Finite element modelling of high velocity impact on plate structures

Milan Žmindák a,*, Zoran Pelagić a, Peter Pastorek a, Martin Močilan a, Martin Vybošťok a

a University of Žilina, Faculty of Mechanical Engineering, Department of Applied Mechanics, Univerzitná 1, 010 26 Žilina, Slovakia

Abstract

The high velocity impact in heterogeneous materials is a very old and complex problem. Accurate numerical simulation of impact events can provide physical insights that cannot be captured by experiments. The high velocity process is very complex and requires the use of reliable and robust constitutive materials models. Currently, for impact modelling in composite structures at low and high speeds is used mainly Finite Element Method (FEM), Boundary Element Method, Finite Volume Method, meshless formulations and recently connection of FEM and element free based formulations. In this paper the finite element models for normal impact were created in ABAQUS/Explicit software. First, the simulations presented here examine the perforation of steel and aluminium plate specimens and then impact load is applied on composite plate with eight layers reinforced by carbon fibers. The Johnson-Cook model for strain rate effects in the material and its compressibility and Hashin failure criterion for damage of composite plate were used in simulations.

© 2016 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).
Peer-review under responsibility of the organizing committee of MMS 2015

Keywords: composite structure; impact loading; Johnson-Cook model; Hashin failure criterion; ABAQUS/Explicit

1. Introduction

Impact loading and wave propagation in heterogeneous material is very old and complex problem [1, 2]. The phenomenon of material and geometric dispersion are so far very little studied. It is complex problem with regard to interaction of pressure and tension phase waves generated on the boundary of inhomogeneous material. The smaller the particles, the greater the number of material interfaces, which interact with each other and wave progresses, the greater the reduction and dispersion. Interaction of the bow shock wave with the secondary pressure waves resulting dissipation, shock-wave attenuation.

* Corresponding author. Tel.: +421 41 513 2962.
E-mail address: milan.zmindak@fstroy.uniza.sk
Currently, for wave propagation modeling in composite structures at high speeds is used mainly Finite Element Method (FEM) [3], Boundary Element Method (BEM), Fast Multipole BEM, respectively, Finite Volume Method (FVM), mesh free formulations [4] and recently connection of FEM and element free based formulations. However very effective methods appear are SFEM [5] and wavelet methods [6]. The commercial program systems LS-DYNA, AUTODYN and PAM CRASH etc. are used in practice.

Numerical simulation is used for completed resolution of the contact impact process. It is based on the theory of stress wave propagation, resulting in an impact of solid to structure composed form composite material and its material penetration. Impact modeling of solids at higher speeds is applied to solve tasks of penetration of solid by shell structure. Note that high speed considered in this paper are in the range to 100–1000 ms\(^{-1}\) and low speeds are the order of tens meters per second. The used material model plays an important role because used materials with the fundamentally different strength properties, which by their cohesion give the structure greater strength by higher resistance towards penetration of small element over this material. Materials properties must be described for different speeds of moving element through the structure. There are especially the modulus of elasticity, Poisson number, thermal expansion of material etc. to the finding these properties are using methods of numerical identification [7]. Often used material models in numerical model of solid penetration with structure are following: Mohr-Coulomb model, Johnson-Cook model, Zerilli-Armstrong model, Steinberg-Guinam model and thermo-mechanical material model.

Generally this paper will focus on the motivations and results, with only brief descriptions of the specific models.

2. Theory background and solution method

The finite element code ABAQUS/Explicit is particularly well-suited to simulate brief transient dynamic events such as drop testing, automotive crashworthiness, and ballistic impact. The governing finite element equations to be solved are:

\[
M\ddot{u} + C\dot{u} + Ku = F^{\text{ext}}
\]  
(1)

with initial conditions given by \(u(t = 0) = u_0, \dot{u}(t = 0) = \dot{u}_0\) and \(M, C\) and \(K\) are the global mass, damping and stiffness matrices, \(R\) is the vector of external loads. 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 [7]. The lumped mass matrix \(M\), allows the program to calculate the nodal accelerations easily at any given time \(t\):

\[
\ddot{u}(t) = M^{-1} \left[ F(t)^{(\text{ext})} - F(t)^{(\text{int})} \right]
\]  
(2)

where \(M\) is the within the explicit methods, as stated further, the time step needs to be small enough to propagate at a maximum speed at one element per time and \(F(t)^{(\text{ext})}\) is the vector of external forces and \(F(t)^{(\text{int})}\) is the vector of internal forces.

