

A General Optimization Methodology for Ballistic Panel Design

K. Krishnan¹
Phoenix, AZ 85040

Dr. S.D. Rajan²
Tempe, AZ 85287

Dr. A.D. Belegundu³
University Park, PA 16802

Ballistic panel designs pose several interesting challenges. Of these, three are quite important. First, there is the challenge to develop appropriate material model that can not only capture the constitutive model (relationship between changes in stresses to changes in strains) but also needs to include appropriate failure criteria. Second, one must develop a finite element model that yields an acceptable accuracy obtained in a timely fashion. Last, a general optimization methodology is needed to yield the optimal solution to design problem. The design problem may be sizing, shape, topology optimization or may even involve material tailoring. In this research, we illustrate how laboratory testing as well as virtual testing have been used to develop the constitutive model including failure characterization. This is followed by the development of a custom yet general object-oriented design optimization program that couples a suite of optimizers with the commercial code, LS-DYNA. The suite of optimizers includes both gradient-based and global optimization techniques. Two case studies involving ballistic panel designs are presented. The panels are made of metal and thermo-pressed composites. The product design challenges include appropriate constitutive model development, development of an efficient but accurate finite element (FE) model and translating the final design into manufacturable product. In each case, the optimized designs are verified experimentally. The results from the case studies show that efficient designs can be obtained from this developed methodology.

Nomenclature

\mathbf{x}	=	Design variables vector
\mathbf{x}^L	=	Lower limit on design variables
\mathbf{x}^U	=	Upper limit on design variables
N_{dv}	=	Number of design variables
N_n	=	Total number of nodes in the FE model
N_e	=	Total number of elements in the FE model
KE	=	Kinetic energy
ϵ_{max}	=	Max plastic strain for the material
M	=	Total mass of the structure
t	=	Thickness of the structure (plate) at any point
t_{min}	=	Minimum thickness allowed
d_z	=	Displacement in the z-direction (or, the measure of Back Face Signature)

Keywords: Shape optimization, ballistic panel, high-speed projectiles, explicit finite element analysis, constitutive modeling.

¹ Design Engineer, Armor Designs Inc., Phoenix, AZ.

² Professor, Dept. of Civil & Environmental Engineering, Arizona State University, s.rajan@asu.edu , 480-965-1712 (contact author). R&D Manager, Hawthorne & York, Intl, Phoenix, AZ.

³ Professor of Mechanical Engineering, Dept. of Mechanical and Nuclear Engineering, Pennsylvania State University, 330 Leonhard Bldg. adb3@psu.edu , 814-863-2115, Associate Fellow, AIAA.

I. Introduction

This paper presents a methodology to optimize the shape of ballistic panels – armor panels that are subjected to high-speed projectiles. A large body of research is available on modeling ballistic plates using a variety of basic commercially available materials such as Dyneema, Spectra, Kevlar, Twaron etc. Jacobs et. al.¹ evaluate the ballistic performance of Dyneema based armor materials against deformable bullets and fragment simulating projectiles (FSPs). They compare the strength, modulus and rupture values of different high strength, high modulus fibres. Utomo et. al.² also use Dyneema fiber, and analyze Dyneema composites by different impact experiments to characterize the fracture phenomena as well as find the material properties. They characterize the delamination pattern of the composite as a function of impact velocity. Cheesman and Bogetti³ present a review of the factors that influence ballistic performance such as the material properties, fabric structure, projectile geometry, impact velocity, boundary conditions etc. Peijs et. al.⁴ study the effects of strain rate and temperature on modulus, strength and fracture of high-performance polyethylene (HP-PE) fibres and composites. Adams⁵ describes the ballistic as well as the material characterization experiments to develop a material model to be used in ballistic simulations. The technical report also provides insight on the physical, mechanical and thermal properties of the projectile materials. Kurtaran et. al.⁶ use the 3-D nonlinear finite element code LS-DYNA to study the ballistic impact of a bullet on the GT model vehicle door. The finite element simulations are conducted using plastic kinematic and Johnson-Cook material models. Shi and co-workers⁷ study the application of thin shell elements and solid shell elements in the modeling of panel-like targets in finite element perforation simulations using LS-DYNA. Buyuk et. al.⁸ discuss the various mesh algorithms like Lagrangian, Eulerian, ALE (Arbitrary Lagrangian-Eulerian) and SPH (Smooth Particle Hydrodynamics) and compare their performance for modeling ballistic impact problems. The paper discusses the significance of meshless methods (or particle methods) in modeling high energy impacts and penetrations that involves large deformations and erosions. Schwer⁹ presents a very interesting introduction to the alternative forms of penetration analysis such as Lagrangian, Eulerian and Smooth Particle Hydrodynamics (SPH) methods. Yen¹⁰ discusses the development of a computational constitutive model to characterize the progressive failure of composite laminates under high velocity ballistic impact conditions. The composite failure model is implemented within LS-DYNA as a material subroutine (MAT 161).

