ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 4, Issue 4 , April 2017

Numerical Simulation of Impact on Aluminum (2024) Plate Copper coated Steel (4340) Cylinder

Vijay Kumar Gupta, AnuragShrivastava, VivekNarain, Shailendra Kumar

P.G Student,Department of Mechanical Engineering, SRIMT, Lucknow, India
Assistant Professor, Department of Mechanical EngineeringSRIMT, Lucknow, India
Assistant Professor,Department of Mechanical Engineering, BBDNITM, Lucknow, India
Assistant Professor, Department of Mechanical Engineering, BBDEC, Lucknow, India

ABSTRACT: This paper presents the results of the energy absorbed and velocity reduction due to impact of 9mm copper coated steel (4340) cylinder on aluminium (2024) plate targets. The impact on aluminum (2024)plate has been subjected to a high velocity range of 400-700m/s. This analysis is done by the ANSYS Workbench 14.0 software under the explicit non-linear dynamic analysis and ANSYS AUTODYN is used as a solver. The results obtained via simulation develop the equation between the Impact velocity and Residual velocity of cylinder within the range of 400-700 m/s., which is also valid for higher range of velocity with less percentage variation.

KEYWORDS: FEM, Energy absorption, ANSYS Workbench, high velocity

I. INTRODUCTION

Past few decades have seen an increasing application of Computer Aided Engineering (CAE) for simulation of crash phenomenon particularly due to development of high computing machines and parallel computing techniques. The increase in safety standards can be attributed to the improvement of structural crashworthiness performance through Finite Element Analysis (FEA). High velocity impact is of interest of many different fields and has been subjected too much research especially in the last 50 years. Over this period of time, the methods used to analyse impact have changed dramatically, as have the disciplines interested in this analysis. Researchers are still trying to get a clear cut picture of impact behaviour. Mainly this applies to the defence, industry. Armor resilience and impact resistance are extremely important in warfare applications.[1] Although there has been a tremendous increase in application of software's to solve the problems related to automotive industry, aerospace industry and drop test of components, still the best is yet to come and research has been very active in all aspects of the application ranging from basic physical phenomena understanding to development of efficient numerical algorithms and finally the development of a general purpose software which can solve almost all kind of problems.. In impact analysis generally two methods is widely used are implicit and explicit method. In recent years, the explicit dynamic finite element method has been widely used in the field of collision impact and has showed strong advantage in dealing with large scale contact problem. Nonlinear dynamic finite element analysis element system ANSYS-WORKBENCH-14 is used to analyse the characteristic of cylinder and plate impact to meet the requirement of safety and stability of cylinder.

II. FINITE ELEMENT MODELLING

For simulation FEM model is prepared exactly similar to the actual physical setup. Aluminum Plate and Copper coated Steel cylinder is prepared using FEM Modeling. In this Modeling distance between Al Plate and Cylinder are minimum for reducing the computation time**.** This is 2-D analysis and Quadrilateral element is used.

ISSN: 2350-0328

International Journal of Advanced Research in Science, Engineering and Technology

Vol. 4, Issue 4 , April 2017

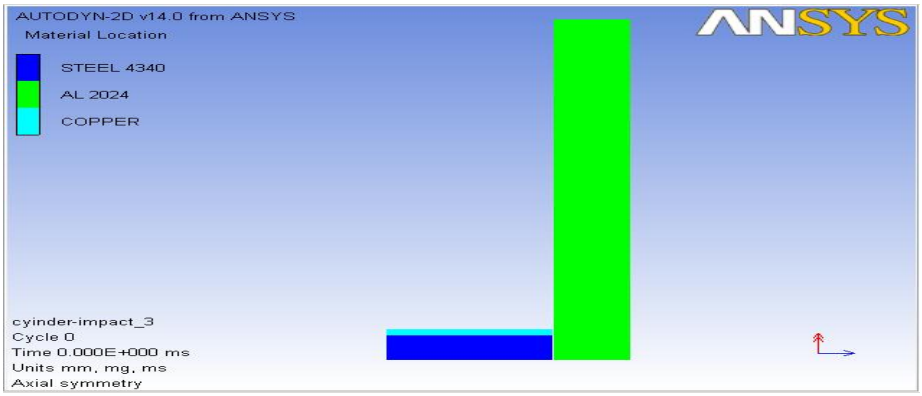


Fig.1 Model of Al Plate and Copper Coated Steel Cylinder

A.TEST SPECIMEN-ALUMINUM PLATE

The Aluminum 2024 Plate is used in this simulation. The material of plate has high yield stress and high stiffness value. The thickness of the plate is 6 mm. For Simulation, Al Plate is modeled, using 2-D quadrilateral element. In order to conduct a realistic simulation of the impact problem, the finite element mesh needs to be relatively dense. Hence the center part of plate is modeled with fine mesh.

