FEM Analysis of High Impact Velocity on Composite Laminated Plates

Suman Anand[1] Himanshu Singh Rathore[2] Ved Parkash[3] Chandra Mohan Kumar[4] Chandan Kumar[5]
[1,2,3,4,5]Department of Mechanical Engineering
[1,2,3,4,5]Swami Keshvanand Institute of Technology, Management & Gramothan, Jaipur

Abstract— High velocity impact analysis is important study for various industries like defense making equipments, space industries, safety helmet industries. These types of problems are very difficult to analysis but computer simulation provides vast power to researchers to conduct these type of researches. The failure of an AL6061-T6 (Metal Alloy), E- Glass and Kevlar-29 composite plate under high-velocity impact from an structural Steel projectile tool was investigated using the nonlinear explicit finite element software, Ansys Autodyn. Two velocities of 350 m/s and 700 m/s were used for FEM simulation. Since AL6061-T6 is more resistant than composite material at various velocity ranges but composite material are light in weight than AL6061-T6 and have more application in various industries. Dimension and mass of tool play important role in failure conditions. Optimization of plate thickness was also done in this study using Ansys optimization tool.

Key words: Ansys Autodyn, FEM Analysis, AL6061-T6, Kevlar-29, E- Glass

I. INTRODUCTION

High velocity impact is of concern to many different industries like defense and aviation industries. Armor flexibility and impact resistance are extremely important in warfare applications. Blast phenomena leads to portion progress which in turn leads to impact. In space travel applications impact plays a vital role in designing the sacrificial armor against the debris. Latest innovations like friction stir welding and repair require the data of impact to read the impact event to exactly assess the damage and repair parameters. In early days metals armors were used, now with advent of composites light weight armor materials are introduced which are more portable. Lighter materials increase the flexibility and portability. As initial velocity of projectile tool is above the ballistic limit, the residual velocity becomes of interest since it can pose threat to the wearer. Therefore, it is vital to have better understanding on the response of products used ballistic velocity limits when struck by projectile at that impact velocity limit before one could design a better engineering product.

II. FINITE ELEMENT ANALYSIS

Finite element analysis is based on the method of domain and boundary dissertation which reduces the infinite number of unknowns defined at element nodes. It has two primary subdivisions. The first utilizes discrete element to obtain the joint displacements and member forces of a structural framework. The second uses the continuum elements to obtain approximate solutions to heat transfer, fluid mechanics and solid mechanics problems. The response of composite materials during ballistic impact can also be determined by using finite element analysis apart from experimental testing. Although this method has become a popular trend in characterizing composite materials, it must be used with a precaution and be always validated by experimental work. It is also doubtful that experimental testing can be replaced totally by finite element analysis; rather it is probably a compliment to each other.

III. PROBLEM DESCRIPTION

Thick composite material made plate was investigated using FEM technique to know high velocity impact analysis of plate. Meanwhile, finite element analysis software used in this study was Ansys Autodyn. The complex response of composite materials coupled with high costs and limited amount of data from ballistic testing has lead to experimental characterisation of ballistic helmet becomes expensive and time consuming. In order to address this issue, finite element analysis can be used as a method to characterise the response of composite ballistic material and to obtain valuable information on parameters affecting impact phenomena.

IV. OBJECTIVE OF RESEARCH

The main focus of this research work is to study the response of thick plate made of composite materials when impacted at high velocity by using finite element analysis. The objectives of this research are:

- To determine the effect of high velocity impacts on thick plates made of composite materials.
- Three materials are simulated to find high velocity impact on their structure. First material is alloy of Aluminium named AL6061-T6, second and third are composite materials used in Aviation industry and war industry etc.
- To analyze the deformation as well as residual velocity distribution of the thick plate when struck by a bullet at velocity of 350 m/s and 700 m/s .
- To evaluate the deformation mechanism occurred on thick plate after the impact.

