

Finite Element Simulation of Ballistic Impact on a Thin Steel Plate

Rajesh P Nair¹, Alex A Earali², Igin John³, Sujith K M⁴

Assistant Professor, Dept. of Ship Technology, CUSAT, Kochi, Kerala, India¹

B. Tech Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India²

B. Tech Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India³

B. Tech Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India⁴

ABSTRACT: There is a large number of phenomenon that occur from the moment a projectile is fired to the moment the effects of its impact can be observed on the target. Providing proper shielding for vehicles and personnel from enemy fire is one of the primary concerns of any defence related project. The depth of penetration, residual velocity of projectile, and deformation pattern of the target are the other major parameters that are of concern in ballistic impact studies. This paper aims to minimize the cost and the time required in such studies using a series of ABAQUS/EXPLICIT numerical simulations for the study of the behaviour of both the projectile and target during both impact and penetration. The results are compared with ANSYS results which was available in literature.

KEYWORDS: Finite Element method, Ballistic mechanics, Depth of penetration

I. INTRODUCTION

Impact is a phenomenon that is associated with a sudden load of large magnitude applied over a short duration of time [1]. Finite Element Method (FEM) is a numerical method used to simulate the mechanical response of continuous media. The primary objectives of this paper is to simulate the impact of the projectile on a steel target using ABAQUS/Explicit and to predict the parameters like depth of penetration, Residual velocity of the Projectile, and the Deformation pattern in the Target.

II. GEOMETRIC MODELLING

ABAQUS/CAE modeling and simulation for Impact analysis consists of projectile and target part. The projectile is having cylindrical body with hemispherical nose shape. The diameter of the projectile is 7.62mm and the shank length is 13.81mm and the target is cylindrical plate with 10 mm thickness. For both projectile and target, quarter of the geometry is used for the simulation since both material and deformation pattern are axisymmetric in nature.

III. MATERIAL PROPERTIES

The correct definitions of the material properties are crucial to obtain reliable results out of the simulation. Both projectile and target are made of steel. During impact and penetration there will be a rise in temperature in both projectile and target. To capture these temperature effects, thermal properties are used in simulation of impact. Johnson Cook model is used for both material deformation and failure simulation. Johnson Cook model is a strain rate and temperature-dependent viscos-plastic material model. Failure model used is Johnson and Cook [2] failure criterion which is based on the continuum damage mechanics.

IV. DEFORMATION PATTERNS

Numerical simulation is carried out for velocities of 500m/s, 1000m/s and 1500m/s using ABAQUS. The comparison of deformation pattern both in projectile and target is compared with results available in Kurtaran et.al [3]. Kurtaran et.al simulated the impact analysis using ANSYS software. Both results show the similar pattern of deformation as shown in Figure 1.

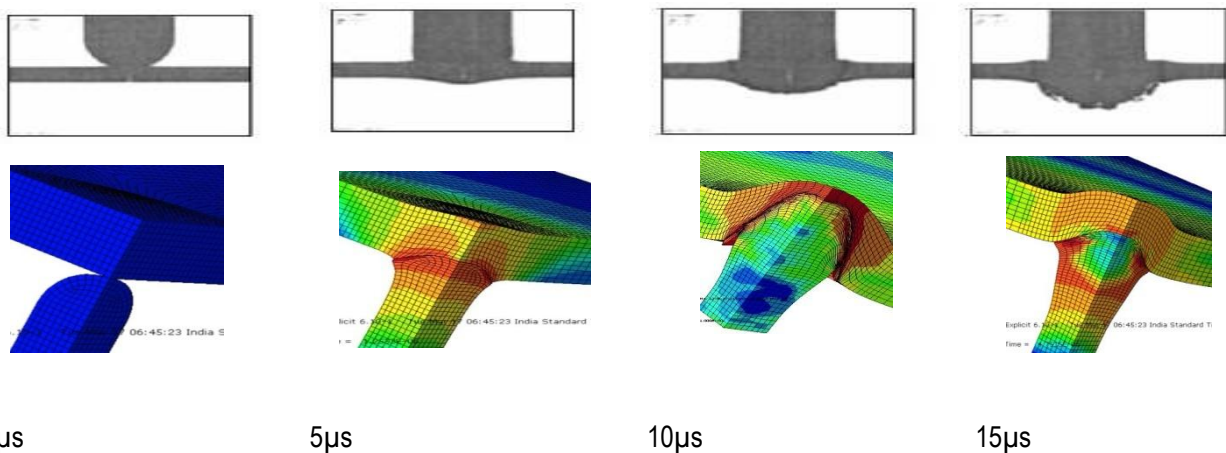


Fig.1: Comparison of Deformation Patterns with results available in literature [3]

Table 1 shows the Depth of Penetration (DOP) obtained for different impact velocities (V) and the results are also matching with ANSYS results.

Table 1: Comparison of depth of penetration with results available in literature [3]

Time (μs)	V=500m/s		V=1000m/s		V=1500m/s	
	DOP using ANSYS ^[3] (mm)	DOP using ABAQUS (mm)	DOP using ANSYS ^[3] (mm)	DOP using ABAQUS (mm)	DOP using ANSYS ^[3] (mm)	DOP using ABAQUS (mm)
1	0.4	0.5	0.9	1.03	1.7	1.5
5	1.5	1.49	3.6	3.5	5.4	5.38
10	3.5	3.6	7.8	7.62	12	11.8
15	5.4	5.3	12	11.7	19.5	18.8

V. SUMMARY

Numerical methods help in the optimization of bullet and target in ballistic impact simulation. ABAQUS software is used for simulation in this project. Modelling of geometry, assigning loads and meshing is easier in ABAQUS compared to ANSYS software. This is because in ABAQUS loads and boundary conditions are applied before meshing which is not the case with ANSYS software. Hence FEM meshing can be modified without affecting loads and boundary condition using ABAQUS software.

REFERENCES

- [1] Rajesh .P. Nair ,Impact analysis using Discrete Element Method, *Ph.D Thesis*, Department of Applied Mechanics, IIT Madras, October 2013.
- [2] G.R. Johnson, W.H. Cook, “Fracture characteristics of three metals subjected to various strains, strain rates, temperatures and pressures”, *Engineering Fracture Mechanics*, 21, 31-48, 1985.
- [3] H. Kurtaran, M. Buyuk, A. Eskandarian, “Ballistic impact simulation of GT model vehicle door using finite element method”, *Theoretical and Applied Fracture Mechanics*, 40, pp. 113–121, 2003.