International Journal of Engineering Research and Development
e-ISSN: 2278-067X, **p**-ISSN: 2278-800X, www.ijerd.com
Volume 7, Issue 11 (July 2013), PP. 75-82

High Velocity Impact Analysis on "Composite Material Block" Using FEM Approach

Suman Anand[1], Kunal Sharma[1]
[1]*Rajasthan Institute of Engineering & Technology (RTU), Bhankrota, Jaipur, Rajasthan*

Abstract:- The failure of an AL6061-T6 and Kevlar-29 composite plate under high-velocity impact from an S-S (structural Steel) projectile tool was investigated using the nonlinear explicit finite element software, ANSYS AUTODYAN. Two velocity ranges 350 m/s and 700 m/s were used for FEM simulation. It was analyzed that AL6061-T6 is more resistant than KELVAR 29 at both ranges but Kevlar 29 is light in weight than AL6061 and have more application in aviation industry. The composite laminate and the tool were venerated by solid FEM elements. The contact between the tool and plate was simulated using a surface-to-surface eroding contact algorithm. It was also observed that tool dimension and mass play important role in failure conditions.

Keywords: - AL6061-T6, Kevlar, high velocity impact, FEM approach, Autodyan

I. INTRODUCTION

High velocity impact is of concern to many different fields and has been the subject of much research, especially in the last 50 years. Over this period of time, the methods used to analyze impact have changed naturally, as have the disciplines interested in these analyses. Researchers are still trying to get a clear cut picture of the impact performance. Mainly this applies to the defense industry. Armor flexibility and impact resistance are extremely important in warfare applications.

Many theories and procedures emerged to study the impact and blast phenomena. 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. Low velocity impacts can cause severe damage to soft material like muscle tissue. 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.

II. FINITE ELEMENT ANALYSIS: EXPLICIT DYNAMICS

The Explicit Dynamics method is made to enable to simulate nonlinear structural mechanics applications involving high velocity impact analysis, stress wave propagation, high frequency response, large deformations, material model and behaviour, structural buckling and failure between bonded surfaces like welds, bolted joints etc.

Explicit Dynamics is most suited to simulate events which take place over very short periods of time, a few milliseconds or less. Simulation results which last more than 1 second can be modelled; however, long run times can be expected. Techniques such as mass scaling and dynamic relaxation are available to improve the efficiency of simulations based results with long durations.

In an Explicit Dynamics simulation based solution, user start with a discretized domain (meshing of geometry) with assigned material properties making with various models available in simulation codes, loads, constraints and initial velocity conditions. This initial state, when integrated in time, will produce motion at the node points in the mesh. Figure a shows clear picture of explicit dynamics simulation.

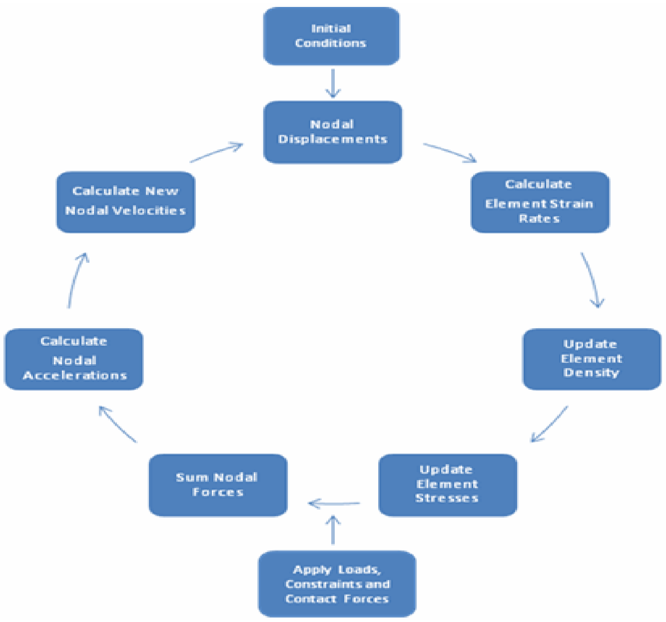

Figure 1 (A) Overview of Explicit Dynamics Methodology [Mayer & Zukas]

