NUMERICAL SIMULATION OF THE LOW VELOCITY IMPACT ON METAL PLATE WITH VARIOUS INFLUENCES PARAMETERS

[1]K.S.Prabakar, [2]E.Thiyagaragan, [3]M.Arun, [4]R.Sathish,
[1] Alpha College of Engineering,Chennai
[2]Hindustan University,Chennai
[3]Apollo Engineering College,Chennai
[4]Mechanical Department, Alpha College of Engineering,Chennai.
***Corresponding author: Email: prasrirad@gmail.com**

ABSTRACT

In this paper main focus on three dimensional finite element analysis of low velocity impact on metal plate to find influences parameter for the damage. The numerical simulation done in Abaqus software explicit solving scheme and metal failure criteria consider for the analysis is ductile damage. In simulation of low velocity impact body to be 150 M/S, 200 M/S and 250 M/S with various mass of the impacting body 50gms, 100gms, and 150gms. The shapes for the study are flat, sphere and sharp edged object. Finally to find the maximum damaged created by the impacting body shape, mass and impact velocity. The conduct force and velocity of impacting body are observed to find the above parameter.

Keywords: Impact; Ductile damage; Abaqus/Explicit; Damage Evolution

INTRODUCTION

The Steel is used in all industry application as one of the form of plate, In most of the industry there is lot quality and standard measure are used to avoid the damage during the impact where ever the human life are involved. It is important to study the response of materials to ballistic impact loads. Applications of this research include body armor, armored vehicles and fortified buildings, as well as the protection of essential equipment, such as the jet engines of an air liner. The impacts are may be in any foam like ballistic impact and runway debris, etc,

The impact object may be different shape, mass and impacting velocity and they are not properly quantified. There is a lot of experimental made for ballistic impact on metal plate and most of the papers are analyzed the velocity of impact body in this paper simulate with various velocity, mass and shape of impact body. In impact the damage of the object are influenced by velocity , mass and shape, in this above parameter shape of the impact body is included because, the while impact happen in between the two bodies the contact forces are developed is various with area of the contact. The numerical simulation was done using finite element method software ABAQUS.

NUMERICAL SIMULATION

Abaqus explicit: The abaqus explicit solving scheme is used for this problem. The explicit is capable of handling dynamic problem associated with metal damage. The explicit dynamics analysis procedure in Abaqus/Explicit is based upon the implementation of an explicit integration rule together with the use of diagonal or "lumped" element mass matrices. The equations of motion for the body are integrated using the explicit central difference integration rule.

$$\dot{\mathbf{u}}^{(i+\frac{1}{2})} = \dot{\mathbf{u}}^{(i-\frac{1}{2})} + \frac{\Delta t^{(i+1)} + \Delta t^{(i)}}{2}\ddot{\mathbf{u}}^{(i)},$$
$$\mathbf{u}^{(i+1)} = \mathbf{u}^{(i)} + \Delta t^{(i+1)}\dot{\mathbf{u}}^{(i+\frac{1}{2})},$$

where $\dot{\mathbf{u}}$ is velocity and $\ddot{\mathbf{u}}$ is acceleration. The superscript (i) Refers to the increment number and $i-\frac{1}{2}$ and $i+\frac{1}{2}$ refer to mid increment values. The central difference integration operator is explicit in that the kinematic state can be advanced using known values of $\dot{\mathbf{u}}^{(i-\frac{1}{2})}$ and $\ddot{\mathbf{u}}^{(i)}$ from the previous increment.

