BALLISTIC IMPACT OF COMPOSITE LAMINATES: EXPERIMENTS AND SIMULATIONS

Felicity J. Guild, Mohamed El-Habti and Paul J. Hogg
Queen Mary, University of London, Mile End Road, London E1 4NS, UK

Keywords: *Ballistic Impact, Finite Element Analysis, Composite laminates*

Abstract

The results presented in this paper are from experimental observations and numerical simulations of ballistic impact on a composite panel. Penetration into ceramic/composite targets has been of interest for researchers in the engineering and military sectors since the sixties when it was first realized that ceramic tiles backed by glass-reinforced plastic provide good lightweight amour systems. The recent advances in this field show a considerable improvement in amour capability.

The overall aim of this project is to optimize the design of ceramic/composite armour using finite element simulations. This paper covers results from composite panels. Excellent agreement is found between experimental observations and numerical simulations of ballistic impact on the panels. This comparison of experimental and predictive data is an essential step in verification of the finite element modeling.

1 Introduction

Traditional armoured vehicles have been protected by steel-based armour. As weapon designers were constantly trying to improve the terminal effect of their projectiles the armour become thicker and heavier. That led to the evolution of ground fighting vehicles of 70+ tonnes [1]. These increase in armour weight reduced the mobility of it users. This led to the conclusion that the traditional armour materials have reach the limit of their usefulness.

Penetration into ceramic/composite targets has been of interest for researchers in the engineering and military sectors since the sixties when it was first realized that ceramic tiles backed by glass-reinforced plastic provide good lightweight amour systems. The recent advances in this field show a considerable improvement in amour capability.

Because of the complicity of the problem up to recently most of the work carried in this field was experimental in nature. However, the expenses of putting together a prototype and then testing it under a variety of conditions similar to that of real applications can be very costly. An effective way of carrying engineering analysis is to use a computer. A wide variety of design options can be tested and the final design could be chosen even before the first prototype is built. Due to the recent advances in computer simulation, the field of numerical analysis has become even more attractive.

The overall aim of this project is to optimize the design of ceramic/composite armour using finite element simulations. The results presented in this paper are from experimental observations and numerical simulations of ballistic impact on composite panels. This comparison of experimental and predictive data is an essential step in verification of the finite element modeling.

2 Experimental work

The aim of these ballistic trials is to determine the behavior of composite plates impacted with two different ball bearings, 0.44gm and 0.88 gm mass, at Normal impact (0° NATO).

2.1 Materials

The composite plates were constructed at Queen Mary using plain weave E-Glass infused using Vinyl ester resin. The plate thickness varied between 1.2 and 4.1 mm, leading to values of areal density between 2.5 and 7.4 Kg m^{-2}.

2.2 Ballistic Impact Testing

High-velocity impact tests on the composite plates were carried out using a gas gun. The actual impact took place in the experimental chamber. The samples were mounted in the test framework. The velocities were varied by altering the pressure from the helium gas bottle. The chronograph was positioned close to the target to measure the velocity of the target immediately before the impact. The projectile material was hard enough to behave elastically during impact, so that damage is only located in the composite. Tests were carried out using both the 0.44 and 0.88 gm ball bearings, on a range of plate thickness. The test arrangement is shown in Figure 1.

Fig. 1 Ballistic Impact Test on Composite Plate

2.3 Determination of Ballistic Limit.

The V_{50} ballistic limit velocity for a material is defined as the velocity for which the probability of penetration of the chosen projectile is 50%. This value is found by performing six ballistic impact tests within a velocity range of 10% for which three shots perforate the plate and three did not. The line of best fit is plotted and the V_{50} value is then obtained.

3 Finite Element Analysis

The numerical study was carried out using a three-dimensional analysis code, MSC Dytran. This program is an explicit finite element code dedicated to analyze dynamic problems associated with large deformation. The software has the capability to remove the failed elements from the mesh; this element removal was used in this analysis to allow the projectile to penetrate the plate.

3.1 Components of the model

Finite element models for the projectile and the plate were built using MSC Patran. The composite plate is modeled using 8-nodes 3D orthotropic Lagrangian solid elements with single point integration. The projectile is modeled using rigid 4-nodes shell elements. The computation time was reduced by modeling only one quarter of the problem utilizing symmetry. The impact area is drawn using a fine mesh; the element size is increased gradually for the remaining plate. A typical mesh for a 2 mm thick plate contained around 36,000 elements.

3.2 Laminate failure

During a ballistic event, penetration, fiber failure, matrix failure and delamination should be taken into account as they are considered the principle damage mechanisms in a ballistically impacted laminate. These damage modes are accumulative and are usually dependent on the impact condition. The description of failure models is based on Hashin failure criteria for a unidirectional layer and been generalized by Yen [2] to characterize the damage in terms of strain components for the plain weave layer. In this study this failure criteria has been defined and implemented using Fortran subroutines.