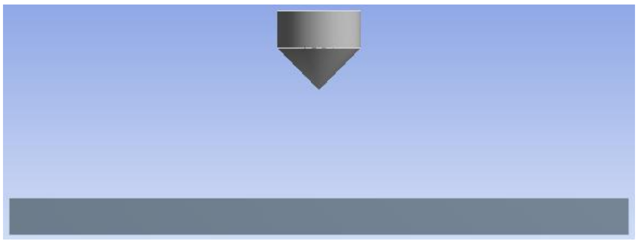

Figure 1 (a) Front View

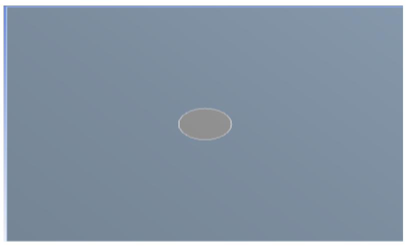

Figure 1 (b) Top View

III. PROBLEM DESCRIPTION

Finite element analysis was conducted for the AL6061-T6/ Kevlar-29 composite plate as shown in Fig. 1.The test plate has dimensions of 150 (length) × 150 (width)× 10 (thickness) (unit: mm). The two opposite edges are free, and the remaining two edges are fixed. The impact velocities considered for test are 350 and 700 m/s in negative z direction. The impactor tool has a diameter of 20 mm and weight of33.4 g. The tool impacts upon the center of the test plate from initial distance of 50 mm as shown in figure 1 (b). For finite element modeling result, a cutting plane condition is used to show difference change at center. The composite material properties of the test plate and the impactor tool are given in Tables 1 to 3, respectively.

IV. ASSUMPTIONS AND BOUNDARY CONDITIONS

The following assumptions are used in current study.

1. Impact tool is considered as rigid in analysis.
2. Initial velocity of tool is in negative z direction only.
3. Automatic mesh sizing tool is used for mesh generation.
4. General material model is used for tool material
5. Time step is selected by assumption only.

V. MATERIAL PROPERTIES

In this study two different composite materials are used for FEM analysis. AL6061-T6 and KEVLAR-29 are those two materials. Al6061 Steinberg Guinean strength model is used and for KEVLAR orthographic material model is used. A material is considered as an orthotropic material if there are three perpendicular directions and has only three perpendicular planes of material symmetry (Datoo, 1991).

Table 1 Plate Material AL6061-T6 (Ansys source)

Property	Value	Unit
Density	2703	Kg/m^3
Sp. Heat	885	J/kgC

Steinberg Guinean Strength		
Initial Yield Stress	2.9E+08	Pa
Max Yield Stress	6.8E+08	Pa
Hardening Constant	125	NA
Hardening Exponents	0.1	NA
Derivative dG/dP	1.8	NA
Derivative dG/dT	-1.7E+07	Pa/C
Derivative dY/dP	0.018908	NA
Melting Temperature	946.85	C
Shear Modulus	2.76E+10	Pa
Shock EOS Linear		
Gruneisen Coefficient	1.97	NA
Parameter C1	5240	m/s
Parameter S1	1.4	NA
Parameter Quadratic S2	0	s/m

Table 2 KEVLAR-29 (hoof 1999)

Property	**Value**	**Unit**
Density	1230	Kg/m^3
Young’s Modulus		
E11	18.5	GPa
E22	18.5	GPa
E33	6.0	GPa
Poisson’s Ratio		
v12	0.25	
v13	0.33	
v23	0.33	
Shear Modulus		
G12	0.77	GPa
G13	5.43	GPa
G23	5.43	GPa
Strength		
X	1850	MPa
Y	1850	MPa
Z	1200	MPa
S12	77	MPa
S13	543	MPa
S23	543	MPa

All properties required for FEM simulation are shown in table 1, 2 and 3 respectively. For tool simple material model is considered and treated it as rigid material.