B.CYLINDER

The Copper coated Steel 4340 cylinder is used in this analysis. For Simulation, Steel cylinder is modeled, using 2-D quadrilateral element. The diameter of the cylinder is 9mm.It shows the cylinder penetration at the initial stages of impact. Impact velocity of cylinder is varying 400 m/s, 500 m/s, 600 m/s & 700 m/s.

III. HIGHLIGHTS OF MESHING AND PRE –PROCESSING

To model the impact, penetration and deformation processes occurring when the cylinder impacts the target & the subsequent deformation of the plate, it is necessary to divide the plate and the projectile into a finite number of regions called elements. The network of the element called the mesh. The computations are then performed by solving the constitutive equations for the deformation of the individual elements in the mesh. In order to conduct a realistic simulation of the impact problem. The finite element mesh needs to be relatively dense in region that will experience high stress gradient and large deformations. This was done in order to minimize the computational time that could otherwise be very large. Model is prepared on ANSYS Workbench-14 and AUTODYN used as a solver in this impact problem. Impact velocity was given to Copper coated steel cylinder which is varying from 400 m/s to 700 m/s along X-direction.

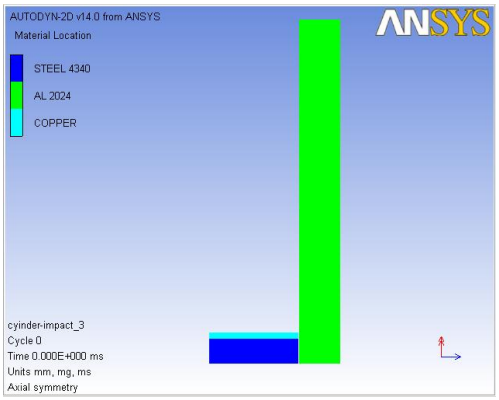
IV. RESULTS

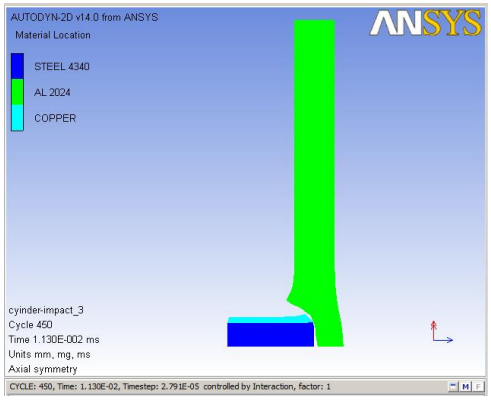
The model was analyzed in ANSYS Workbench for given loading and boundary conditions. The Copper coated Steel 4340 Cylinder was impacted with different incident velocities at plate which is made of Aluminum 2024 and the following results were obtained-

ISSN: 2350-0328

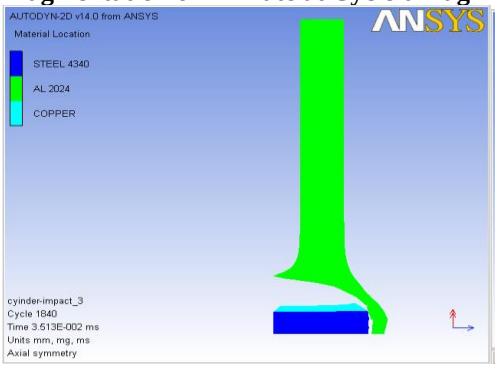
International Journal of Advanced Research in Science, Engineering and Technology

