glass/epoxy composite plate

Now as it is established that results obtain from both MATLAB coding and ANSYS simulation have sink with the analytical results.

3. CONCLUSIONS

The composite plates are analysed considering different process parameters like fiber orientation, stacking sequence and boundary conditions. The simulations were also carried out on different materials like glass/epoxy, graphite/epoxy and boron epoxy composites. Both symmetric and unsymmetrical composite plates were analyzed using mechanical conditions with the help of ANSYS simulation tool. The results obtained by classical laminate theory were compared with the results obtained from hand calculation and they are in unison with each other. The classical laminate model was also established using MATLAB simulation tool and the results were in good agreement with FEA results obtained from ANSYS.

REFERENCES

[1]. Autar Kaw, „Mechanics of Composite Materials", 2e, 2006.

[2]. SushilB.Chopade et al., "Design and Analysis of E-Glass/Epoxy Composite Monoleaf Spring for Light Vehicle", IJIRSET, Vol. 4, Issue 1, January 2015.

[3]. G. Alfano et al., "MITC nite elements for laminated composite plates", Int. J. Numer. Meth. Engng 2001; 50:707-738.

[4].B. Sidda Reddy et al., "Bending analysis of laminated composite plates using finite element method", International Journal of Engineering, Science and Technology, Vol. 4, No. 2, 2012, pp. 177-190.

[5]. Patil Deogonda et al., "Mechanical Property of Glass Fiber Reinforcement Epoxy Composites", IJSER, Volume 1 Issue 4, December 2013

[6]Pandya B.N. „Kant T., "Finite Element Analysis of Laminated Composite Plates using a Higher-Order Displacement Model". Composites Science and Technology Vol 32 (1988) pp 137-155.

[7]Somashekar B.R., Pratap G.,Ramesh Babu .C, "A Field – Consitent , Four-Node,Laminated ,Anisotropic Plate/Shell Element" , Computers &Structures Vol. 25, No. 3. pp. 345-353 ,1987.

[8]Xiao-Ping Shu, Kostas P. Soldatos "Cylindrical bending of angle-ply laminates subjected to different sets of edge boundary conditions". International Journal of Solids and Structures Vol 37 (2000) pp 4289-4307

[9]Robbins Jr DH, Reddy JN. Modelling of thick composites using a layerwise laminate theory. International Journal for Numerical Methods in Engineering 1993; 36:655–677.

[10]Gaudenzi P, Mannini A, Carbonaro R. Multi-layer higher order finite elements for the analysis of free-edge stresses in composite laminates. International Journal for Numerical Methods in Engineering 1998; 41:851–873.

[11]Zienkiewicz OC, Taylor RL, Too JM. Reduced integration techniques in general analysis of plates and shells. International Journal for Numerical Methods in Engineering 1971; 3:275–290.

[12] Zienkiewicz OC, Taylor RL. The Finite Element Method, vols. I, II. McGraw Hill: New York, 1991