IFERP
connecting engineers... developing research

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 5, May 2017

Modelling and finite element based simulation of delamination damage of a single lap joint laminated composite

[1]Muppiri Vijay Kanth, [2]Mohammed Khaja Nizamuddin, [3]Gurrapu Sneha
[1][2][3] ME Student Department of Mechanical Engineering, VCE, Hyderabad, 500031, India

Abstract: -- composite laminates are most widely applied in the field of engineering where weight ratio and delamination are the most influencing key factors. A Finite Element (FE) based simulation technique is carried in studying the delamination of single lap joint. FE based software ANSYS (ACP) has been used for this simulation. The pre-post ANSYS ACP model was developed to check the effect of delamination region with different number of plies and to check the resistance to delamination. Three-dimensional non-linear FE analysis have been carried out to study the effect on surface interlaminar stress in the bonded lap joint. In the further study, frictional forces and pressure distribution on the laminated are analyzed using the penalty formulation. Contact FE analysis is performed in order to prevent interpretation of the laminated structures.

Keywords: ANSYS (ACP), Delamination, FE analysis, Frictional force, Interlaminar stresses, Laminates.

1. INTRODUCTION

Composites are most widely applied in different fields due to its extensive properties like light weight, high strength, strength related to weight, corrosion resistance, high impact strength, design flexibility, part consolidation, dimensional stability, nonconductive, nonmagnetic, radar transparent, low thermal conductivity and durability. In this paper, we will discuss the study of delamination of single lap joined composite by a finite element based simulation technique through FE based software ANSYS(ACP). In this software, we will use pure penalty formulation to get the simulation and results. The main aim of this paper is to get the convergence of different parameters like pressure, force reactions, frictional stress by applying the boundary conditions like displacement. The results obtained after simulation will show the convergence and the graphs are plotted for the study. The composite materials used are epoxy resin UD and resin epoxy which form a layer by layer composite model.

2. WORKING IN ANSYS (ACP)

For the modelling and simulation of the work two basic models of composite materials are designed and they are lap joined or bonded by cohesive zone in the static structural module in ANSYS workbench.

Engineering data:
In this part, the materials used for the model are defined. The materials used in this work are composite materials i.e. epoxy resin UD(230Gpa), resin epoxy.
A cohesive zone with separation distance debonding is also defined.

Geometry:
In this part, sketching and generation of the surfaces is done and splitting of faces is also done to the required dimensions.

Model:
In this part, the geometry is made in to a model by generating mesh on the surface and naming the split surfaces according to the requirement.

Set up:
In this part, a pure solid model is created by adding the required ply in the layer by layer order by defining the orientation of layers.

In the same pattern, another model is created with some changes in the design and dimensions.
After creating the two models they are uploaded in the static structural module for the analysis. In this module, we define the boundary conditions and the parameters to be analyzed and analysis settings. After defining all these things, the simulation and results are obtained by solving the problem.

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 5, May 2017

3. RESULTS AND DISCUSSION

Directional deformation

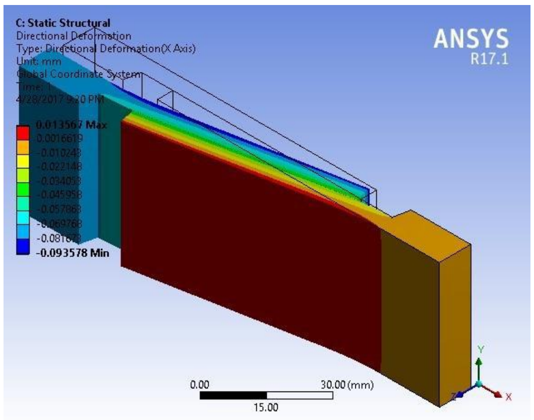


Fig.1

Table. 1

Time [s]	Minimum [mm]	Maximum [mm]
0.2	-1.8638e-002	2.6362e-003
0.4	-3.7295e-002	5.2908e-003
0.55	-5.1293e-002	7.2868e-003
0.625	-5.8294e-002	8.2867e-003
0.7	-6.5296e-002	9.2882e-003
0.75625	-7.0618e-002	1.0111e-002
0.8125	-7.5971e-002	1.0964e-002
0.89687	-8.3901e-002	1.2141e-002
1.	-9.3578e-002	1.3567e-002

The results obtained from the table. 1 and the above graph show the time vs deformation in which it is observed that at from the starting of deformation to the end of the deformation the values are increased as the time steps are increased and then falls to zero as the contact region ends. The maximum and minimum deformations are observed from the fig. 1 clearly.