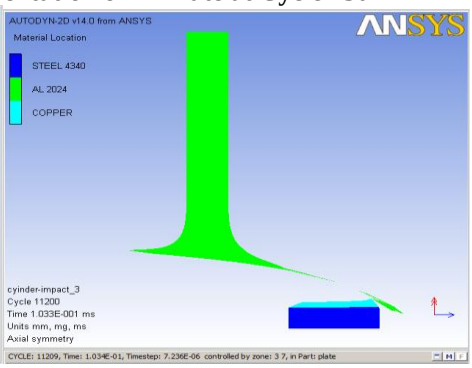
Vol. 4, Issue 4 , April 2017





Fragmentation of Al Plate at Cycle 0Fragmentation of Al Plate at Cycle 450





Fragmentation of Al Plate at Cycle 1840Fragmentation of Al Plate at Cycle 11200

Numerical simulation is taken into consideration in this analysis over experimental and mathematical simulation. In this Numerical analysis a linear equation is developed by which different residual velocities are obtained. The result shows the graph between Fragment velocity v/s times, Kinetic energy v/s Time. Table 1.1 shows four different cases of the fragment velocity (Incident velocity) and correspondent to Residual velocity.

Cases	Fragment Velocity	Residual Velocity
1.	400	350
2.	500	446
3.	600	543
4.	700	639

Table No. 1.1

The same analysis of impact is again performed and following data are obtained

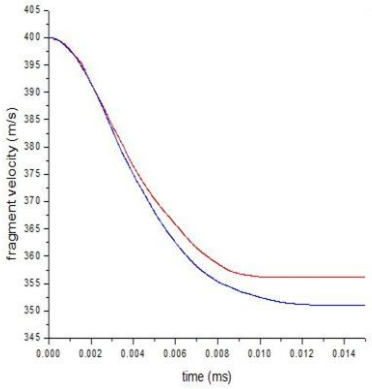
Cases	Fragment Velocity	Residual Velocity
1.	400	357
2.	500	448
3.	600	544
4.	700	642

Table No. 1.2

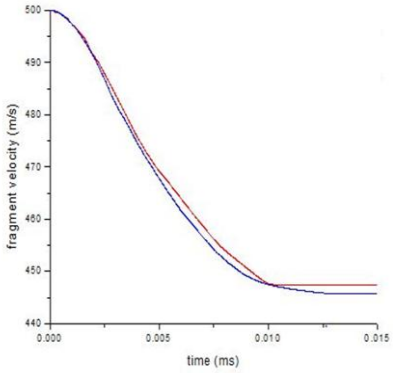
On the basis of table No.1.1 and 1.2 the graph between different fragment velocity and residual velocity are shown

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 4, Issue 4 , April 2017

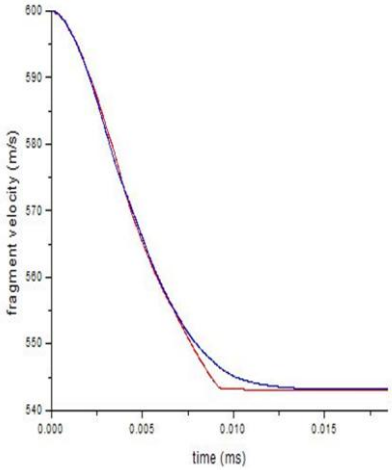
below –



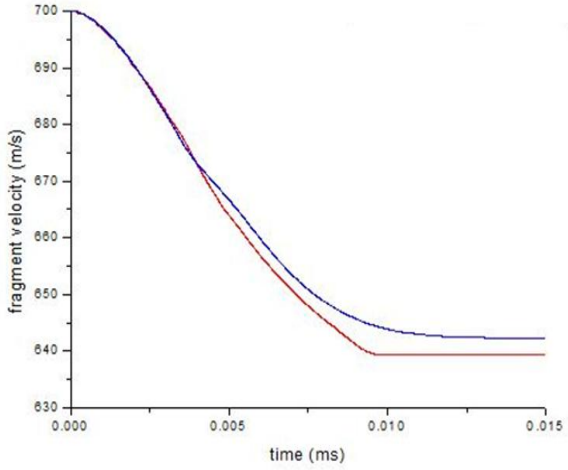
Comparison of fragment velocity impacted by projectile on a Steel 4340 Plate at 400m/s



Comparison of fragment velocity impacted by projectile on a Steel 4340 Plate at 500m/