B, Numerical simulation setup

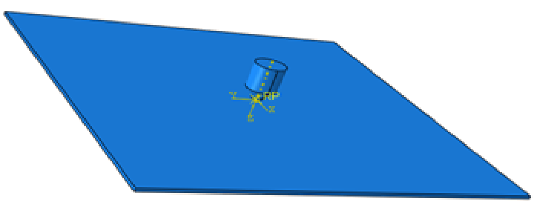

Fig. 1. Numerical setup

The Steel plate having 0.005 m thickness and square plate size of 0.05m is taken for the numerical simulation and the impacting object part consider having high stiffness of young's modulus is higher than the plate to make damage on the plate alone. The numerical simulation models both plate and the impact object are place to touch each other to avoid unwanted solving time of the impacting object travel. The contact in between the object is given by general contact method in Abaqus interaction module. The Abaqus general contact automatically generates the contact pair on each step time increment by means of searching the contact pair on each solution increments. The general contact algorithm in Abaqus/Explicit uses tracking algorithms to ensure that proper contact conditions are enforced efficiently. The interaction properties in between the impact object is taken co-efficient of friction value is 0.3 and thermal effect during the impact was neglected to avoid the complexity in the numerical simulation. The velocity and shape are directly varies with models, the mass of the impact body varied with volume kept it constant and change the density of the metal to meet the desire mass of the object 50 gms , 100gms and 150 gms.

TABLE 1: PARAMETERS -EACH IMPACTING BODY SHAPE

Si no	Mass in gms	Velocity in M/s
1	50	150
2	50	200
3	50	250
4	100	150
5	100	200
6	100	250
7	150	150
8	150	200
9	150	250

MATERIAL MODEL

Isotropic hardening: The plasticity hardening was Isotropic hardening used the yield surface changes size uniformly in all directions such that the yield stress increases (or decreases) in all stress directions as plastic straining occurs. Abaqus provides an isotropic hardening model, which is useful for cases involving gross plastic straining or in cases where the straining at each point is essentially in the same direction in strain space throughout the analysis. If isotropic hardening is defined, the yield stress, can be given as a tabular function of plastic strain and, if required, of temperature and/or other predefined field variables. The yield stress at a given state is simply interpolated from this table of data, and it remains constant for plastic strains exceeding the last value given as tabular data.

TABLE 2: MATERIAL PROPERTY OF THE PLATE

Elastic Property

Si No	Property	Value	Unit
1	Density	7800	Kg/m^3
2	Young's Modulus	207800000000	N/M^2
3	Poisson's ratio	0. 3	A NA

Plastic property

Yield Stress in N/M^2	Plastic Strain
1220000000	0
2440000000	1

Ductile Damage

Fracture strain	1
Stress Triaxiality	0
Strain Rate	0

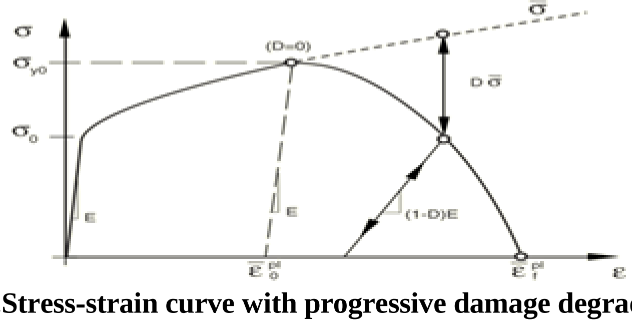

Fig.1.Stress-strain curve with progressive damage degradation

Ductile damage: The model assumes that the equivalent plastic strain at the onset of damage is a function of stress Triaxiality and strain rate.Two main mechanisms can cause the fracture of a ductile metal: ductile fracture due to the nucleation, growth, and coalescence of voids; and shear fracture due to shear band localization. Based on

Journal of Chemical and Pharmaceutical Sciences ISSN: 0974-2115
phenomenological observations, these two mechanisms call for different forms of the criteria for the onset of damage (*Hooputra et al., 2004*). The characteristic stress-strain behavior of a material undergoing damage. In the context of an elastic-plastic material with isotropic hardening, the damage manifests itself in two forms: softening of the yield stress and degradation of the elasticity. The solid curve in the figure represents the damaged stress-strain response, while the dashed curve is the response in the absence of damage. As discussed later, the damaged response depends on the element dimensions such that mesh dependency of the results is minimized.

In the figure $\sigma_{y0}$and $\bar{\varepsilon}_0^{pl}$ are the yield stress and equivalent plastic strain at the onset of damage, and $\bar{\varepsilon}_f^{pl}$ is the equivalent plastic strain at failure; that is, when the overall damage variable reaches the value $D = 1$. The overall damage variable, D, captures the combined effect of all active damage mechanisms and is computed in terms of the individual damage variables, d_i, The value of the equivalent plastic strain at failure, $\bar{\varepsilon}_f^{pl}$, depends on the characteristic length of the element and cannot be used as a material parameter for the specification of the damage evolution law.