V. FEM MODELING DETAILS

In this study test plate materials are AL6061-T6, KEVLAR-29 and E-Glass where as tool projectile material is structural steel. Targeted test plate is square and tool is in shape of bullet. Various researchers found that boundary conditions used for high velocity impact analysis play very less role in final results. So reduction in test plate size is not affecting the FEM results and user can reduce computational time. rigid body formulation is used for tool projectile, no deformation is taken into consideration in tool projectile and flexible body formulation is used for test plate. To save computational time reduction in size of projectile and test plate and distance between tool and plate is good criteria for FEM modeling. In Figure 1 and 2 meshed domain of plate and tool were shown.

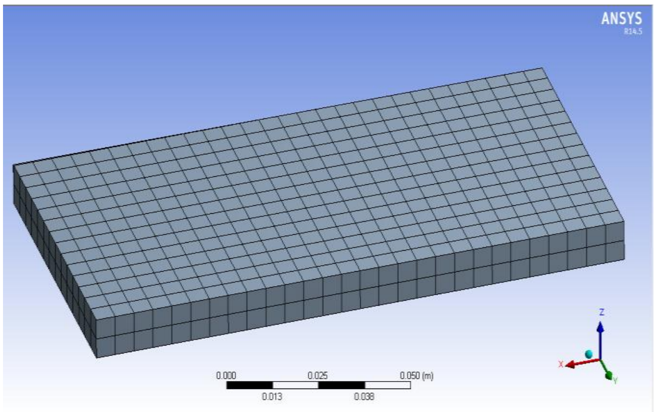

Fig. 1: Meshed Domain of Plate of AL6061-T6

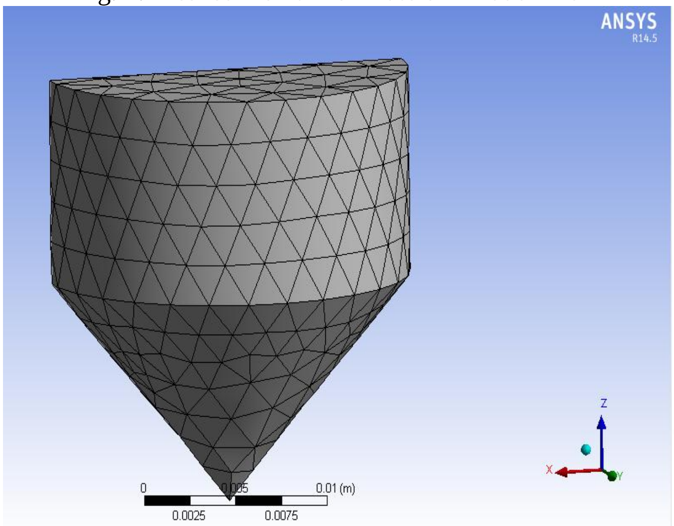

Fig. 2: Meshed Domain of Tool of Structural Steel

Material properties of all materials were taken from Ansys Data Book. Table 1 shows the properties of structural steel.

Property	Value	Unit
Density	7850	Kg/m^3
Young's Modulus	2E+11	Pa
Poisson's Ratio	0.3	
Bulk Modulus	1.66E+11	Pa
Shear Modulus	7.692E+10	Pa
Sp. Heat	434	J/kg C

Table 1: Properties of Structural Steel

VI. RESULT DISCUSSION

The shear stresses, equivalent stress, deflection of test plate and velocity decrement of the projectile tool are the important parameters. The structural behavior of test plate after 1µ sec time analysis at two velocities 350 m/s and 700 m/s are shown in Figure.

A. *Equivalent Stress Profile:*

The equivalent stress represents envelope of direct and shear stress components and based on various failure theories. In Ansys various failure theories are merged but in our case von-misses failure criteria was used for equivalent stress analysis.