Table 3 Tool: 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

VI. RESULT AND DISCUSSION

The shear stress, equivalent stress, deflection of test plate and velocity decrement of the impactor tool are the important parameters in the FEM simulated results for a high impact velocity on a composite test plate. The behaviour of test plate after 1µ sec time analysis at two velocities (350 m/s and 700 m/s) are shown in figure 2 to figure 5 for various parameters

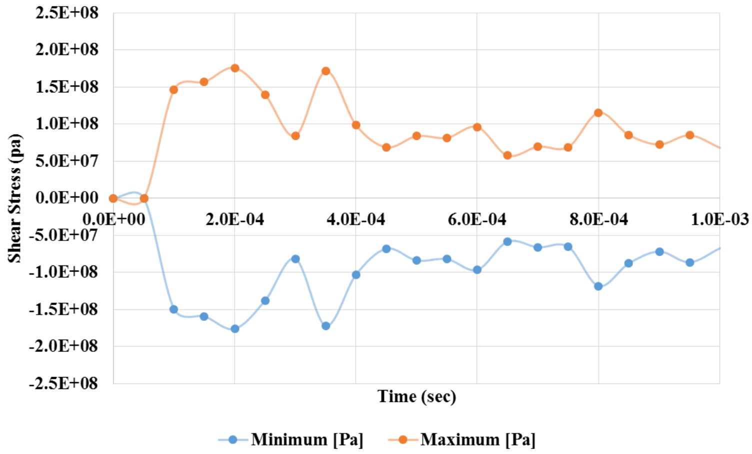

Figure 2 (a) Shear Stress at Vz=350 m/s (AL6061)

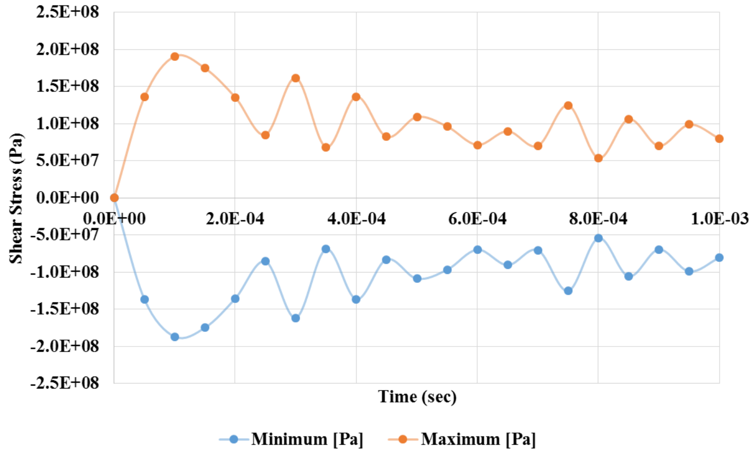

Figure 2 (b) Shear Stress at Vz=700 m/s (AL6061)

As shown in figure 2 it was clear that development of shear stress is high at 700 m/s velocity range for both materials used in FEM simulation. But max shear stress development was obtained in Al6061-T6 plate as shown in figure 2 (b).

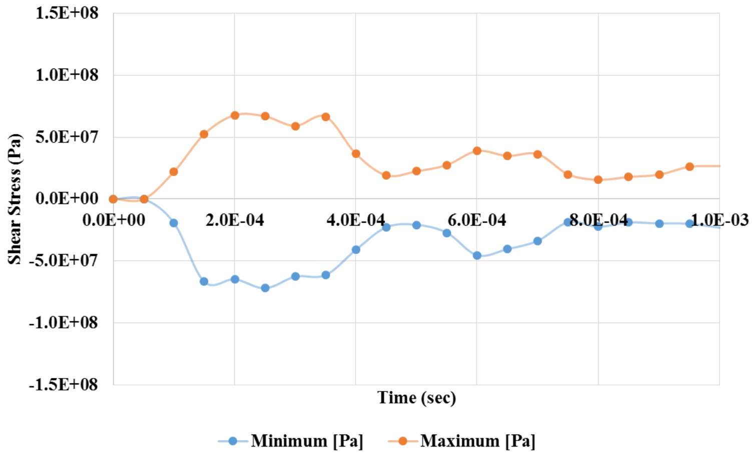

Figure 2 (c) Shear Stress at Vz=350 m/s (KEVLAR-29)

