ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/12314005.1138156

VALIDATION STUDIES OF THE SIMPLIFIED MODEL OF THE MISSILE WITH CUMULATIVE HEAD

Kamil Sybilski, Robert Panowicz, Damian Kołodziejczyk, TadeuszNiezgoda

Military University of Technology, Department of Mechanical Engineering
Kaliskiego Street 2, 00-908 Warszawa, Poland
tel.: +48 22 683 73 48, +48 22 683 94 61, fax: +48 22 6839355
e-mail: ksybilski@wat.edu.pl. rpanowicz@wat.edu.pl
dkolodziejczyk@wat.edu.pl, tniezgoda@wat.edu.pl

Abstract

During ongoing conflicts around the world, some of the most dangerous threats are rocket-propelled grenades. They are light, easy to use and cheap, and their penetration reaches 900 mm of armoured RHA steel. Therefore, in many experimental and numerical laboratories there are researches of different, in aspect of shape, dimensions and used material, types of rod armours protecting against such a threat carried out. This kind of research must be confirmed in the last part of design by appropriate field tests. However, at the beginning of the design process, it is important to find a fast method of testing the developed solutions in a manner, which allows us to observe and measure as many process parameters as possible. One of such tests is research with the use of simplified models built in an appropriate scale. Small dimensions and simple construction result in the fact that the cost of manufacturing of models is much lower, and time is shorter. Tests on the simplified models are also possible to be carried out in laboratory conditions, what reduces the costs and makes the measurements of physical properties easier.

The article describes experimental investigations of the impact of a simplified missile with a shape charge jet head model into a thick plate and steel rods. The construction of a missile, the used equipment and apparatus as well as the proceedings of the experiment are presented. The paper provides information about the structure of FE model and both initial and boundary conditions of the examined system. The results of numerical analysis are presented and compared with the results from experimental tests.

Keywords: *RPG, simulation, FEM analysis, safety, slat armour, rod armour*

1. Introduction

Fast growing techniques give the opportunity for better knowing and understanding phenomena occurring in the nature. Increasing possibilities of new research measuring equipment, allows observation and testing wider phenomena, in an extended range. Unfortunately, the price of these equipments increases along with their technical capabilities. In many cases, these appliances are very sensitive to external impulses, what excludes their use in some tests. For such tests, we can include, among others, observation of a pressure wave formed during detonation and all kinds of crashing of bodies moving at high velocity, especially when at least one of them is subjected to fragmentation.

The example of very difficult tests, from the perspective of measurements and recording processes, is impact of a missile with a cumulative head into the rod armour. This phenomenon is an object of research carried out by the team of Department of Mechanics and Applied Computer Science, Faculty of Mechanical Engineering at Military University of Technology, within the frameworks of the project on the rod armour against this kind of threat [1, 2, 3]. During interaction of the missile with the rod armour, the detonation of explosive material placed inside the missile can occur, what causes the burst of both its parts and the elements of the armour as well as their scattering in a radius of several tens of meters. Those shrapnels frequently have significant sizes (in relation to a missile size) and move at very high velocity, therefore they can destroy fragile measuring equipment in an easy way. Also, a pressure wave formed during detonation is as strong as it may have devastating effects on the surroundings elements. Therefore, in the preliminary stage of tests, the measurement was limited to observation of phenomena using a fast camera. The camera was placed about 80 meters from the armour, perpendicular to the axis of the projectile. This observation allowed determination of the moment of the explosive material detonation and determination whether the interaction of the touching bodies lead to unsealing of the projectile head[4]. This unsealing causes extraction of the explosive material to the outside (seen as dark grey shadow), thus significantly weakening the destructive forces of the projectile.

In the next stage of experimental tests, there was introduced a modification in the method of observation. The modification consisted in setting a mirror behind the armour. The mirror was placed in such a way that the camera could record perpendicular and normal view to the axis of the projectile at the same time. Thus, it was possible to determine a level of deformation of the projectile and armour elements. However, it was not still enough to validate a numerical model of the missile, which was planned to be used for fast tests of new constructions. Especially, it was very important to determine an adequate method to describe the behaviour of the material used in a real projectile and an influence of the friction coefficient. Therefore, the authors decided to perform an additional test using a simplified model of the projectile in the laboratory.

In the paper, experimental tests of the simplified model of PG-7G projectile and numerical analysis, in which the authors tried to reflect behaviour of real structures are described. In the final part, the conformity results assessment, and thereby the assessment of the correctness of the description of the analyzed structure are carried out.

