RESEARCH ARTICLE OPEN ACCESS

Experimental and numerical evaluation of plasticity model with ductile damage to applied in project of shielding sheet metal.

Bruno Cesar Pockszevnicki*, Pedro Américo Almeida Magalhães Júnior**
*(Mechanical Engineering Department, Pontifical Catholic University of MinasGerais,BeloHorizonte, Brazil)
**(Mechanical Engineering Department, Pontifical Catholic University of MinasGerais, Belo Horizonte, Brazil)

ABSTRACT
This work aims to develop a methodology for numerical evaluation via finite elements applied to projects shields sheet metal. To validate the methodology physical test were conduct and were compare with the numerical simulation. In the simulation, a plasticity material modelit was used at high strain rates, besides the insertion of a damage model through forming limit diagram (FLD) to capture the initiation of damage and energy criteria for propagation of the fracture. The tested shielding design is for the II-A protection level using the type 9mm ammunition.
Keywords- Ballistic Impact, Finite Element, Plasticity, Damage, Blindage.

I. INTRODUCTION

In blindage projects for security and defense area, different materials are employ to form the solution of the shield. The understanding what happens with each material inserted in solution is necessary and desired. One part of the development process more usedto evaluate project proposals are experimental verifications.

Instead of only physical test, the generation of numerical models using the finite element method can be a great strategy to reduce time and cost, optimizing resources and ensuring structural performance, and enable increased understanding of the ballistic impact phenomenon.

In the case of shields project the main phenomenon to be study is structural impact with perforation. To study perforation, a damage variable to capture the penetration should be considered in the model as the other numerous parameters and mechanical properties that arerequired according [1] when they usefinite element technique applied to modeling shield composite.

Therefore, phenomenon and characterization of materials are the variables to be study in understanding the art of shielding. To start in the area using the finite element method, one methodology is present in this paper aimed at a portion of the armored project in which only metal components are used.

The first aim of studying, is start with metallic materials where is possible to setup models capable of describing the elastic-plastic behavior and damage, providing evolution from the initiationof damage, propagation until final damage, a perforation.

Thus, evaluation parameters are establish to have a comparison between physical tests and numerical simulation, the idea is to enable the use of modeling for cases of shielding levelII-A [2], [3].

First is to present the experimental evaluation performed using a projectile-type 9mm FMJ (Full Metal Jacket) with ammo tip ogival.

Following is present equating of the plasticity model with damage using the finite element modeling. The results obtained are present in each operation and a comparison between physical test and numerical simulation is evaluate.

II. BALLISTIC IMPACT – PHYSICAL TEST

For the assessed physical test, it was consider a dimension plate of300x300mm.It was tested two steel plates, one called AISI / SAE 1010 with a thickness of 2.9mm, and a steel dedicated to the shielding called Ramor 500 with a thickness of 2.5mm.The test was perform at a distance of 7m using a 9mm projectile with ogival tip, mass 10.5g and the theoretical speed of 384m/s.

From the impact, some measurements were perform, the final thickness of the plate in impacted area, also the final displacement caused by the impact. In these tests, none perforation occur as shown in Figures 1 and 2.

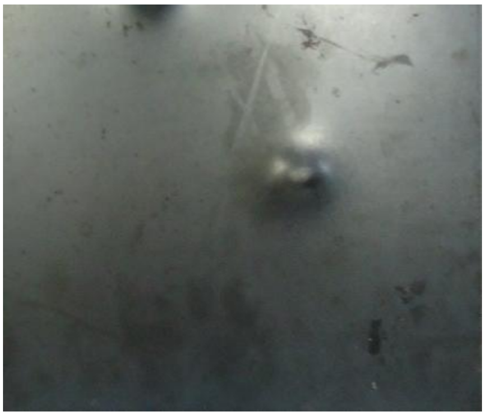

Fig. 1 – Steel plate of AISI/SAE 1010

Fig. 2 – Steel plate ofRamor 500

In figure 3 is show the densification obtained in the thickness of steel AISI / SAE 1010, while the plate Ramor500 steel did not show any change in deformation. To evaluate the densification in thickness, the plate was cut into the impact region to ensure the final thickness measurement.

At steel Ramor500 only marks on the outer surface were obtained from impact. No additional testing was conducted, only the verification that this steel with 2.5mm of thickness could be used forblindage level II-A.

Only steel AISI/SAE 1010it was checked the changing of cross section. The final thickness measured on the plate was 2.6mm.

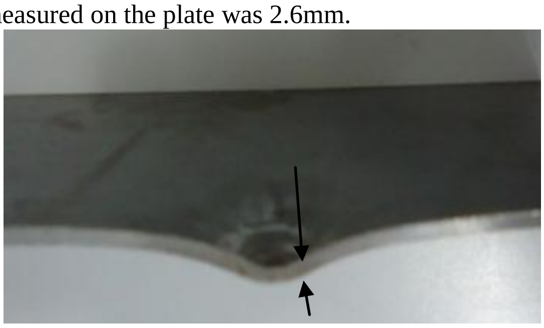

Fig. 3 – Cut of plate in region of impact

III. Finite Element Modeling

The problem of finite element was develop in the framework of the dynamics, using explicit integration algorithm. Elements of shell typewith a high degree of refinement in the impact area used, 0.1mm of refinement in the impact region, as shown in Figure 4.