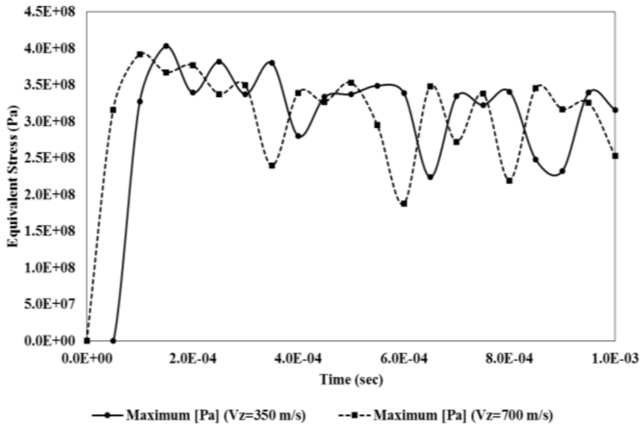

Fig. 3: Equivalent Stress for AL6061-T6

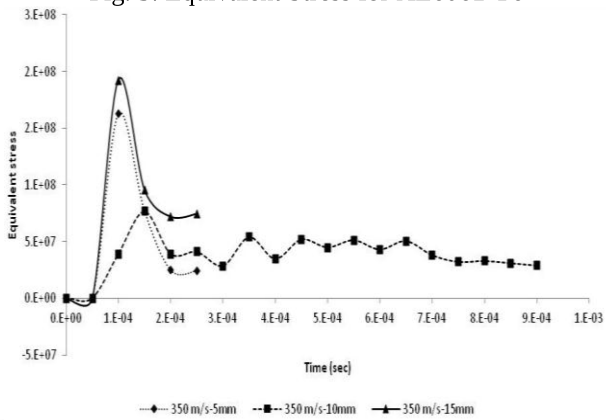

Fig. 4: Equivalent Stress for E- Glass at velocity of 350 m/s

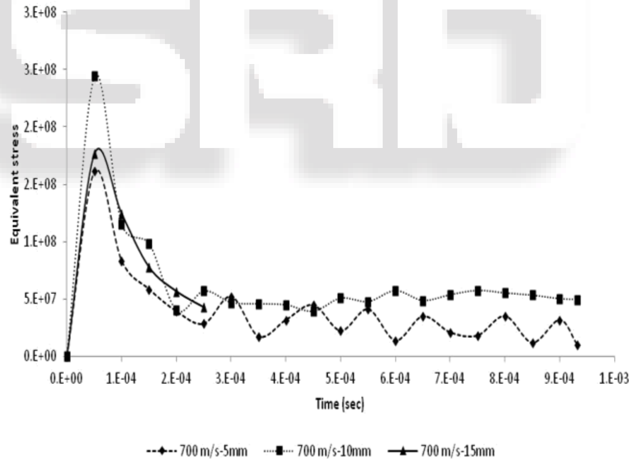

Fig. 5: Equivalent Stress for E- Glass at velocity of 700 m/s

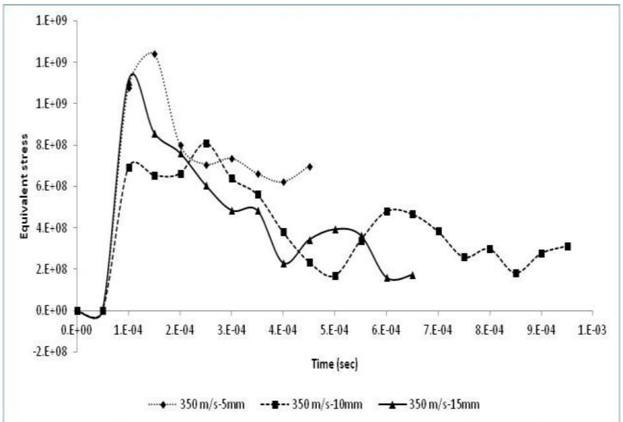

Fig. 6: Equivalent Stress for Kevlar-29 at velocity of 500 m/s

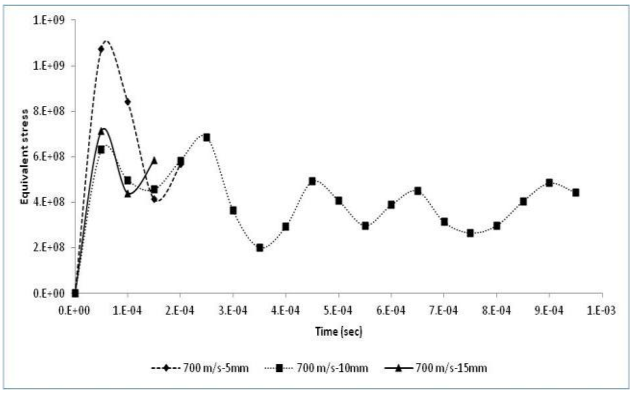

Fig. 7: Equivalent Stress for Kevlar-29 at velocity of 700 m/s

When thickness of plate was decreased or increased it was shown that equivalent stress was decreased or increased and all these changes are because of material properties used in Ansys simulation environment. From above figure it was shown that when thickness was increased equivalent stress was also increased. It was observed that when orthographic material model was used equivalent stress was show more reliable results because in this model direction effect was merged in failure theory.