3.3 Delamination

Delamination is simulated using discrete interfaces inserted between the laminate ply mesh.

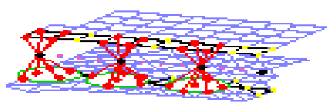

Fig. 2. Spotwelds used to Model Delamination

By this method, nodes on opposite sides of an interface where delamination is expected are tied together using spring elements. If the constraint forces exceed some criterion, the constraint is released and the delamination grows. Spotweld/spring elements are used to hold together two sub-laminates, as shown diagrammatically in Figure 2. The analysis calculates the forces generated in the spring elements as it ties two nodes together, and predicts failure if the magnitude of the total force or its components exceeds specified strength values.

4. Results

4.1 Experimental Results

The V_{50} ballistic limit velocities for the composite plates were determined; the results are shown in Figure 3; the results are plotted as a function of areal density As expected, the ballistic limit increases with increasing areal density. Increasing the mass of the ball bearing reduces the ballistic limit. However, this reduction is small since the surface area of contact has increased.

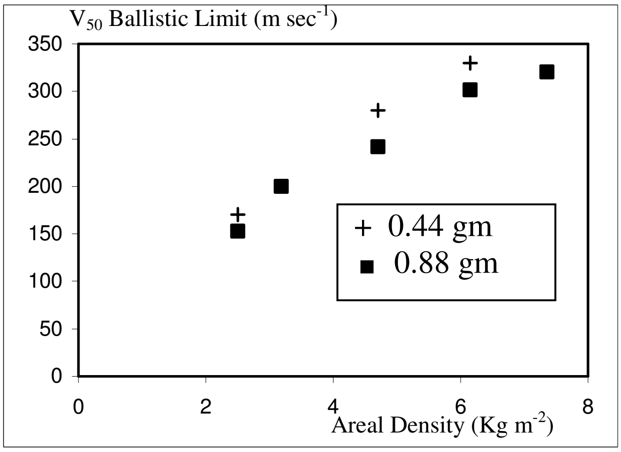

Fig. 3. Variation of Measured Ballistic Limit

4.2 Damage Region

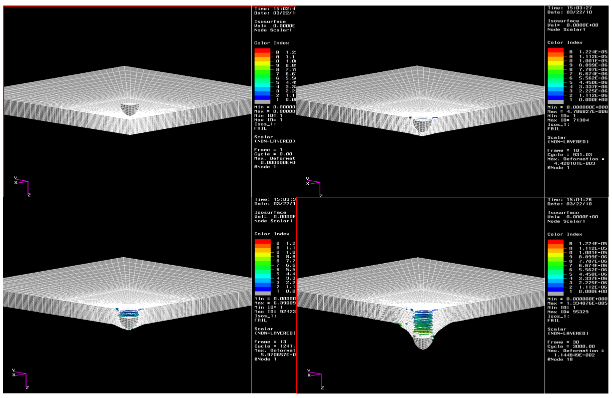

Fig. 4 Impact at High Velocity

Results from the finite element simulations are shown in Figure 4. These results are from a 4.2 mm thick composite plate impacted at 900 m sec^{-1} with a 0.88gm ball bearing. This impact velocity is far higher than the ballistic limit. The damage and delamination area is small and local.

Further results from the simulations are shown in Figure 5. These results are from the same composite plate impacted with the same ball bearing at 300 m sec^{-1}. It is clear that the ball bearing has not penetrated the plate, but the impact has caused a larger damage area including delamination.

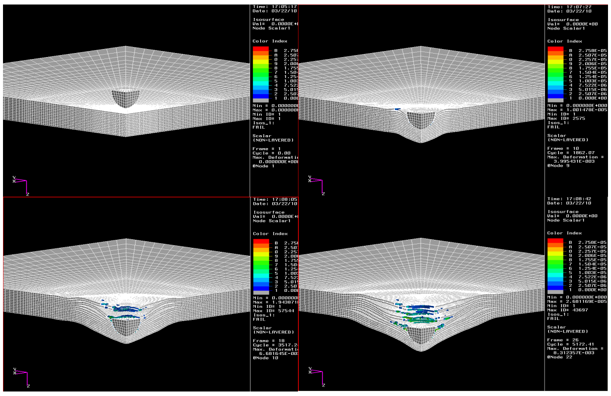

Fig. 5. Impact at Low Velocity

The proportion of energy absorbed by the different failure processes is shown in Figure 6. These are approximate results only, from a high velocity ballistic impact on an 8 mm thick composite plate. It is clear that most energy is dissipated from fibre failure.