2. Construction of simplified PG-7G missile

During developing the simplified model of PG-7G projectile, the main assumption was making possible conducting of all the tests in the laboratory and the lack of any shrapnels, which could damage the measuring equipment. Therefore, the authors decided to build a simplified and smaller model called MPG-2. MPG-2 consists of three basic elements (fig 1.): ballistic cap, head and shank. It does not include explosive material and any other dangerous materials. During developing the shape, the authors endeavoured to reflect, keeping to the scale, the mass, basic dimensions and the position of the centre of mass of the real object. A lot of attention has been paid to reflection of the front part of the head, because this element is the most deformed during impact into obstacle. The projectile was made of the same material as the real head (PA 6).

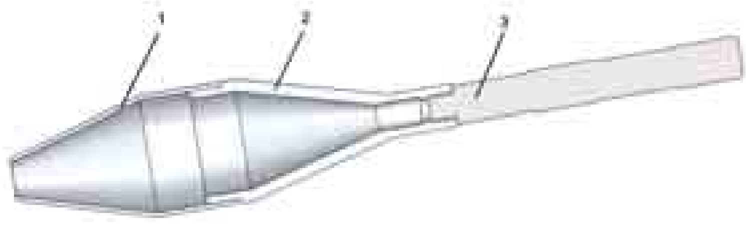

Fig. 1. Construction of MPG-2 projectile: 1 - ballistic cap, 2 - head, 3 - shank

The dimensions of the rear part of the MPG-2 were forced by an available flinging system used during tests. A shank diameter needed to be selected in order to make possible placing it in the barrel (barrel calibre d = 9,65 mm). The total projectile track in the barrel was 69 mm. To accelerate the model, there was used a one-step gunpowder system, depending on the type and mass of gunpowder, which enables the examined model to accelerate from 100 to 200 m/s.

3. Testing stand description

Experimental tests were conducted in two stages. In the first stage, projectile MPG-2 hit into the plate sloped at angle 30 . A stand scheme used in this stage is shown in Fig. 2. It consists of a rigid steel plate, arms for regulation and mounting elements (screws, frame, pedestal). Angle regulation of the testing stand is accomplished by screwing the plate to the right holes in the arms. Spacing of the holes has been selected to provide angle adjustment every 5 degree. Steel plate thickness was 20 mm, what causes lack of deformation during impact with the projectile.

The second stage projectile MPG-2 impact into rods with circular cross sections (diameter was 5 mm). the rods during tests were assembled on the stand consisted of side plates, squares, clamps, pedestal and frame (Fig. 3). The excisions were performed in the pedestal, making it possible to rotate the stand at the angle of 0 ± 45°. Similar excisions were performed in squares. In this case, they are used to rotate the rods plane in the angles range of 0 ± 60°. The rods were made of 45 steel and they were 380 mm long. Their ends, were holding down to squares surfaces with the use of screws. The flinging system was set so that the projectile has always hit between the bars at velocity of 150 m/s.

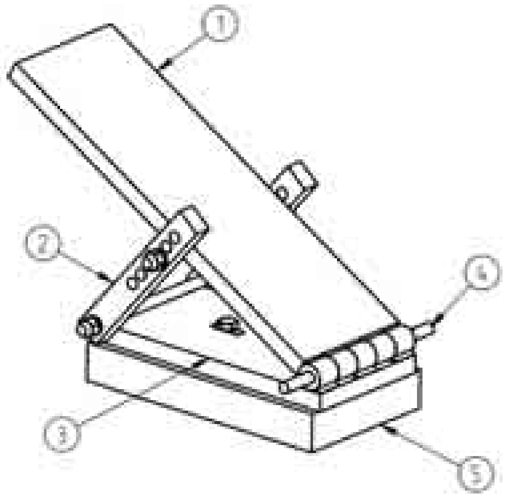

Fig. 2. Construction of the stand with the sloped plate: 1. Plate, 2. Arms for regulation, 3. Pedestal, 4. Pin, 5. Frame

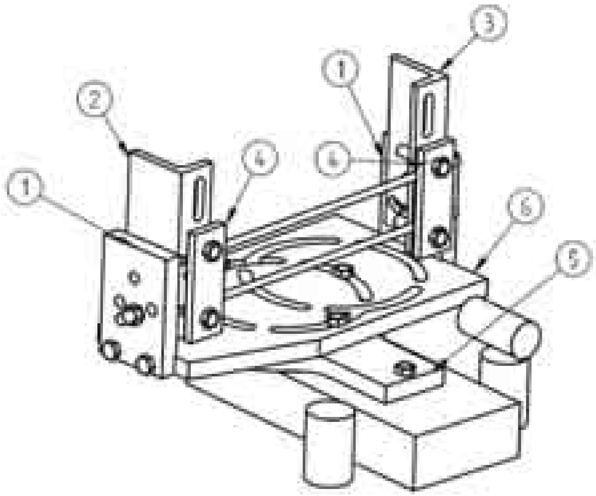

Fig. 3. Construction of the stand with rotated and sloped rods: 1. Side plates, 2. - 3. Squares, 4. Clamps, 5. Pedestal, 6. Frame

The tests were recorded using a fast camera set in a short distance from the stand. Owing to this fact and the lack of any elements between the camera and the stand, it was possible to precisely record the impact process. Phantom V12 camera was used to record the course of the experimental test.