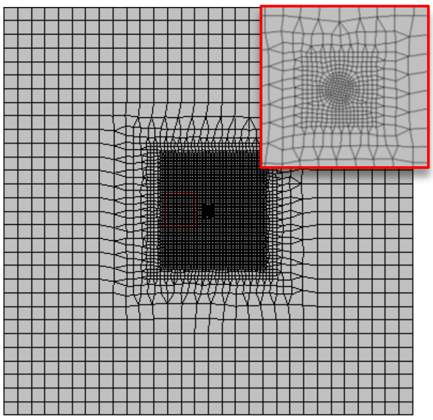

Fig. 4 – Type of mesh refinement adopted

The plasticity model used is the Johnson-Cook that consider the effect of the strainrate$\dot{\varepsilon}$, the initiation of damage is governed by the formulation of the FLD forming limit curve and the propagation of damage based on the strain energy.The plasticity's law from is given by equation 1.

$$\sigma = A + B\left(\varepsilon_p\right)^n \left(1 + c\ln\frac{\dot{\varepsilon}}{\dot{\varepsilon_0}}\right) \quad (1)$$

where,

A, is yield stress;

B, is hardening coefficient;

n, exponent of hardening;

c, coefficient of strain rate

$\dot{\varepsilon_0}$, strain rate of reference.

The initiation criterion of damage, is introduced via forming limit curve that starts when the variableω_{FLD} =1.The forming limit diagram was proposedby Keeler andBackofen in 1964 [4], that is used to measure the amount of material that resists to deformation before the initiation of necking, as shown in Figure 5.

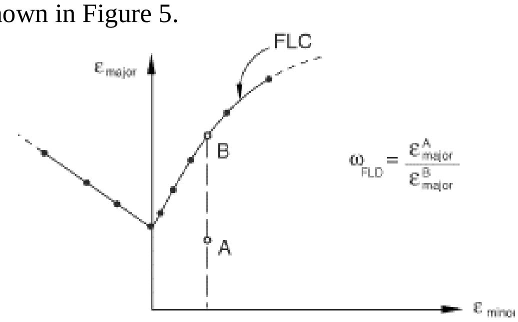

Fig. 5 – FLD, FormingLimitDiagram

Where:

ω_{FLD}, state variable of damage, which increases with plastic deformation.

$$\omega_{FLD} = \frac{\varepsilon_{máxima}}{\varepsilon_{FLD}^{máxima}\left(\varepsilon_{mínima}, \theta\right)} \quad (2)$$

ε_{major}, maximum deformation limit,
ε_{minor}, minimum deformation limit;

Already, the formulating of damage propagationusedis the one proposed by Hillerborg (1976) [5] where it uses the fracture energyG_f given by:

$$G_f = \int_0^{u_f} \sigma_y du_f \tag{3}$$

At this equation,u_f is the plastic equivalent displacement, and it is consider through following rule:

- Before damage start:$u_f = 0$
- After damage start:$u_f = \frac{2G_f}{\sigma_y}$
- At fracture:$u_f = \frac{2G_f}{\sigma_y}$

σ_y, is yield stress,

The graph of Figure 6 shows the relationship between stress and plastic displacement of fracture.

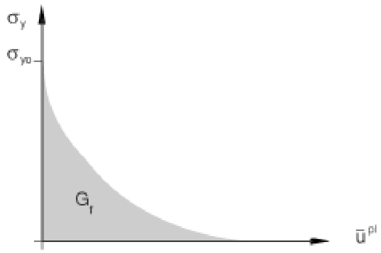

Fig. 6 - Fracture energy evolution

The law of evolution of the stresses with damage is given by equation 4.

$$\sigma^d = (1 - d)\sigma^u \tag{4}$$

where:

σ^d, is a stress damaged and σ^u a stress undamaged.

d,is a damage variable.

IV. Results

According to physical test evaluation, the maximum displacement measured was 10.5mm in curvature, presented at figure 7.The final thickness measured at the center of impact region was 2.6mm. It was proposed,a form of the measuring the maximum displacementconsidering thickness.

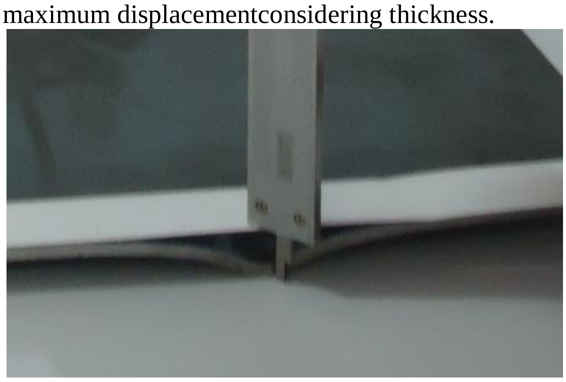

Fig. 7 - Measurement of the displacement due to the impact.