Much superior structural shapes can be designed with optimization methodology compared to ad-hoc techniques. This is substantiated by a considerable amount of trial-and-test runs some of which are presented here. In this paper, a general design optimization methodology is created for the design of ballistic panels. The methodology is general enough to be applicable to several classes of design problems involving ballistic events. The numerical examples deal specifically with meeting NIJ Level III standards¹¹; however, the methodology can be applied for a much wider class of problems. The procedure involves integration of a numerical optimization technique with LS-DYNA simulation code. In Section II, we discuss the design optimization problem. LS-DYNA finite element modeling of ballistic impact is discussed in Section III. Sections IV-VI detail the shape optimization methodology. Numerical results are presented in Section VI and conclusions in Section VI.

II. Problem Definition

The schematic for the basic design problem is shown in Fig. 1. A ballistic panel is to be designed that will contain all the projectiles that it is subjected to. Depending on the standards that need to be satisfied, there may be several additional performance requirements that need to be met as we will see later on.

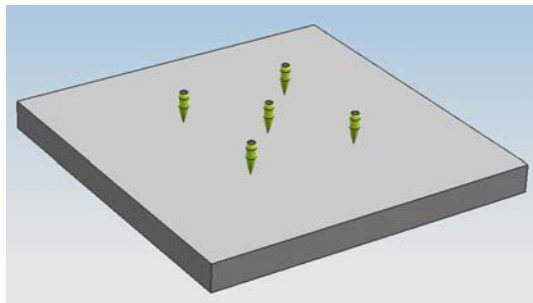


Figure 1. Schematic diagram showing ballistic panel subject to multiple shots

Ballistic panel design problems can be formulated a number of ways. For example, one can formulate the design problem for a metallic panel (e.g. aluminum) as follows.

Problem P1Find \mathbf{x} : the optimal layout and shape of the plateTo minimize $f(\mathbf{x}) = M$, the weightSubject to $\epsilon_{\max} \leq \epsilon_{\max}^a$ (1)

$$t \leq t_{\max}$$

$$\mathbf{x}^L \leq \mathbf{x} \leq \mathbf{x}^U$$

Problem P2Find \mathbf{x} : the optimal layout and shape of the plateTo minimize, $f(\mathbf{x}) = t$, the thicknessSubject to $\epsilon_{\max} \leq \epsilon_{\max}^a$ (2)

$$\mathbf{x}^L \leq \mathbf{x} \leq \mathbf{x}^U$$

The term *layout* refers to material distribution. One can obtain solutions to both problems P1 and P2 provided the objective and constraint functions can be evaluated accurately. As we will see in the next section, when damage and erosion (element deletion) are involved in the finite element analysis, tracking a response quantity as the plastic strain does not lead to (mesh) converging solutions.

An alternate formulation is to pose the design problem as follows.

Problem P3Find \mathbf{x} : the optimal layout and shape of the plateTo minimize $f(\mathbf{x})$, the weightSubject to $KE \leq KE_{\text{target}}$ (3)

$$\mathbf{x}^L \leq \mathbf{x} \leq \mathbf{x}^U$$

where KE is the kinetic energy of the projectile and one can take KE_{target} as a small fraction (1-5%) of the initial kinetic energy. Our numerical experience with ballistic simulations has shown that a global measure such as the kinetic energy of an identifiable part of the FE model exhibits convergence as opposed to a pointwise (or, element related) property such as the plastic strain.

III. LS-DYNA Modeling Considerations

To understand the design problem formulation especially with regards to the accuracy of the finite element analysis, a prototype design problem was created. Problem P3 was used in the design of a uniformly thick aluminum plate. The finite element model has two parts – the bullet and the aluminum (Al 6061-T6) plate.