B. *Deformation Profile:*

When a projectile tool was impact on test plate there was a deformation on plate which was changed according to magnitude of velocity of tool, material properties of plate, local environment of system, shape of tool, boundary conditions of plate etc. Figures show deformations for various test setups performed on simulation environment of Ansys Software.

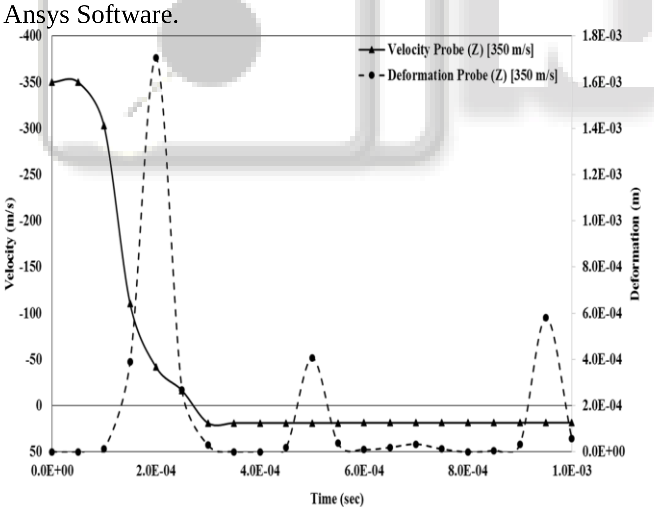

Fig. 8: AL6061-T6 Plate Deformation at Tool Velocity 350 M/S

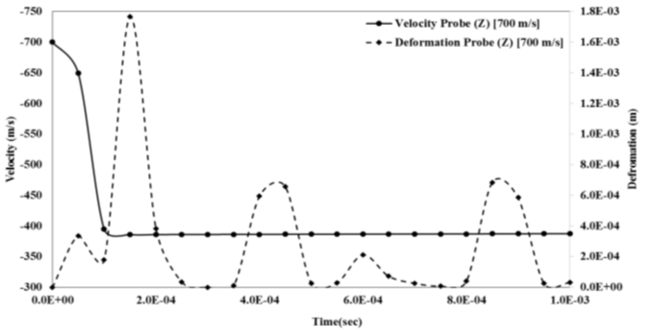

Fig. 9: AL6061-T6 Plate Deformation at Tool Velocity 700 M/S

When deformation profile for Al alloy made plate was analysis it was observed that plate got hole when velocity was approached to 700 m/s which was shown in figure. At velocity 350 m/s tool changed its direction because of material elastic strength which resists this high velocity impact.