In numerical modeling via finite element method, the maximum displacement was 10.32mm, figure 8, and final thickness evaluated at impact region was 2.49mm, figure 9.

To expose the gain of the analysis using the elastic-plastic model with damage, rather than the conventional plasticity in Figure 10, is present a simulation considering a model material without any law of damage evolution and without the inclusion of deformation rate effect.

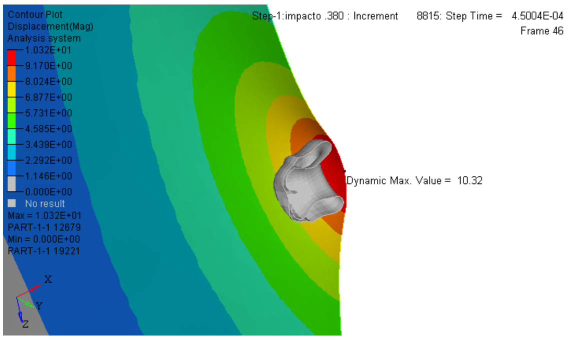

Fig. 8 –Total displacement on the plate, curvature displacement.

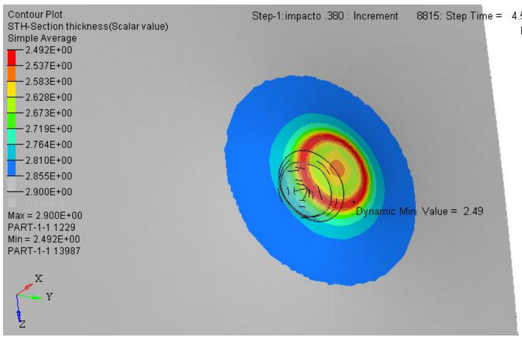

Fig. 9 - Final thickness of plate in simulation

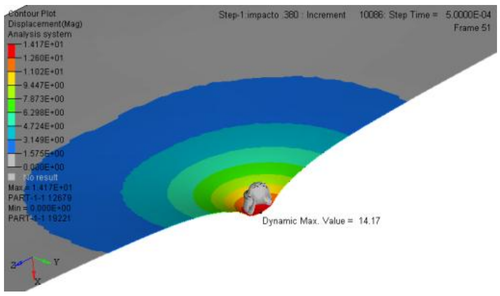

Fig. 10 - Result of curvature displacement for conventional plasticity

V. Conclusion

The shell element used to modeling plasticity and damage via finite element method presented values with a higher degree of correlation than 95%, while the use of the conventional plasticity the response was quite different. The use of shell elements to capture the behavior of the material proposed due to the low computational cost, unlike the approach using solid elements.

Noted that the deformation rate must be employed, and to modeling a component in fracture situation, the inclusion of damage variable is highly recommended.

The accuracy of the results is drive by the use of elastic-plastic curves for different strain rates as more curves are used, better the model describe structural behavior.

As for the criterion of damage initiation is necessary a caveat, because a good characterization of deformation limits, maximum and minimum deformation to predict beginning of material necking, as established by the forming limit diagram FLD also assist in better representation damage.

The energy criterion for the damage propagation is use due to an attempt to make the nondependent problem of mesh size even adopting a high refinement in the impact area.

This methodology becomes possible, after the calibration of material model, the useof finite element method for verification of different types of projectiles. Also provides a prediction of the final displacement field and with the model is able to study the structural behavior of metal sheets during an impact ballistic. Finally, assist the armored project to avoid perforations.

Acknowledgements

Grateful for professorPedro by support and PUC Minas, CAPES, and companies Vision Graphic andRuukki. Also thankful to Brazilian Federal Police and TiroUrbano Defensive School.

REFERENCES

[1] B. Gama, J. Gillespie, Finite element modeling of impact, damage evolution and penetration of thick-section composites, *International Journal of Impact Engineering, 38(4)*, 2011, 181-197.

[2] Norma NEB/T E-316, Proteção Balística de Carros de Passeio, *CTEX, 200.*

[3] NIJ Standard-0101.06, Ballistic Resistance of Body Armor,*U.S. Department of Justice*,1985.

[4] S. P. Keeler, A. Backofen, Plastic instability and fracture in sheets stretched over rigid punches, *ASM Transactions Quarterly*, 56, 25–48, 1964.

[5] A. Hillerborg, M. Modeer, P.E. Petersson, Analysis of Crack Formation and Crack Growth in Concrete by Means of Fracture Mechanics and Finite Elements, *Cement and Concrete Research*, 6, 773–782, 1976.