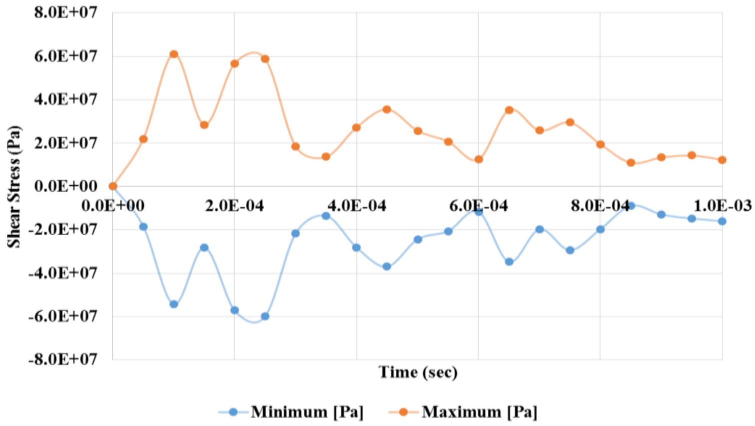

Figure 2 (d) Shear Stress at Vz=700 m/s (KEVLAR-29)

As we know Al6061 has very high density than KEVLAR material and because of it AL6061 has more resistant than KEVLAR but elastic limit of KEVLAR was more than AL6061 and try to deform less than Al6061 which was shown clear in figure 3.

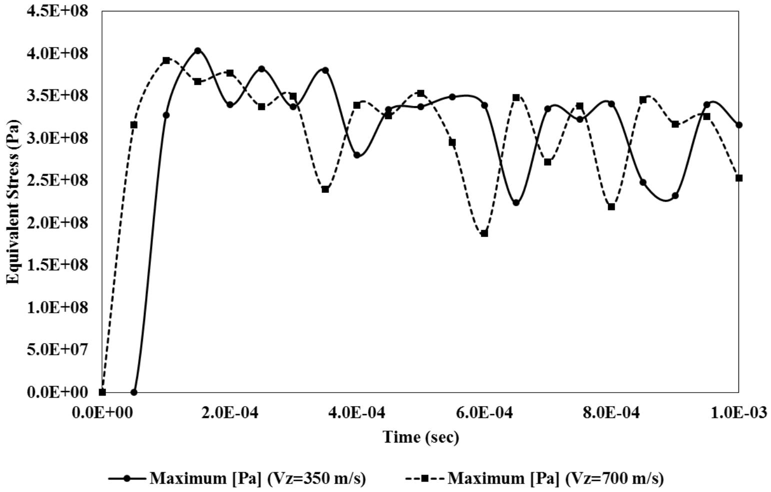

Figure 3 (a) Equivalent Stress for AL6061-T6

In this study it was try to observe velocity decrement of tool impactor on various velocities ranges at different material plates. From figure 4 (a) it was shown that when tool impact with 700 m/s velocity at test plate made of Al6061-T6 was deform at 1.0E-04 sec from initial time of impact of tool and after that tool get uniform velocity in same direction.