Mesh: The element C383R element was taken for the plate and the element parameters are kept constant (30603 elements) for all load cases and for impact object kept constant for the shape (20000 Elements). The mesh adaptability is not included in the study due to solving time.

RESULT AND DISCUSSION

Simulation was made for combination of mass, velocity and shape of the impact body are totally 27 load cases are simulate The output from the numerical simulation velocity of the impact object and contact force generating are measured. From the observation sharp nosed body creates maximum damage on the plate and in feature study thermal effect and mesh adaptive was included in the numerical.

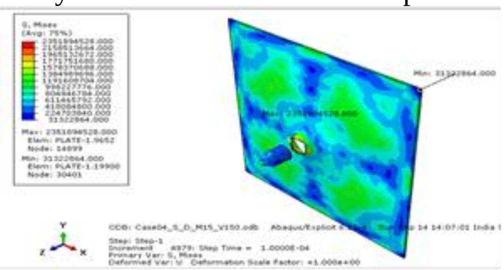

Fig.3.Stress plot for sharp nosed impact object at 100 gms and imapct velocity of 150m/s

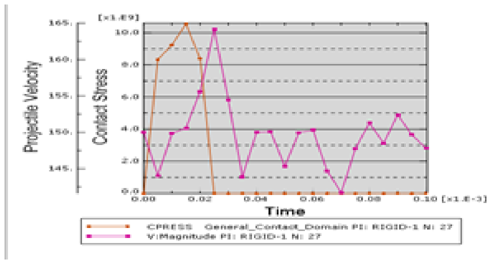

Fig.4.Conatct forces and Imapct body velocity graph for sharp nosed impact object at 100 gms and imapct velocity of 150m/s

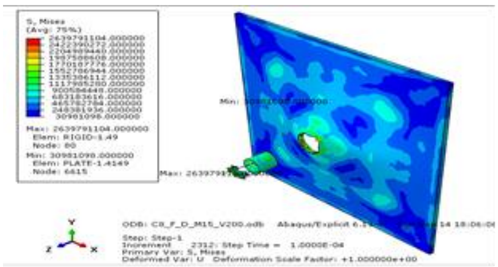

Fig.5.Stress plot for Flat nosed impact object at 100gms and imapct velocity of 200 m/s

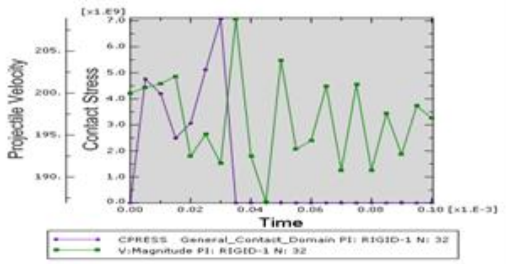

Fig.6.Conatct forces and Imapct body velocity graph for Flat nosed impact object at 100gms and imapct velocity of 200 m/s

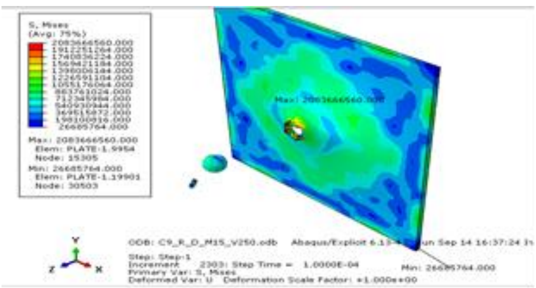

Fig.7.Stress plot for Round nosed impact object at 150gms and imapct velocity of 250 m/s

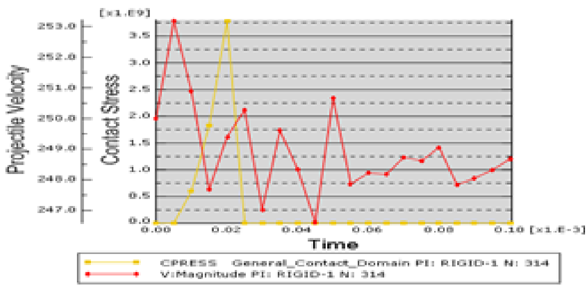

Fig.8.Conatct forces and Imapct body velocity graph for Round nosed impact object at 150gms and imapct