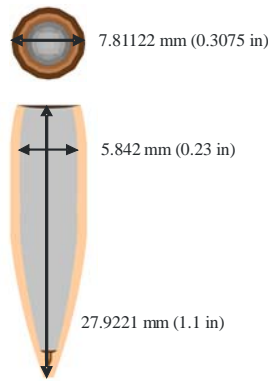


Figure 2. Top view and cross-sectional dimensions of the bullet (NATO 7.62 x 51 mm M80 Ball, or 0.308 Winchester bullet)

The bullet is comprised of a core of pure lead surrounded by copper jacket and the dimensions are shown in Figure 2. There is also a negligible amount of soft lead filler between the core and jacket which is not modeled. Both these materials are modeled as an elastic-plastic material with kinematic hardening. Applicable material values are listed in Tables 1 and 2. As shown in Figure 3, the aluminum panel has the dimension of 0.152 m x 0.1524 m (6.0 in x 6.0 in) with initial thickness of 0.02032 m (0.8 in).

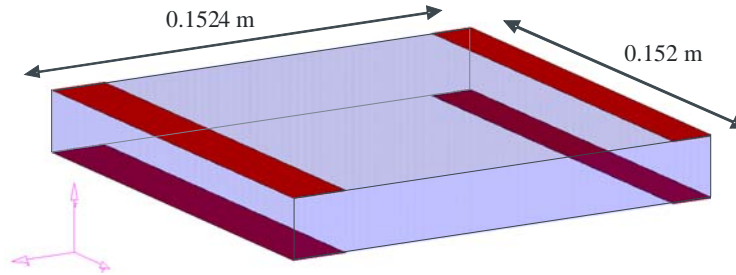


Figure 3. Panel dimensions and boundary conditions (nodes fixed in the red shaded area)

Table 1. Plastic Kinematic Constants for Lead Core¹²

Density (kg/m ³)	Young's Modulus (GPa)	Poisson's Ratio	Yield Stress (MPa)	Shear Modulus (GPa)	Tangent Modulus (MPa)	C (/s)	p	Beta
11,270	17.0	0.4	8.0	4.7	15	600	3	0.2

Table 2. Plastic Kinematic Constants for Copper Jacket¹³

Density (kg/m ³)	Young's Modulus (GPa)	Poisson's Ratio	Yield Stress (MPa)	Shear Modulus (GPa)
8,860	115	0.307	76	44

The Johnson-Cook material model is used to represent the Al 6061-T6 panel, as it captures the strain rate and temperature effects much better the elastic-plastic model. The relevant constants are listed in Table 3.

Table 3. Johnson-Cook constants for Al 6061-T6¹⁴

A (MPa)	B (MPa)	C	n	m	D_1	D_2	D_3	D_4	D_5
324	114	0.002	0.42	1.34	0.071	1.248	-1.142	0.147	0.0

The failure strain (erosion criteria) of the bullet jacket and core are 1.0 and 2.5, and 5.0 for the aluminum panel. When element strain reaches its erosion criteria, the element is deleted or eroded from the model. Since the mesh density can significantly affect the results of analysis, a convergence study was performed. Figure 4 shows the kinetic energy of the bullet core for the 7 different mesh densities. Mesh density ranges from 'coarse' for mesh 1 to 'fine' for mesh 7. The graph shows that the coarser meshes 1-4 actually arrest the bullet (physically incorrect as verified through experiments) while the finer meshes 5-7 allow for bullet penetration through the panel. Mesh 5 is the coarsest mesh to follow the trend line that best represents the physics of the problem.

The LS-DYNA analysis times⁴ for the different mesh densities are shown in Table 4. The mesh sizes characterized by number of elements and nodes are in Table 5. Increasing the mesh density has a significant impact on the analysis time. Mesh 5 takes 993 seconds for a single analysis.