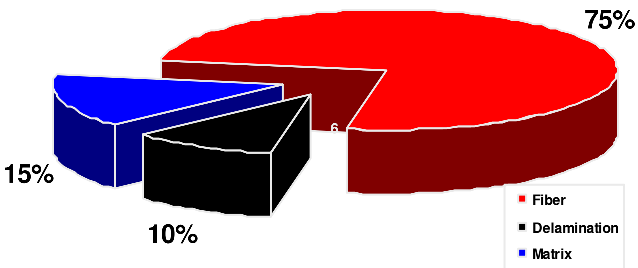

Fig. 6 Proportion of Energy absorbed by the Different Failure Mechanisms

The damage region predicted by the simulations and observed experimentally are compared in Figure 7. These results are from a 10 mm thick plate impacted by a 2.8 gm projectile. Excellent agreement is found both for the deformation and the extent of the damage.

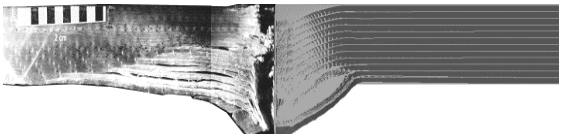

Fig. 7. Comparison of Experimental (left) and Numerical (right) results

4.3 Ballistic Limit Velocity

The values of V_{50} ballistic limit for impact on composite panels using the two sizes of ball bearing have been measured experimentally (see section 5.1). The ballistic limit was also determined using the finite element simulations. Successive simulations were repeated to find the limit velocity when penetration occurred.

The experimental and numerical results for the 0.44 gm ball bearing are compared in Figure 8; the results for the 0.88 gm ball bearing are compared in Figure 9.

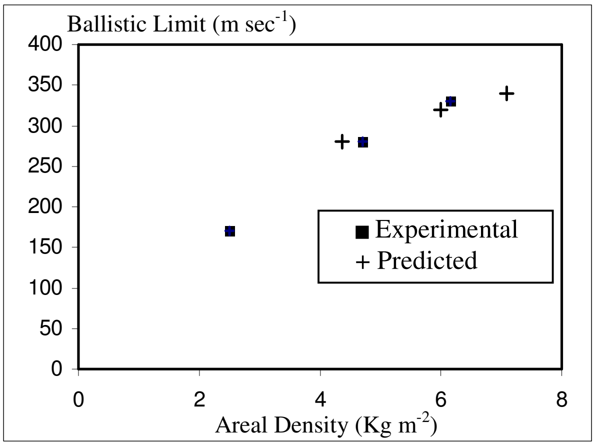

Fig. 8. Comparison of Experimental and Predicted Values of Ballistic Limit for 0.44 gm Ball Bearing

Excellent agreement is found between experimental and predicted results for both masses of ball bearing. Taking into account the expected experimental errors, the results can be described as in complete agreement.

These results verify our finite element approach. The methods will now be applied to ceramic/composite armour. The design of such armour, particularly in respect of the relative thickness of the materials, will be derived.

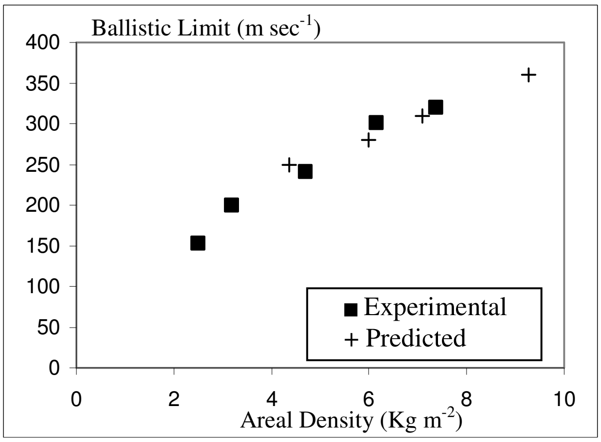

Fig. 9. Comparison of Experimental and Predicted Values of Ballistic Limit for 0.88 gm Ball Bearing

5 Concluding Remarks

Experimental measurements of ballistic impact on composite panels for a wide range of areal weight have been carried out using two different masses of ball bearing. Similar impact events have been simulated using finite element analysis. Very good agreement has been found between the experimental and numerical results for both observations of damage and deformation. Further, values of measured ballistic limit are in very good agreement with the values gained from the simulations. This correlation forms a verification of our finite element simulations. These simulations can now be extended to optimize the design of ceramic/composite armour. Finite element simulation has been shown to be a very powerful technique to predict the behaviour of composite panels under ballistic limit. These methods could be extended to a wide range of composite materials.

6 References

[1] Fink K. “Performance metrics for composite integral armour” *Journal of Thermoplastic composite material*, Vol. 13, PP 417-431, 2000

[2] Yen C. “Ballistic impact modeling of composite materials” *7th International LS-DYNA User Conference*, Dearborn, Michigan, May 2002.

Acknowledgements

The authors would like to thank Fuyou Zhu, Queen Mary, University of London, for his help with the experiments.