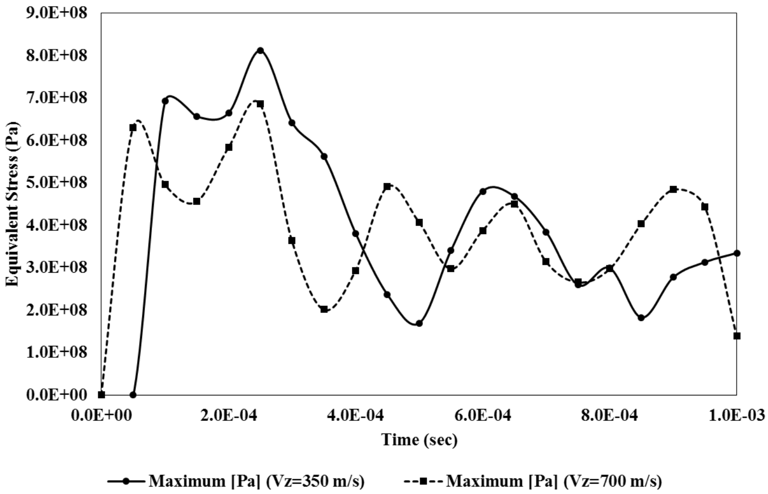

Figure 3 (b) Equivalent Stress for KEVLAR-29

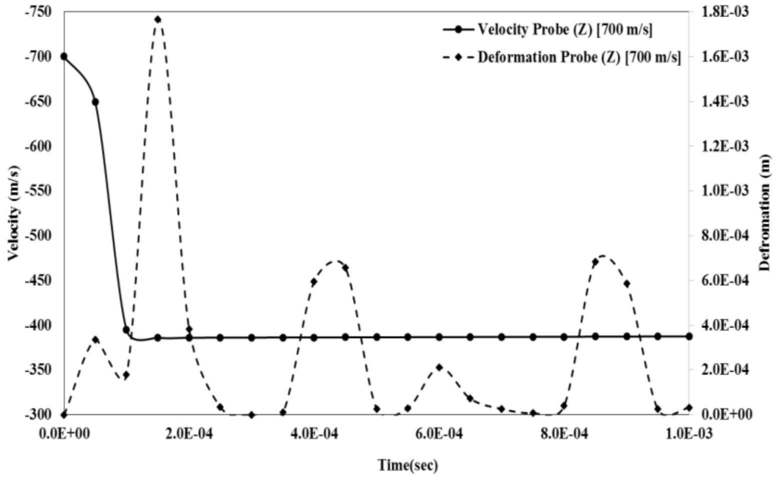

Figure 4 (a) AL6061-T3 Plate Deformation at Tool velocity 700 m/s

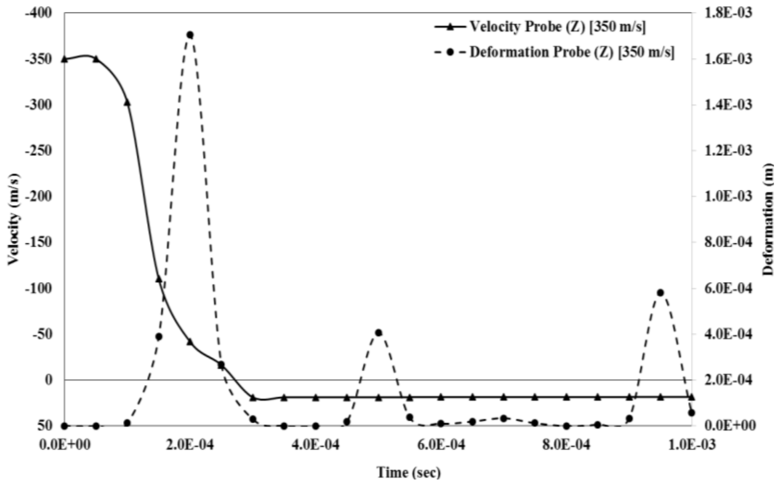

Figure 4 (b) AL6061-T3 Plate Deformation at Tool velocity 350 m/s

From analysis of figure 4(b) it was shown that tool was try to deform plate at velocity range of 350 m/s but after 3.0 E-04 sec tool reflect back in current direction and gain uniform velocity till end time. Here tool cannot make hole in test plate like previous velocity range.