Frictional stresses

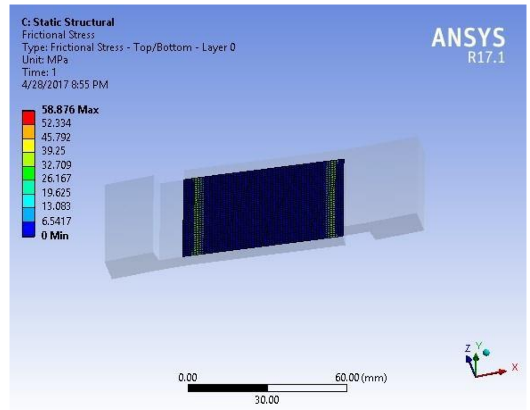
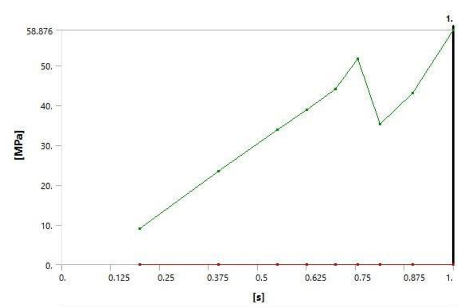


Fig. 2

Table. 2

Time [s]	Minimum [MPa]	Maximum [MPa]
0.2		8.9685
0.4		23.398
0.55		33.802
0.625		38.945
0.7	0.	44.028
0.75625		51.623
0.8125		35.25
0.89687		43.134
1.		58.876



from the above results obtained it is observed that the minimum frictional stress at any instant of time is zero and the maximum frictional stress goes on increasing with in

IFERP
connecting engineers... developing research

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering
(IJERMCE)
Vol 2, Issue 5, May 2017

the bonded or contacted region and then falls down after the contact region. Fig. 2 shows the Ansys result with frictional stresses clearly.

Pressure

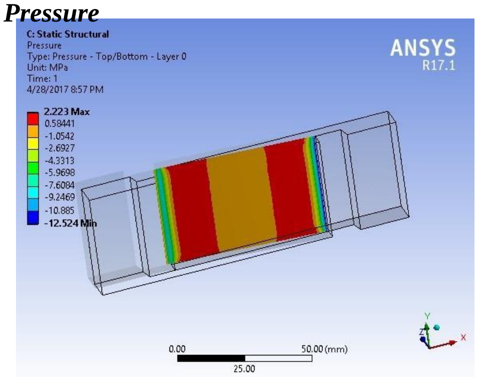
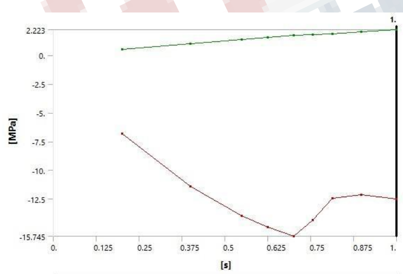


Fig. 3

Table. 3

Time [s]	Minimum [MPa]	Maximum [MPa]
0.2	-6.8326	0.5032
0.4	-11.386	0.99226
0.55	-13.985	1.3559
0.625	-14.967	1.5371
0.7	-15.745	1.7159
0.75625	-14.359	1.8078
0.8125	-12.428	1.8716
0.89687	-12.11	2.0154
1.	-12.524	2.223



From the above results it is observed that the pressure increases as the delamination takes place and reaches the maximum value at the end of the contact region as it is clearly observed from the fig. 3.