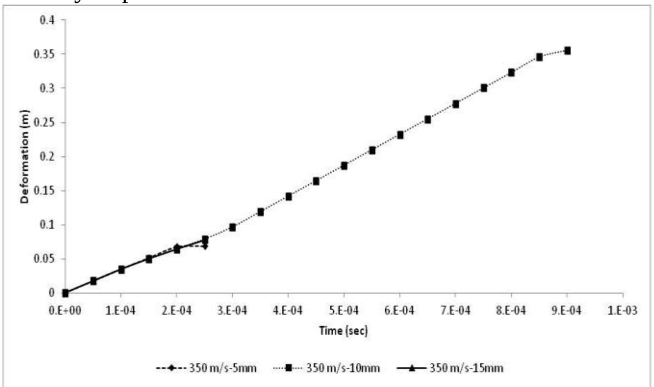

Fig. 10: E- Glass Plate Deformation at Tool Velocity 350 M/S

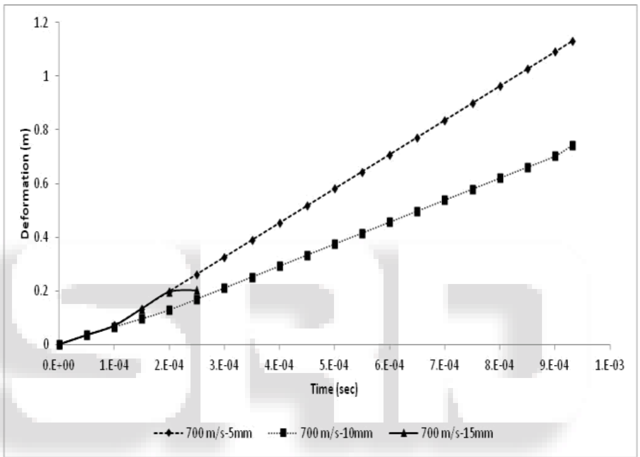

Fig. 11: E- Glass Plate Deformation At Tool Velocity 700 M/S

When this deformation was occurring on plates, rigid tool give its kinetic energy to plate and that's why deformation take place on local area of plate. Conservation of momentum was used to calculate kinetic energy changed for system. In this study no deformation was occur on tool because it was treated as rigid material.

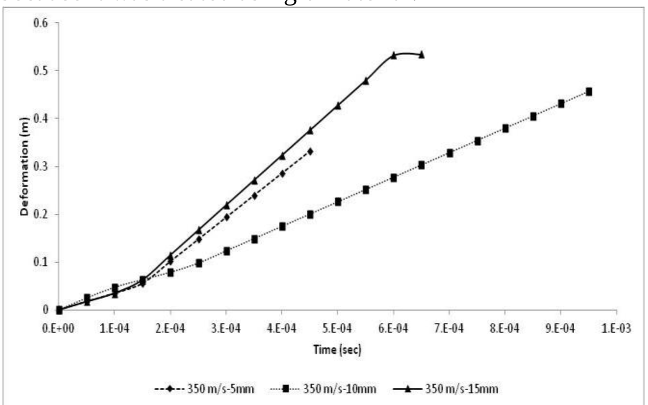

Fig. 12: Kevlar-29 Plate Deformation At Tool Velocity 350 M/S

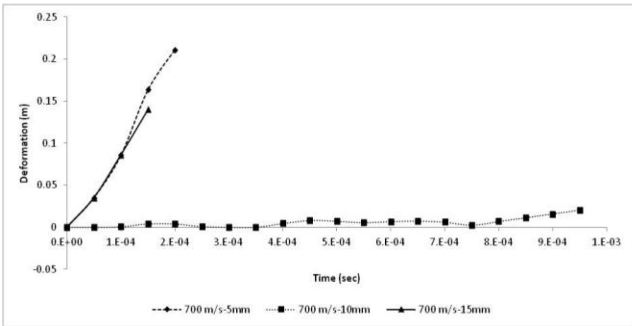

Fig. 13: Kevlar-29 Plate Deformation at Tool Velocity 700 M/S

Deformation profile for KEVLAR-29 plate at various velocity ranges for three different thickness of plates and it was observed that plate got hole at velocity 350 m/s or higher for all thickness of plate. In figures deformation unit was gone to very high but this was post processing error from software. Most of the deformation was occur in 4.0E-04 sec of impact.