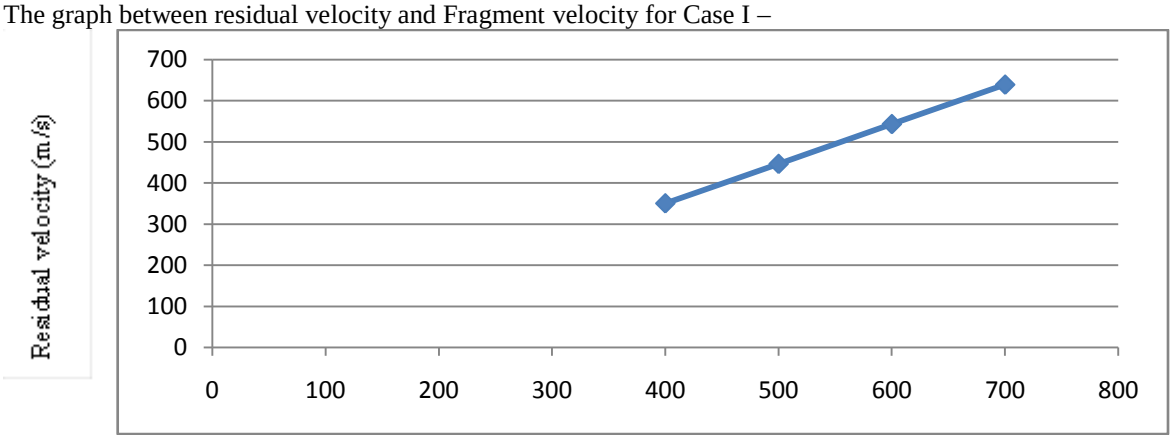


Comparison of fragment velocity impacted by projectile on a Steel 4340 Plate at 600m/**s**

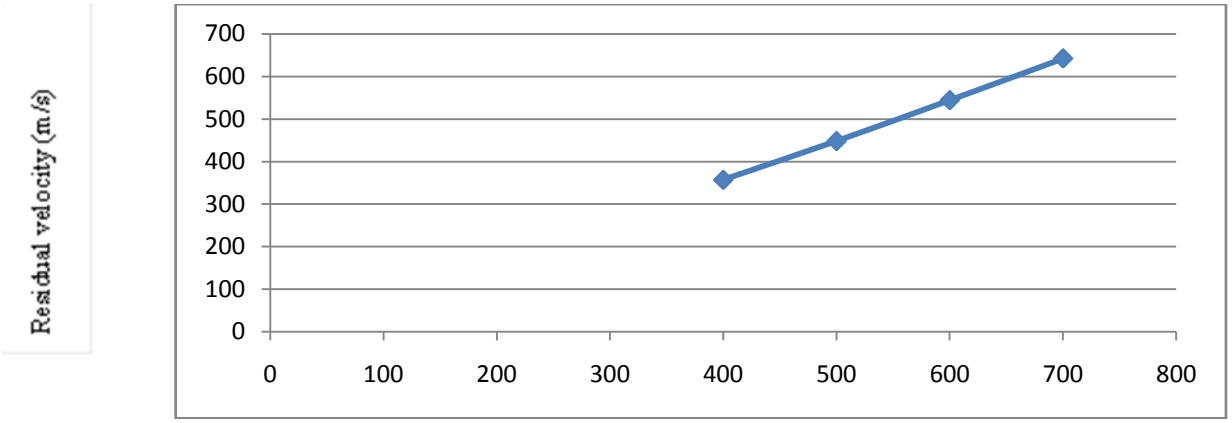


Comparison of fragment velocity impacted by projectile on a Steel 4340 Plate at 700m/**s**

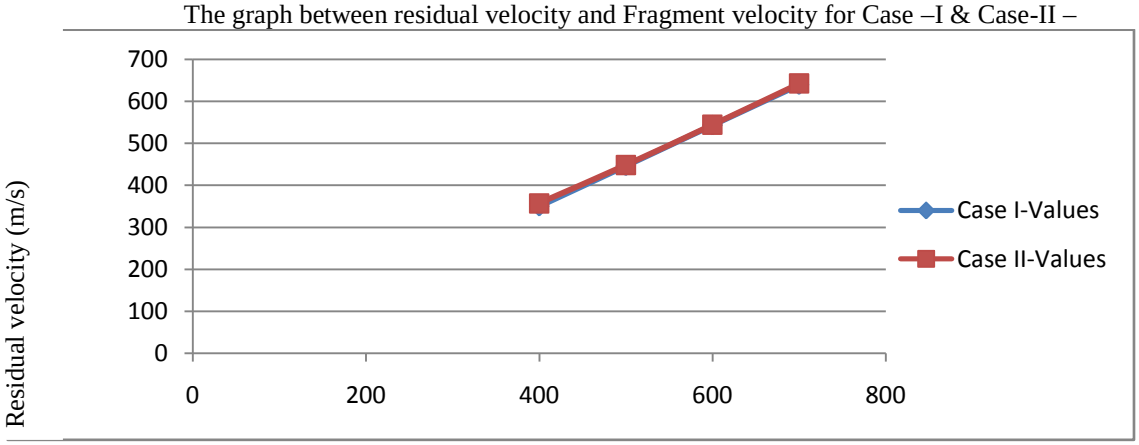
The graph between residual velocity and Fragment velocity for Case I –



ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 4, Issue 4 , April 2017

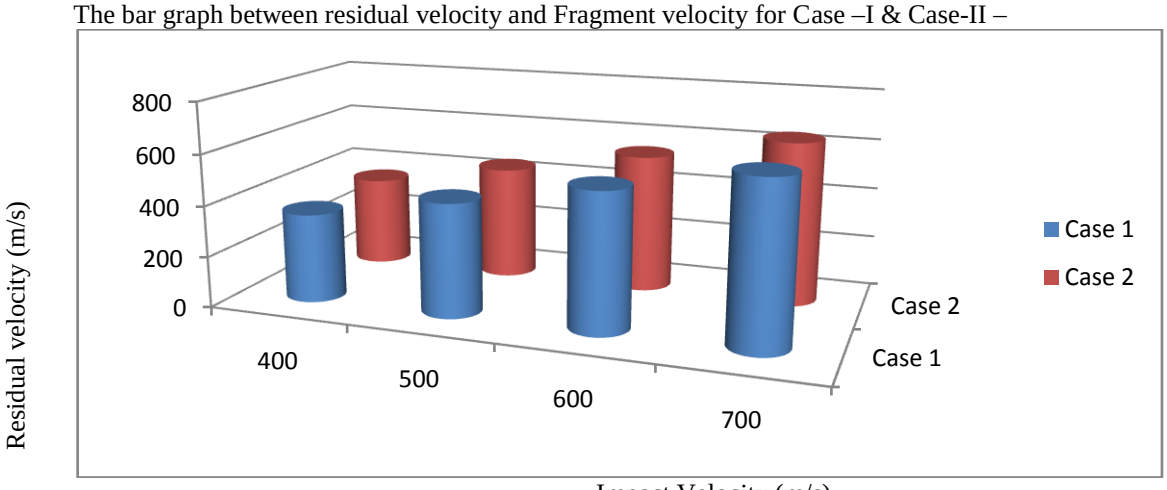


The graph between residual velocity and Fragment velocity for Case –I & Case-II –



Since graph of case I and case 2 are overlapped one over other. So this shows that the residual velocity obtained from both cases for same impact velocity are nearly same.

The bar graph between residual velocity and Fragment velocity for Case –I & Case-II –



From case 1 and case 2 the residual velocity which is obtained from different impact velocity is vary linearly, so on solving these point we find out a linear equation in the form of-

$Y = 0.96X - 34$

Where

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 4, Issue 4 , April 2017

Y: Residual Velocity X: Impact/Fragment Velocity
Whose slope m is 0.96 and constant c is -34

For Case 1

Table-1.3. Percentage Variation in Case I

S.NO	Fragment Velocity(m/sec)	Experimental Velocity(m/sec)	Numerical Velocity(m/sec)	% Variation
1.	400	350	350	0
2.	500	446	446	0
3.	600	543	542	0.18
4.	700	639	638	0.15

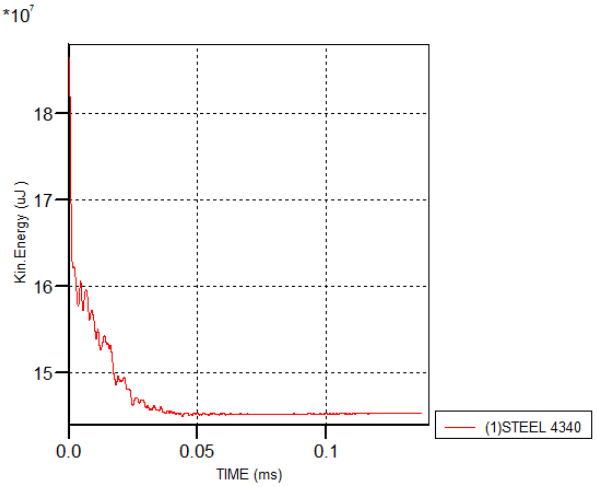
For Case 2

Table-1.4. Percentage Variation in Case II

S.NO	Fragment Velocity(m/sec)	Experimental Velocity(m/sec)	Numerical Velocity(m/sec)	% Variation
1.	400	357	350	1.9
2.	500	448	446	0.44
3.	600	544	542	0.36
4.	700	642	638	0.62