⁴ Dell Precision WS390, Intel Dual Core 2-2.66 GHz, 3 GB RAM, MS Windows XP SP 2

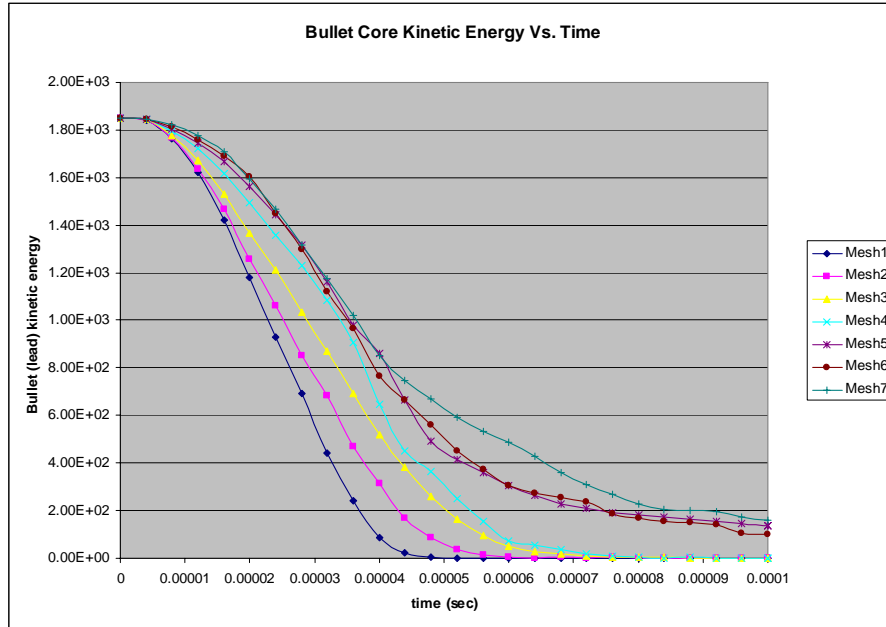


Figure 4. Kinetic energy of the bullet core as a function of mesh density

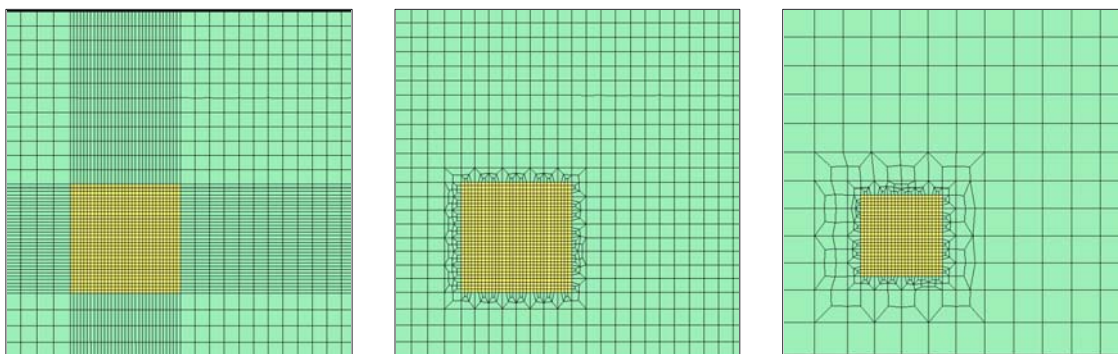
Table 4. Single analysis times as a function of mesh density

	<i>Mesh1</i>	<i>Mesh2</i>	<i>Mesh3</i>	<i>Mesh4</i>	<i>Mesh5</i>	<i>Mesh6</i>	<i>Mesh7</i>
<i>Single Analysis Time (sec)</i>	16	23	60	362	993	2125	7420

Table 5. Number of elements and nodes in meshes

	<i>Mesh1</i>	<i>Mesh2</i>	<i>Mesh3</i>	<i>Mesh4</i>	<i>Mesh5</i>	<i>Mesh6</i>	<i>Mesh7</i>
<i>Num. of elements</i>	7,632	9,248	11,456	19,136	43,712	69,568	152,000
<i>Num. of nodes</i>	8,705	10,667	13,205	21,737	48,397	75,809	132,121

To reduce the time, transitional mesh area were added around the point of impact and the outer elements were made uniformly coarse. The time can be further reduced by reducing the bullet impact area and making the outer elements even larger. Figure 5 shows the original mesh and the modifications made to reduce the analysis time – 993 s, 688 s and 433 s.



(a) Original mesh

(b) Transition 1

(c) Transition 2

Figure 5. Mesh refinement technique adopted to reduce the simulation time.