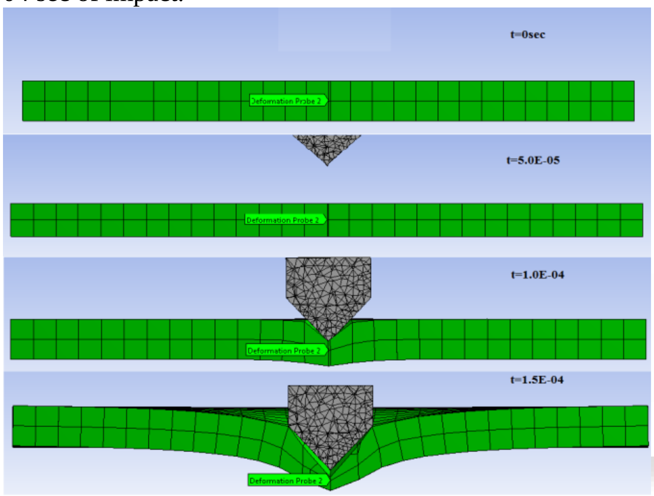

Fig. 14: AL6061-T6 Deformation Rate at Vz=350 M/S

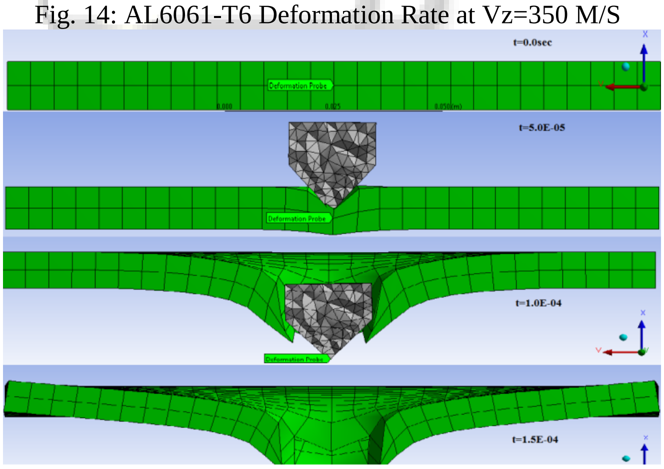

Fig. 15: AL6061-T6 Deformation Rate at Vz=700 M/S

VII. CONCLUSION

From this study it was clear that impact velocity was dependent on impactor mass and volume also. The main focus of this analysis has to study the explicit dynamic model of the laminate that accurately predicts the ballistic response of laminated plate at different velocities and thickness. Many results, e.g. Equivalent stress profile, Shear stress profile, Energy profile, Deformation profile encourage the use of computer simulation using FEM approach in the design of ballistic light armors in order to avoid the time consuming and expensive experiments. The present finite element model also successfully simulated the progressing damage from the initial impact to the final penetration of the composite plate.

VIII. FUTURE SCOPE

For getting a such real world simulation the media of transition like fluid medium may be implemented with some viscosity and damping effect. The thermal output model coupled with laminate theory gives reasonable results for predictions of mechanical behaviour under load. Also thermal effects are to be considered in terms to study the frictional heat generated and possible outcomes in the process.