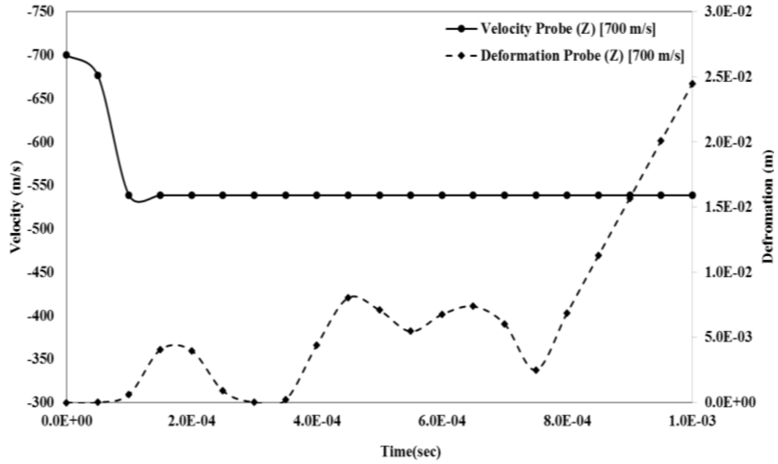

Figure 4 (c) KEVLAR-29 Plate Deformation at Tool velocity 700 m/s

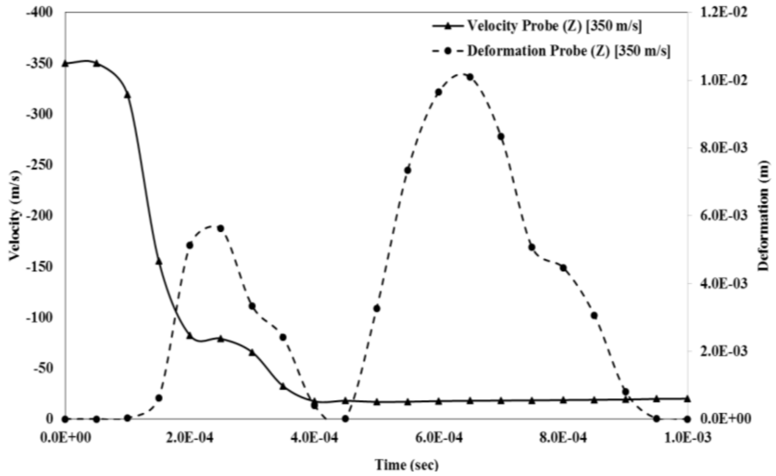

Figure 4 (d) KEVLAR-29 Plate Deformation at Tool velocity 350 m/s

Tool velocity decrement analysis for KEVLAR-29 material was shown in figure 4 (c) and 4 (d). It was shown that for both velocity ranges of tool was make holes in test plate made of KEVLAR composite material. Due to high elastic limit of material more deformation was shown rather than Al6061 made plate.