Force reactions

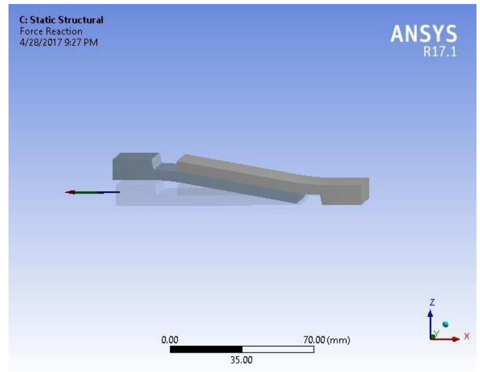
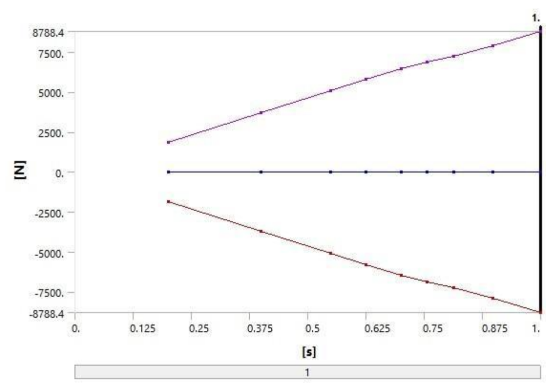


Fig. 4

Table. 4

Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
0.2	-1861.9			1861.9
0.4	-3703.7			3703.7
0.55	-5082.1			5082.1
0.625	-5769.8			5769.8
0.7	-6456.7	0.	0.	6456.7
0.75625	-6866.5			6866.5
0.8125	-7214.			7214.
0.89687	-7906.7			7906.7
1.	-8788.4			8788.4



ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering
(IJERMCE)
Vol 2, Issue 5, May 2017

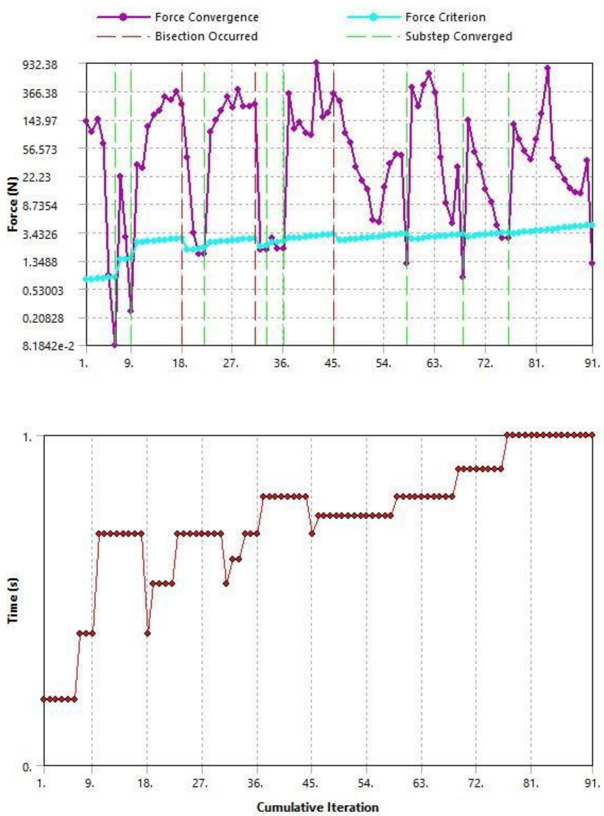


Fig. 5

Fig. 5 shows the force convergency which is obtained from Ansys solver and this shows that the results obtained are optimum. From the fig. 4 and table. 4 the force reactions are observed i.e. the reactions are obtained only in X direction because the boundary condition is given in X direction and the other directions are fixed.

4. CONCLUSIONS

From the force reactions, it is observed that convergency is obtained and which concludes that the results obtained are optimum. From part of results it is clear to conclude that every parameter reaches to maximum value at the end of the contact region and then suddenly falls down, which shows the nature of the bonded region at the time of delamination or separation.

5. REFERENCES

[1] Panigrahi, S.K and Pradhan, B., 2007, "Adhesion failure propagation in adhesively bonded single lap laminated FRP composite joints", Journal of Adhesion science and technology, 21, pp. 379-398.

[2] Panigrahi, S.K and Pradhan, B., 2007, "Delamination damage analyses of adhesively bonded lap shear joints in laminated FRP composites", International journal fracture, 148, pp, 373-385.

[3] Panigrahi, S.K and Pradhan, B., 2007, "Three-dimensional failure analyses and damage propagation behavior of adhesively bonded single lap joints in laminated FRP composites", Journal of reinforced plastics and composites, 26, pp, 183-201.