We note, that for fast dynamical problems such as wave propagations, where the propagation of the wave through the structure is analyzed, explicit methods are an ideal choice. The time incrementation scheme in Abaqus/Explicit is fully automatic and requires no user intervention.

Stability condition is given by terms of the highest eigenvalue in the system as (with no damping):

\[
\Delta t \leq \frac{2}{\omega_{\text{max}}}
\]  
(3)

where \(\omega_{\text{max}}\) is the maximum natural circular frequency. The calculation is based on Courant-Friedrichs-Lewy condition (CFL condition) for solving partial differential equations numerically by the method of finite differences [8]. Most FEM programs check size of all finite elements during time step calculation [9].
3. Description of problem and modelling approach

For the initial calculations, a geometrically simple case was chosen: a projectile impact on a steel and aluminum plate. The steel plate dimensions are $60 \times 60 \times 2$ (Fig. 1) and the composite plate has dimensions $120 \times 120 \times 2$ mm. Due to the symmetry, only quarter of the geometry was modelled to save the computational cost (Fig. 2). The composite plate is composed from the eight-node brick hexahedral elements with one integration point (C3D8R) and 50000 elements were used in the simulation. A refined, uniform mesh was used in the impact region indicated in the Fig. 3. FE mesh for the shell geometry was created using 5450 linear triangular elements S3R and for solid geometry has been used 39,520 linear SC6R brick elements.

The ABAQUS/Explicit simulations presented here examined the penetration of plate specimens. The projectile has cylindrical shape with semi-spherical fillet with a radius $R = 5$ mm. Since results from ballistic experiments showed negligible deformation, plastic deformation of the projectile is not considered. The projectile impacts the plate perpendicularly, right is center of the plate with a defined initial speed $v_i = 100$ ms$^{-1}$.

Fig. 1. Geometry used for composite plate and impactor. Fig. 2. Boundary conditions for composite plate and impactor.

Fig. 3. 3D FE mesh for plate.
3.1 Material models

Johnson-Cook (J-C) plastic material model was used to model the flow stress behavior of ductile materials. Material of steel plate is Steel 4340-C30 and material of aluminum plate is Aluminum 6061-T6. The hardening is a particular type of isotropic hardening in which von Mises $\sigma$ is expressed as function of the equivalent plastic strain $\varepsilon^{pl}$, equivalent plastic rate strain $\dot{\varepsilon}^{pl}$ and a dimensionless temperature $T^{*m}$ [10–12]:

$$\sigma = [A + B(\varepsilon^{pl})^n] [1 + C \ln \left(\frac{\varepsilon^{pl}}{\varepsilon_0}\right)] (1 - T^{*m})$$  \hspace{1cm} (4)

where $A,B,C,$ and $m$ are material parameters, $n$ is strain hardening exponent, $\dot{\varepsilon}^{pl}/\varepsilon_0$ is the normalized equivalent plastic strain rate (typically normalized to plastic strain rate of 1.0 s$^{-1}$), $T^{*m}$ is the homologous temperature defined as:

$$T = (T - T_{room}) / (T_{melt} - T_{room})$$  \hspace{1cm} (5)

where $T$ is the material temperature, $T_{melt}$ is the melting temperature, and $T_{room}$ is the room temperature. The equation for equivalent plastic deformation is given as:

$$\varepsilon^{pl} = \left[ d_1 + d_2 \exp \left( d_3 \frac{\rho}{\theta_{melt}} \right) \right] \left[ 1 + d_4 \ln \left( \frac{\dot{\varepsilon}^{pl}}{\dot{\varepsilon}_0} \right) \right] (1 - d_5)$$  \hspace{1cm} (6)

where $\rho$ is pressure and $d_1 - d_5$ are constants. In Table 1 are listed all parameters of J-C material model for steel and aluminium.