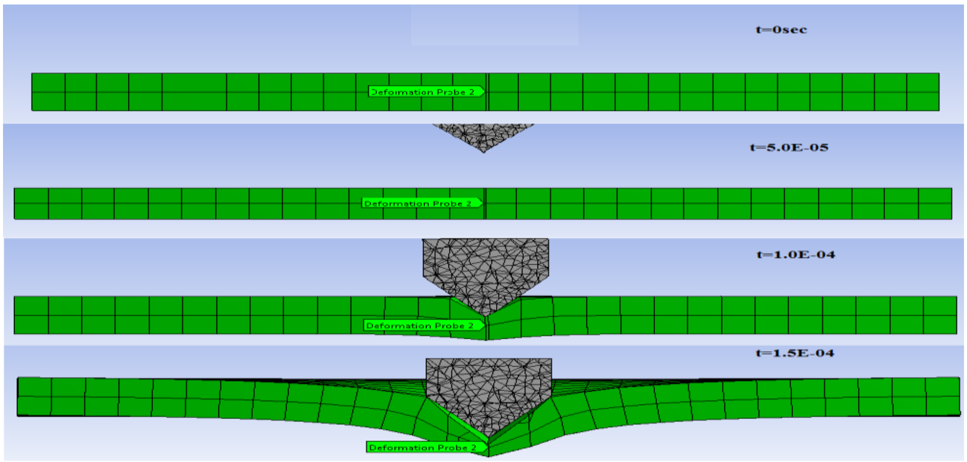

Figure 5 (a) AL6061-T6 Deformation rate at Vz=350 m/s

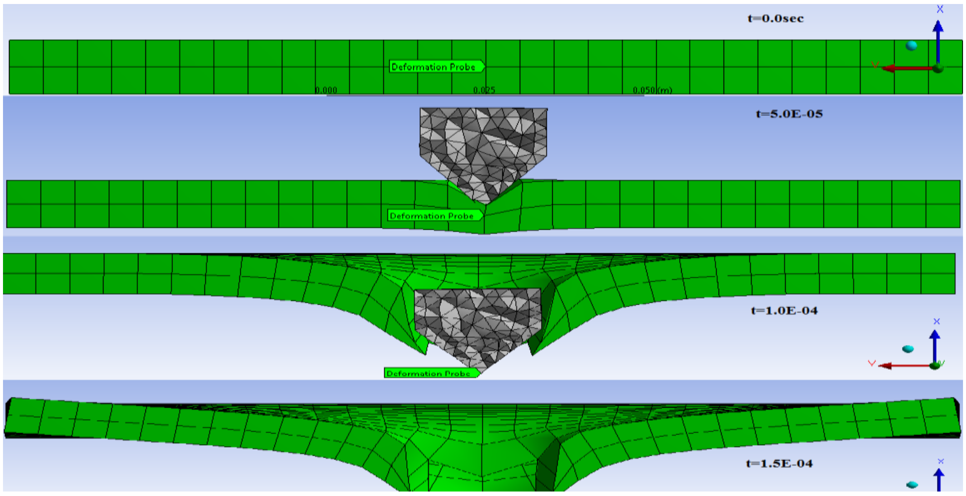

Figure 5 (b) AL6061-T6 Deformation rate at Vz=700 m/s

Figure 5 is visual representation of deformation in various time ranges at two velocity ranges for Al6061-T6 composite plate. It clearly shows that high velocity impact play important role in testing of materials importance in aviation industry. This figure shows capability of FEM technique.

VII. CONCLUSIONS

A finite element model using Ansys Autodyan was developed to simulate the high-velocity impact reaction of anAL6061-T6 and Kevlar29 composite plate. The interaction between the impactor tool and the laminate was simulated using a surface-to-surface eroding contact algorithm. Numerical analyses were conducted at two impact velocities of 350 and 700 m/s of a structural steel impactor. From this study it was clear that impact velocity was dependent on impactor mass also. The present finite element model also successfully simulated the progressing damage from the initial impact to the final penetration of the composite plate.

REFERENCES

[1]. Meyers, M. A., (1994) “Dynamic behavior of Materials”, John Wiley & Sons, ISBN 0-471-58262-X.

[2]. Finite Element Simulation of Ballistic Impact in Ballistic Studies , MSME thesis, Lexington, KY 2003.

[3]. Ipson, T.W. and R.F. Recht. “Ballistic-penetration Resistance and its Measurement.”Vol. 15, pp 247-256, 1975

[4]. “Penetration Equations Handbook for Kinetic Energy Penetrators.” Joint Technical Coordinating Group for Munitions Effectiveness, 61 JTCG/ME-77-16, Oct. 1985.

[5]. Zukas, Jonas A. “Survey of Computer Codes for Impact Simulation.” High Velocity Impact Dynamics. Ed. Jonas A.Zukas. New York: John Wiley and Sons, pp. 593-714, 1990.

[6]. Chen, E.P. “Finite element Simulation of Perforation and Penetration of Aluminum Targets by Conical-Nosed Steel Rods.” Mechanics of Materials, Vol. 10, 1990.

[7]. Bamman, D.J., “Prediction of Ductile Failure in Metal Structures.” Applied Mechanics Department, Sandia National Laboratory.

[8]. Barrett, David M. “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.

[9]. Yoo, Y.H. and M. Lee. “A Three-Dimensional FE Analysis of Large Deformations for impact Loadings Using Tetrahedral Elements.” Computational Mechanics. Vol 30, pp. 96-105, 2003.