Journal of Chemical and Pharmaceutical Sciences ISSN: 0974-2115

velocity of 250 m/s

CONTACT FORCE

Mass	Flat nosed	Sphere Nosed	Sharp nosed
Velocity 150 m/s			
50	8289220000	2544600000	9102180000
100	9140470000	2609820000	10544500000
150	7196840000	2700250000	8902890000
Velocity 200 m/s			
50	6736640000	2669830000	5799670000
100	5410820000	4357670000	11441200000
150	4998460000	3788700000	7176370000
Velocity 250 m/s			
50	6736640000	2669830000	5799670000
100	5410820000	4357670000	11441200000
150	4998460000	3788700000	7176370000

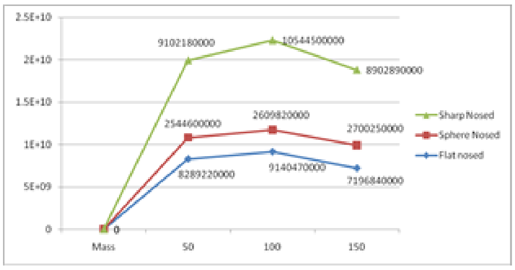

Fig.9.At velocity 150 Contact stress for impact bodies

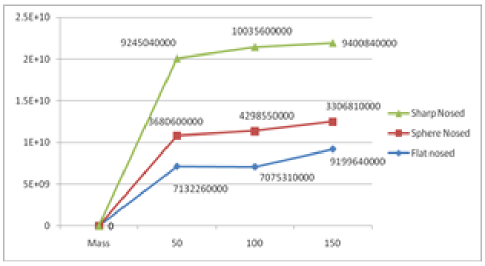

Fig.10.At velocity 200 Contact stress for impact bodies

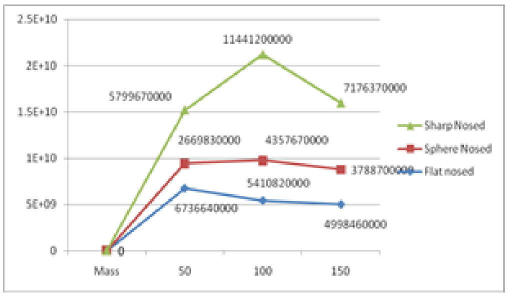

Fig.11.At velocity 250 Contact stress for impact bodies

REFERENCES

Charles E. Anderson Jr.and Sol R. Bodner, Ballistic impact: The status of analytical and numerical modeling" International Journal of Impact Engineering,Volume 7, Issue 1, 1988, Pages 9–35

Hooputra, H., H. Gese, H. Dell, and H. Werner, A Comprehensive Failure Model for Crashworthiness Simulation of Aluminium Extrusions, International Journal of Crashworthiness, vol. 9, no.5, pp. 449–464, 2004

S. Dey, T. Børvik, X. Teng, T. Wierzbicki and O.S. Hopperstad, On the ballistic resistance of double-layered steel plates: An experimental and numerical investigation, international Journal of Solids and Structures, 44 (2007) 6701–6723

S.N. Dikshit†, V.V. Kutumbarao, and G. Sundararajan, The influence of plate hardness on the ballistic penetration of thick steel plates, International Journal of Impact Engineering,Volume 16, Issue 2, April 1995, Pages 293–320

T Børvik, M Langseth, O.S Hopperstad, and K.A Malo, Perforation of 12 mm thick steel plates by 20 mm diameter projectiles with flat, hemispherical and conical noses: Part I: Experimental study, International Journal of Impact Engineering,Volume 27, Issue 1, January 2002, Pages 19–35

T. Børvik, M. Langseth, O.S. Hopperstad, and K.A. Malo, Ballistic penetration of steel plates, International Journal of Impact Engineering,Volume 22, Issues 9–10, October 1999, Pages 855–886

W. J.Stronge "Impact Mechanics" Cambridge university press