Since percentage variation in both cases is very small.So the numerical value obtained from the linear equation is shown in the table no. 4.3 and 4.4 for both cases and on compairing experimental velocity and numerical velocity the percentage variation which obtained is very less. So the linear equation which is formed from the experimental velocity is valid.

Graph between Kinetic Energy Vs Time for steel



V. CONCLUSION

In this study, it is aimed at developing the relation between Impact velocity and residual velocity and also determining the Impact behavior of copper coated steel cylinder and Aluminum Plate is performed by using ANSYS Workbench. A

ISSN: 2350-0328

International Journal of Advanced Research in Science, Engineering and Technology

Vol. 4, Issue 4 , April 2017

study was conducted on a Steel cylinder which is copper coated and Al Plate impacted at different velocities. A FEM model was modeled in ANSYS Workbench, boundary condition and loads were applied to it, key file was generated before submitted to ANSYS AUTODYN. The Output were viewed, required animation result and graph were plotted in AUTODYN. The study is conducted by varying the velocity of projectile cylinder and keeps the constant value of plate thickness. In these analysis four different cases is considered. After the analysis of impact at different velocity it generates the graphs between Impact Velocity v/s residual velocity, Impact velocity v/s Time and Kinetic Energy v/s Time. When the graph is plotted in between Impact Velocity and residual velocity a linear line is obtained so that a linear equation is developed. This equation is valid when same type of modeled is used for performing the Impact for the velocity likely to 400 m/s to 800 m/s. By the help of this equation we find the residual velocity.

VI. FUTURE SCOPE

From conclusion the study may be carried out on different type of coating used on projectile cylinder. Also Impact is performed at different angles to study the changes in behavior of material. In current study research study only normal incidence is considered. For getting much real world simulation the media of transition like fluid medium is implemented. Also thermal effects are to be considered to study the frictional heat generated and possible outcome in process. Impact Analysis at different materials is still a field of complete understanding. So there is a lot of analysis to be carried out to unveil the complex behavior of metal for applications in high speed and Impact events. The extensions of the present study can be listed as follows:

- Impact tests can be held by using the same materials and same configuration. The results of these tests can be used to validate the numerical simulation results.
- In the simulations only the copper coating is used in projectile. The copper coated Steel cylinder can also be modelled with some other materials and the simulations can be performed with this projectile.

Multiple tiles can be added to the simulation in order to investigate the interaction between these tiles. Also some rubber materials can be used between these tiles in order to absorb shocks coming from neighboring tiles.

REFERENCES

[1] SaiKiranChelluru, “Finite Element Simulation of Ballastic Impact on Metal Composites plates,” INDIA 2004.
[2]MYUNG,JONG CHULL JO & SANG JIN JEONG (Korea Institute of Nuclear Safety) Investigates the dynamic response characteristics of a structure impacted by a high Speed projctile, 2006
[3] WaseemSarwar and Nasir Hayat from (UET Lahore) studies “Crash Simulation and Analysis of a Car Body Using Ansys”,2007
[4]Afshari. A and Rajai, (University of Illinois, Chicago, USA) investigates the Finite Element Simulation of the role of Helmet in Reducing Head Injuries, 2008.
[5]Sanita ZIKE studies An Experimental and Numerical Study of Low Velocity Impact of Unsaturated polyester/Glass Fiber Composite, 2011
[6]Mi Yan,Wang Tao School of Automation (Beijing Institute of Technology) investigates “Finite Element Analysis of Cylinder Piston Impact Based on ANSYS,” 2012
[7]ZuzankaTrojanováinvestigates “High Strain Rate Behaviors AN AZ31 + 0.5 Ca Magnesium Alloy”, 2012.
[8]SumanKarinvestigates “Analysis of Ship Structures using Ansys.