Thin-Walled Structures xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Thin-Walled Structures

journal homepage: www.elsevier.com/locate/tws

Full length article

Deformation behavior of single and multi-layered materials under impact loading

Asheesh Sharma, Rohan Mishra, Sanyam Jain, Srikant S. Padhee, Prabhat K. Agnihotri*

Department of Mechanical Engineering, Indian Institute of Technology Ropar, Rupnagar, 140001 Punjab, India

A R T I C L E I N F O

Keywords:
Abaqus explicit
Impact analysis
Critical failure velocity
Multilayer materials
VUMAT

A B S T R A C T

Finite element (FE) simulations using Abaqus/Explicit are performed to study the deformation behavior of materials under impact loading. Various configurations including monolithic and multi-layered plate having combinations of ceramic, metal and composite material layers are investigated to determine the critical failure velocity V_{cf} as a function of layer thickness and stacking. While cylindrical impactor is assumed to be rigid, Johnson-Cook (JC), Johnson-Holmquist (JH2) and Hashin 3D and Puck criteria is used to characterize damage/failure in metal (Al and steel), ceramic (SiC) and composite (carbon fiber/epoxy) layer respectively. Constitutive equations for composite material are supplied via user subroutine VUMAT. The results of FEM simulations reveal that Ceramic-Al-Carbon fiber/epoxy multilayer plate provides most desirable combination with higher critical failure velocity, lower average density, lower pressure and displacement at the back plate as compared to other material combinations considered in this work. Moreover, the analysis presented shows that the numerical approach developed can be used as a tool to predict the geometry and material combinations of a multilayer system to improve its resistance against impact loading.

1. Introduction

The study of interaction between two impacting bodies, known as impact dynamics or terminal ballistics has many crucial applications. Bullet impact on armor [1], occupant and pedestrian safety during automobile accidents [2], tool drop on aircraft wing [3], are few examples where impact dynamics plays an important role. An in-depth understanding of deformation behavior of materials under impact loading helps not only in designing better products but more importantly saving human life. Knowledge of material response under impact loading will help in estimating, enhancing and extending life and performance of any structure. Impact dynamics broadly depends upon two primary variables, namely the geometry and material of impacting bodies. Most often geometry of the impacting bodies gets dictated by the design parameters other than impact requirements: an aircraft wing has to be of an aero foil configuration, an armor has to be the shape of human body with limited thickness to enable mobility in combat zone, and a bullet has to be of conical shape for effective penetration and so on. Ultimately, it boils down to the response of material(s), which dictates the performance of the structure under impact loading. Various theories have been proposed to model the average behavior of metals [4,5], composites [6] and ceramics [7] under impact loading. Depending upon their nature, different materials have found their niche domain of application. While ceramic materials are most commonly used as front layer in armor for protection against bullet and shrapnel due to their high penetration resistance capability, epoxy and epoxy-based composites are ubiquitous in automobile industry for energy absorption due to high strain to failure and hyper-elastic nature [8]. Domain dependent application of materials is most common solution strategy for any design engineer and some kind of saturation in material choice has been reached recently in this regard [9]. Nonetheless it is well understood that these solutions are not extremal and further performance enhancement in the form of weight reduction or/and higher level of energy absorption is possible with intelligent design approach.

One of the possible ways in which the performance can be enhanced is to use a multi-material solution preferably through tuned geometry and stacking sequence. The basic idea is to enhance/amplify the performance of material(s) through structural tailoring. It is reported that the response of material to impact load changes with the change in target plate configuration [10] as well as shape of the impactor [11]. For example, the impact resistance of double layer steel plate found to be better in comparison to monolithic steel plate having same thickness [10]. Similarly, it is shown that by changing the ductility of steel in lower layer there is a considerable increased in ballistic resistance of double layered steel shields [12].

* Corresponding author.
E-mail address: prabhat@iitrpr.ac.in (P.K. Agnihotri).

http://dx.doi.org/10.1016/j.tws.2017.08.021
Received 29 December 2016; Received in revised form 15 August 2017; Accepted 16 August 2017
0263-8231/

Nomenclature

A	Yield stress
B	Hardening constant
C	Strain rate constant
n	Hardening exponent
ε_p	Effective plastic strain
To	Reference temperature
Tm	Melting temperature
P	Pressure
q	Mises stress
d_1,d_2,	failure parameters
σ_i^*	Normalized intact stress
σ_f^*	Fractured equivalent stress
$\dot{\varepsilon}$	Actual strain
$\Delta\varepsilon^p$	Equivalent plastic strain increment during a cycle of integration
ε_f^p	Plastic strain to fracture under constant pressure
L_c	Minimal distance between a point where d = 0 and a fully damaged point, d = 1
T	Tensile strength
K_1	Equation of state constant
K_2	Equation of state constant
K_3	Equation of state constant
Fs	Failure strain
β	Fraction of damage energy to convert to internal pressure
ρ_o	Initial density

Motivated by the above observations and to further explore the feasibility of multi-material solution to impact problems, Finite Element (FE) based simulation has been performed in this study. Simulation of a rigid cylindrical impactor on a deformable plate has been carried out in Abaqus/Explicit. In the first set of studies, single material is used for the plate to check the accuracy, convergence and meshing requirement for the simulation. In the second set of studies, multilayer material combinations are used through the thickness of the plate. In all the simulations, the plate thickness is kept constant while the impactor velocity gradually increased to record the minimum velocity at which impactor will penetrate through the plate made of a given material or materials combination.