REFERENCES

[1] C.T. Lim, V.P.W. Shim, Y.H. Ng, “Finite Element Modeling of the Ballistic Impact of Fabric Armor”, International Journal of Impact Engineering, Vol. 28, Issue 1, PP. 13-31, Jan 2003
[2] Hasan Mahfuz, Yuehi Zhu, Anwarul Haque et al., “Investigation of High Velocity Impact on Integral Armor using FEM”, International Journal of Impact Engineering, Vol. 24, Issue 2, PP. 203-217, Feb 2000.
[3] J. Timothy Hartness, Peter O. Sjoblom, Tobey M. Cordell, “On Low Velocity Impact Testing of Composite Materials”, Journal of Composite Materials, Vol. 22, Issue 1, PP. 30-52, Jan 1988.
[4] Potts J. Frank & Oler J. Walter, “Finite Element Applications with Micro-computers”, ISBN 0-133-17439-5, Prentice Hall Englewood Cliffs, N.J., 1989.
[5] Artur Portela & Abdel Charafi, “Finite Elements using Maple: A Symbolic Programming Approach”, ISBN 3-540-42986-7, Springer-Verlag Berlin Heiddelberg, New York, 2002.
[6] Serge Abrate, “Impact on Composite Structures”, Cambridge University Press, 1998.
[7] N. K. Naik, P. Shrirao, “Ballistic Impact Behaviour of Woven Fabric Composites: Parametric Studies”, Aerospace Engineering Department, PP. 105-116, 2004.
[8] Lexinston & K.Y., “Finite Element Simulation of Ballistic Impact in Studies”, MSME Thesis, 2003.
[9] M. H. Datoo, “Mechanic Of Fibrous Composites”, Elsevier Applied Science, 1991.
[10] M. Grujicic, J. S. Snipes, S. Ramaswami, R. Yavari, “Discrete Element Modeling and Analysis of Structural Collapse/Survivability of a Building Subjected to Improvised Explosive Device (IED) Attack”, Advances in Materials Science and Applications, Vol. 2, Issue. 1, PP. 9-24, March 2013.
[11] Autar K. Kaw, “Mechanical of Composite Materials”, CRC Press, 2006.
[12] “Penetration Equation Hand Book for Kinetic Energy Penetrations”, Joint Technical Coordinating Group for Munitions Effectiveness, 61 JTCG/ME-77-16, Oct 1985.

[13] Jack Van Hoof, "Modelling of Impact Induced Delamination in Composite Materials", ISBN 0-612-52335-7, National Library of Canada, 1999.
[14] A.P.Mouritz, E Gellert, P.Burchill, "Review of Advanced Composite Structures for Naval Ship and Submarine", Composite Structural, PP. 21-42, 2000.
[15] N.K. Naik, A.V. Doshi, "Ballistic Impact Behavior of Thick Composites: Parametric Studies", Aerospace Engineering Department, PP. 447-464, 2008.
[16] R.A.W. Mines, A.M. Roach, N. Jones, "High Velocity Perforation Behaviors of Polymer Composite Laminates", International Journal of Impact Engineering, PP. 561-588, 1999.
[17] Amit Kanvinde and Gregory Deierlein, "Prediction of Ductile Fracture in Steel Moment Connections during Earthquakes using Micromechanical Fracture Models", 13th World Conference on Earthquake Engineering, PP. 297, 2004.
[18] M.A.G. Silva, C. Cismasiu, C.G. Chiorean, "Numerical Simulation of Ballistic Impact on Composite Laminates", International Journal of Impact Engineering, Vol. 31, PP. 289–306, 2005.
[19] Antonio Avila, Duarte, Marcelo I. Soares, "The Nanoclay Influence on Impact Response of Laminated Plates", Latin American Journal of Solids and Structures, Issue 3, PP. 3-20, 2006.
[20] David M. Barrett, "A Study of the Modeling of Ballistic Impact and Penetration of Thin Plates using DYNA-3D", M.S.M.E. Thesis, Mechanical Engineering Dept., University of Washington, Seattle, WA, 1993.