<table>
<thead>
<tr>
<th>Material</th>
<th>$A$</th>
<th>$B$</th>
<th>$n$</th>
<th>$\theta_{melt}$</th>
<th>$\theta_{transition}$</th>
<th>$m$</th>
<th>$c$</th>
<th>$\dot{\varepsilon}_0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel 4340-C30</td>
<td>792</td>
<td>510</td>
<td>0.26</td>
<td>1793</td>
<td>293.2</td>
<td>1.03</td>
<td>0.014</td>
<td>1.0</td>
</tr>
<tr>
<td>Aluminium 6061-T6</td>
<td>324.1</td>
<td>113.8</td>
<td>0.42</td>
<td>925</td>
<td>293.2</td>
<td>1.34</td>
<td>0.002</td>
<td>1.0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material</th>
<th>$\rho$</th>
<th>$d_1$</th>
<th>$d_2$</th>
<th>$d_3$</th>
<th>$d_4$</th>
<th>$d_5$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steel 4340-C30</td>
<td>830</td>
<td>0.05</td>
<td>3.44</td>
<td>2.12</td>
<td>0.002</td>
<td>0.61</td>
</tr>
<tr>
<td>Aluminium 6061-T6</td>
<td>703</td>
<td>-0.77</td>
<td>1.45</td>
<td>0.47</td>
<td>0.0</td>
<td>1.6</td>
</tr>
</tbody>
</table>

Impact test was simulated at a speed of 100 m s$^{-1}$ and the plate was supported on all edges. FE mesh of steel and aluminium plate was created from 50,000 linear volume elements. The ABAQUS/Explicit simulations presented here examine the penetration of aluminium plate samples impacted with steel rod with a hemispherical. In this simulations steel rod has penetrated fully through the structure. The maximum von Mises stress for aluminium plate reaches 451 MPa (Fig. 4) and for steel reaches 1600 MPa (Fig.5).
In the next simulations were considered composite plate with dimensions $120 \times 120 \times 2$ mm composed from eight layers. The material of present composite is an AS4/PEEK quasi-isotropic laminate. For the simulation of impact damage has been used four types of orientation layers (lay-up): $[0/0/0/0]_s$, $[0/0/90/90]_s$, $[45/45/45/45]_s$, $[90/0/0/0]_s$. The impact velocity was the same as for a metal sheet, $v = 100$ ms$^{-1}$. Material parameters of the laminate plate are listed in Table 3. Fig. 6 and Fig. 7 show the dependence of acceleration and velocity on time at the node where is maximum displacement. The maximum value of acceleration is $5.2e+08$ ms$^{-2}$ and maximum value of velocity is 139 ms$^{-1}$.

Table 3. Material properties of AS4/PEEK.

<table>
<thead>
<tr>
<th>$E_{11}$</th>
<th>$E_{22}$</th>
<th>$v_{12}$</th>
<th>$G_{12}$</th>
<th>$G_{13}$</th>
<th>$G_{23}$</th>
<th>$\rho$</th>
<th>$X_T$</th>
<th>$X_C$</th>
<th>$Y_T$</th>
<th>$Y_C$</th>
<th>$S_L$</th>
<th>$S_T$</th>
</tr>
</thead>
<tbody>
<tr>
<td>(GPa)</td>
<td>(GPa)</td>
<td>-</td>
<td>(GPa)</td>
<td>(GPa)</td>
<td>(GPa)</td>
<td>(MPa)</td>
<td>(MPa)</td>
<td>(MPa)</td>
<td>(MPa)</td>
<td>(MPa)</td>
<td>(MPa)</td>
<td>(MPa)</td>
</tr>
<tr>
<td>138</td>
<td>10.2</td>
<td>0.3</td>
<td>5.7</td>
<td>5.7</td>
<td>3.7</td>
<td>1570</td>
<td>2070</td>
<td>1360</td>
<td>86</td>
<td>230</td>
<td>186</td>
<td>86</td>
</tr>
</tbody>
</table>
4. Conclusion

In this paper for the analysis of laminate composite plates two models are used. The first is the solid based model and other is shell based model. There were also compared four different arrangement of the layers of the composite. As a criterion damage the composite plate was used Hashin damage model. The results obtained show that the von Mises stress have approximately the same value for all types of arrangements of the layers. For solid model, and also for the shell model was the largest von Mises stress in the arrangement of layers $[0/0/90/90]$, the lowest von Mises stress was in the arrangement of the layers $[90/0/0/0]$. For the arrangement of layers $[90/0/0/0]$, $[45/45/45/45]$, is the lowest von Mises stress for shell based model. The largest shear stresses were in the arrangement of the layers $[45/45/45/45]$, for solid as well as shell based model. The largest deformation was at the area of impact, which gradually spread to the depth of the material. From the results we can see that the orientation of the layers in the composite structures can have a significant effect on the behavior of the structure.

Acknowledgement

This work was supported by the Slovak Research and Development Agency under the contract No. APVV-0736-12 and by the Slovak Grant Agency VEGA 1/0983/15.
References