2. Computational model

For the simulations, Finite Element models are developed in Abaqus/Explicit, incorporating various material and damage models. Computational cost and accuracy are two conflicting requirements in these type of simulations and to strike a balance between the two, special modeling techniques have been adopted which are explained in the relevant sections.

2.1. Geometry, boundary conditions and convergence

2.1.1. Geometry

The simulation involves impact of an impactor on a plate of chosen material at a certain velocity as shown in Fig. 1(a). The plate is 10 mm thick with rest of the two dimensions being 200 mm × 200 mm and entire boundary being constrained. The impactor is a rigid cylindrical rod of radius 5 mm with a weight of 28 g striking at the centre of the plate. As the entire system is symmetric about AA′ and BB′, only one quarter of the plate and striker is modeled to reduce the computational cost in FEM simulations as shown in Fig. 1(b).

2.1.2. Boundary conditions

Symmetric boundary conditions are imposed along the face OB and OA′.

- X-Symmetric boundary conditions are imposed on OB face as $U_1 = 0$
- Y-Symmetry boundary conditions are imposed on OA` face as $U_2 = 0$
- ENCASTRE boundary conditions are imposed for remaining sides of the plate as $U_1 = U_2 = U_3 = 0$

Since the zone of interest is near the impact point, the quarter plate is partitioned into two domains through parting line CDE. A finer mesh is used in the domain OCDE with relatively coarser mesh used in the rest of the domain (see Fig. 1(c)) to reduce the number of elements and hence computational cost of FEM simulations. Convergence study is carried out to determine the minimum element size.

2.1.3. Meshing

All the parts are meshed using C3D8R (Cubic, 8-node, 3D element with reduced hourglass control) elements, which are mostly used in impact studies [13], in all the FEM calculations. The size of the element is decided through convergence analysis of the single plate (aluminium) FEM model. To this end, the FEM simulations are performed with different mesh sizes and the residual K.E. is compared as a function of element size (see Fig. 2). It is observed from this analysis that the

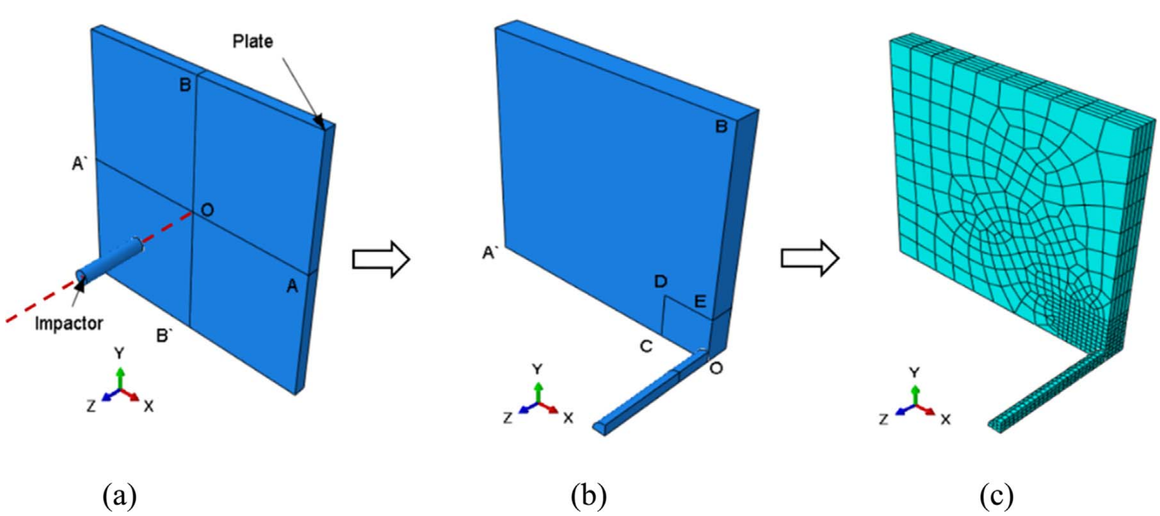

Fig. 1. (a) Full geometry (b) computational model after consideration of symmetry and (c) meshed geometry used in FEM simulations.