IV. Design Optimization

A FORTRAN computer program has been developed to implement the above procedures. Implementation details are available in a companion paper¹⁵. The shape optimization version of the program has been used in this study. Coordinate files are written for every population generated and are used checking the distortion of the mesh. A high function value is returned for a distorted mesh; otherwise the LS-DYNA solver is invoked. LS-DYNA writes nodal and element related time history outputs to ASCII files called ‘nodout’ and ‘elout’ respectively. FORTRAN routines are written to open these files and calculate the function values. The suite of numerical optimization techniques includes Method of Feasible Directions (MFD), Genetic Algorithm (GA) and Differential Evolution (DE). In this paper only MFD is used in the numerical design optimization problems.

V. Numerical Results

We present results from two design optimization problems – design of a flat aluminum plate (as discussed in Section III) and design of a composite plate to meet NIJ Level III standards.

Design Optimization of Aluminum Ballistic Plate

The shape optimization was carried out to obtain the optimal design. Problem formulation P3 was used with KE_{target} as 5% of the initial kinetic energy of the bullet (or, 4.85 J). Shape optimization was carried out using the Natural Shape Optimization Approach^{16,17}. The generated velocity field essentially changes the thickness of the plate. The Stage 1 design (shape) optimization yielded a design shown in Figure 6. The initial design was infeasible.

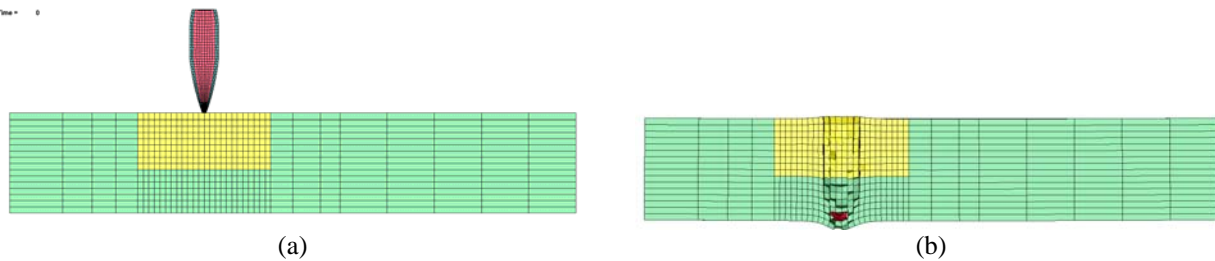


Figure 6. Optimized Design (a) Initial FE model (before impact) (b) Final FE model

Table 6. Stage 1 Optimization Results

Number of Iterations	Maximum KE of Bullet (J)	Maximum Plastic Strain	Initial Thickness (in)	Final Thickness (in)
20	0.324	0.38	0.8	1.05

As a final step (Stage 2), optimization was carried out starting with the resulting thickness of stage 1 but with a finer mesh – the number of elements through the thickness is increased from 16 to 18. The differences in the two stages are illustrated in Table 7 with the final (optimal) design shown in Fig. 7. The final thickness decreased slightly.

Table 7. Comparison of Stage 1 and 2 Optimization Results

	Total Number of Elements	Number of Iterations	Maximum KE of Bullet (J)	Maximum Plastic Strain	Initial thickness (in)	Final thickness (in)
Stage 1	21,664	20	0.324	0.38	0.8	1.05
Stage 2	23,516	18	0.815	0.4	1.05	1.04

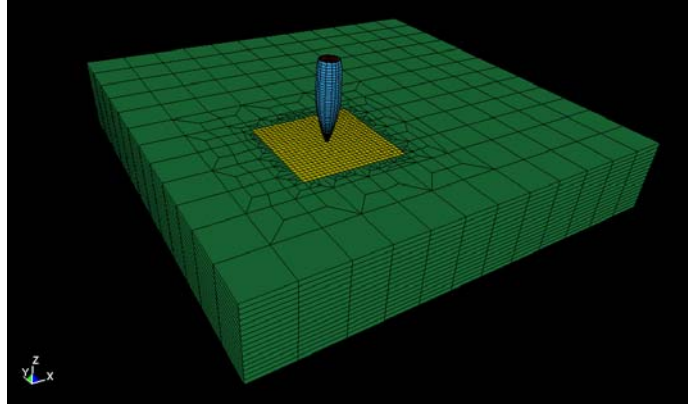


Figure 7. Stage 2 - Final optimized aluminum panel

Design Optimization of Composite Plate to Meet NIJ Level III Standards

The composite plate is primarily made of thermo-pressed high modulus polyethylene fiber (HPPE). A modified form of Problem Formulation P3 was used as follows.

$$\begin{aligned}
 &\text{Find} && \mathbf{x} : \text{the optimal layout and shape of the plate} \\
 &\text{To} && \text{minimize } f(\mathbf{x}), \text{ the weight} \\
 &\text{Subject to} && KE \leq KE_{\text{target}} \\
 & && d_z \leq 44 \text{ mm} \\
 & && \mathbf{x}^L \leq \mathbf{x} \leq \mathbf{x}^U
 \end{aligned} \tag{4}$$

where the additional constraint is to limit the displacement at the back face or the non-strike face of the plate (also called the Back Face Signature or BFS requirement). The plate geometry is shown in Fig. 8. The bullet model was identical to the one used with the aluminum plate design. Only one shot was modeled using the rationale that the first shot is the most crucial shot. This rationale was backed up by several experimental results.

The HPPE material was modeled using Chang-Chang composite failure model. The material parameters used in the model were obtained through a combination of laboratory tests and parameter fit using experimental data from ballistic tests of actual plates.

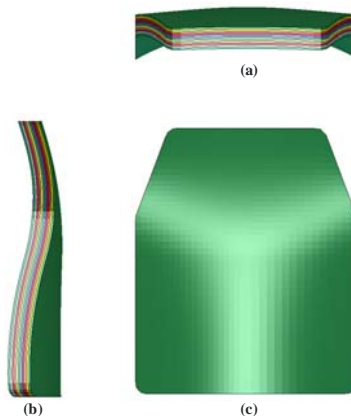
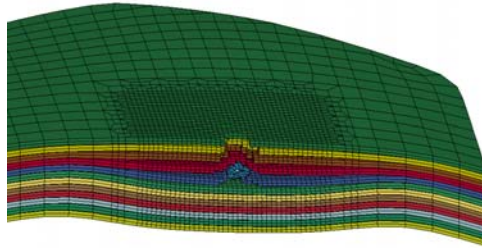


Figure 8. (a) Top (b) side and (c) front perspective view of the ballistic plate

During the first stage optimization, the velocity field was created to essentially change the thickness of the plate. While the design shape changes made it possible to find the smallest thickness to arrest the bullet, the BFS requirement could not be met until the plate thickness was well over an inch. The cross-section of the optimized HPPE plate is shown in Fig. 9(a) while Fig. 9(b) shows the cross-section through the plate where the first shot was fired.



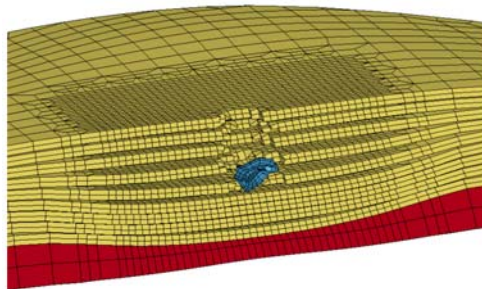
(a) FE Simulation



(b) Actual Panel

Figure 9. Optimized HPPE Ballistic Plate

To take care of the BFS requirement with a smaller thickness plate, a material change was made in the back face of the plate – a layer of foam material was added below the HPPE layer with the intent of attenuating the shock waves that cause the large displacements on the back face. The foam material was modeled as low density foam. Once again the design (shape) optimization runs were made to yield the correct thickness of HPPE keeping the foam thickness constant. The optimized result is shown in Fig.10, satisfies all the NIJ requirements. With the foam-backing, the BFS value decreases by about 40% compared to the HPPE-only plate.



(a) FE Simulation



(b) Actual Panel

Figure 10. Optimized foam-backed HPPE Plate

VI. Concluding Remarks

A general methodology for design optimization of ballistic panels is presented. The general framework makes it possible to carry out shape and layout optimization while preserving the accuracy of the finite element analysis predictions. The accuracy of the constitutive model dictates the accuracy of the obtained results. The finite element mesh not only controls the accuracy of the results but also dictates how long the design optimization process takes. Our experience (comparing experimental and simulation results) has shown that with proper adjustment of the model parameters, it is possible to obtain very accurate FE results in a reasonable amount of time. Future challenges include decreasing the wall clock simulation time and finding innovative materials (or tailoring material behavior) to design thinner and lighter panels while meeting various threat levels.

Acknowledgments

The authors would like to thank James St. Ville, President, Armor Designs Inc., Phoenix for partially funding the research work. Special thanks to Saurabh Bansal for conducting lab experiments to determine material properties, for manufacturing the plates for NIJ Tests, and for analyzing the test results for feedback in improving the material model. Thanks also to Sohui Yu in building and running the finite element and design optimization simulations.

References

-
- ¹ Jacobs, M. J. N. and Van Dingenen, J. L. J., "Ballistic Protection Mechanisms in Personal Armour." *Journal of Materials Science*, 36, 3137-3142, 2001.
- ² Heru Utomo, B. D., Van der Meer, B. J., Ernst, L. J. and Rixen, D. J., "High Speed Fracture Phenomena in Dyneema Composite." *Key Engineering Materials*, 353-358, 120-125, 2007.
- ³ Cheeseman, B. A. and T. A. Bogetti, "Ballistic Impact into Fabric and Compliant Composite Laminates", *Composite Structures*, 61, 161-173, 2003.
- ⁴ Peijs, T, Smets, E. A. M., and Govaert, L. E., "Strain Rate and Temperature Effects on Energy Absorption of Polyethylene Fibres and Composites", *Applied Composite Materials*, 1, 35-54, 1994.
- ⁵ Adams. B., "Simulation of Ballistic Impacts on Armored Civil Vehicles", Technical Report, Eindhoven University of Technology, undated.
- ⁶ Kurtaran, H, Buyuk, M. and Eskandarian, A., "Ballistic Impact Simulation of GT Model Vehicle Door using Finite Element Method", *Theoretical and Applied Fracture Mechanics*, 40, 113-121, 2003.
- ⁷ Shi, G., Guo, J. and C. Lu, "Efficient Modeling of Panel Like Targets in Perforation Simulation", *Institute of High Performance Computing*, Singapore, undated.
- ⁸ Buyuk, M, Kan, C. D. S., Bedewi, N. E., Durmus, A. and S. Ulku, "Moving Beyond the Finite Elements, a Comparison Between the Finite Element Methods and Meshless Methods for a Ballistic Impact Simulation ", *8th International LS-DYNA Users Conference*, 2004.
- ⁹ Schwer, L. E., "Preliminary Assessment of Non-Lagrangian Methods for Penetration Simulation", *8th International LS-DYNA Users Conference*, 2004.
- ¹⁰ Yen, C. F., "Ballistic Impact Modeling of Composite Materials", *7th International LS-DYNA Users Conference*, 2002.
- ¹¹ National Institute of Justice, Ballistic Resistance of Personal Body Armor NIJ Standard-0101.04, September 2000.
- ¹² Barauskas, R. and A. Abraitene, "Computational Analysis of Impact of a Bullet against the Multilayer Fabrics in LS-DYNA." *International Journal of Impact Engineering*, 34, 1286-1305, 2007.
- ¹³ <http://www.matweb.com>
- ¹⁴ Lesuer, D. R., G. J. Kay and M. M. LeBlanc, "Modeling Large-Strain High-Rate Deformation in Metals." *Presented at the Third Biennial Tri-Laboratory Engineering Conference Modeling and Simulation*, Pleasanton, CA, November 3-5, 1999.
- ¹⁵ Belegundu, A. D. and Rajan, S. D., "Shape Optimization of Metal Plates Subject to Blast Pressure Loading", *Engineering Optimization 2008 Conference*, Rio De Janeiro, Brazil, June 2008.
- ¹⁶ Belegundu, A.D. and Rajan, S.D., "A Shape Optimization Approach Based on Natural Design Variables and Shape Functions", *Computer Methods in Applied Mechanics and Engineering*, 66, 87-106, 1988.
- ¹⁷ Rajan, S.D., Chin, S-W and Gani, L., "Towards a Practical Design Optimization Tool", *Microcomputers in Civil Engineering*, 11, 259-274, 1996.