4. Numerical model

In order to validate the description method of the analyzed structure, a numerical model of the projectile and the above described stands was performed. The greatest emphasis was put on the precise representation of the MPG-2 projectile. To represent its geometry, especially thickness, the projectile was divided into six parts (Fig. 4) with different values of thickness. Five parts, assembled into a ballistic cap, were described by four-node finite elements with four points of integration. The shank was described by 3D eight-node elements with one point of integration.

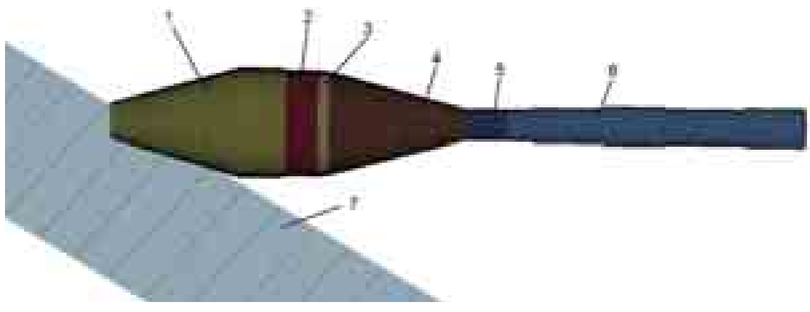

Fig. 4. Numerical model of the projectile. Projectile consists of 6 parts (1-6). 7 is the plate

To describe behaviour of materials used in the projectile, a multilinear material model of Piecewise-linear-plasticity type was applied. The model allows the representation of the material behaviour using lines from, for example, experimental tests. Moreover, it also allows the introduction of a failure model based on erosion and Cowper-Symonds hardening model. In the presented model both effects were used.

During numerical tests, simplified models of the stands were used. During impact into the sloped plate, the model of the stand was simplified only to the plate. It was built using four-node elements, which were given rigid properties (due to the lack of strains during the experimental tests). A numerical model of the stands with rods was simplified only to the rods built of eight-node elements. These elements were given material properties the same as in the case of 45 steel. The ends of the rods were fully fixed.

A boundary condition in the form of contact type automatic-single-surface was introduced between all the parts. This contact assumed that each part of the model is possible to contact with another ones during numerical calculation.

All the numerical calculations were performed in LS-Dyna software program [5], which is used, among other, to analyze fast changing phenomena with a finite element method. It solves a dynamic equation of movement using an explicit integration method, which is widely applied in analyses of highly nonlinear problems, in which high strain, deformation and strain rate occur.

5. Results

Figures 5-7 present photos taken during the experimental tests of projectile impact into the plate. Few characteristic points could be separated during this process. In the initial phase of the impact, the ballistic cap is crushed at simultaneous sliding of this element on the plate. In the next stage, the shank is broken, as a result of a sudden change of the projectile direction. After that, the shank still follows the front part of projectile and hits into the plate. The impact into the plate is so strong that it causes bending of the shank.

Fig. 5. Impact into the plate - beginning of bending of the projectile rear part

Fig. 6. Impact into the plate - total breaking of the projectile rear part

Fig.7. Impact into the plate – the shank hits into the plate

A similar course of a deformation and damage process was observed during the numerical analysis (Fig. 8-10). The difference was only in the manner of shank breaking. During the real test, the material was cracking and the broken parts remained partially connected with rest of the ballistic cap. In the case of numerical analysis, the damage process was obtained through erosion. The elements, which reached failure strain, disappeared. As a result, the shank was fully disconnected from the rest of the projectile at a certain moment.

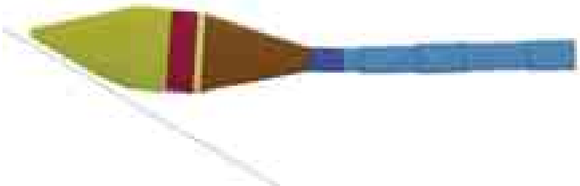

Fig. 8. Impact into the plate

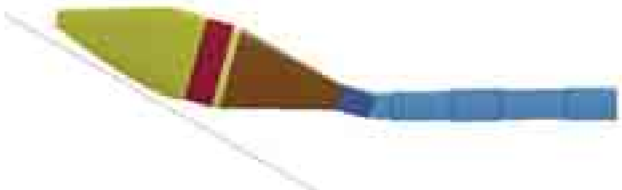

Fig. 9. Impact into the plate - the projectile rear part is fully disconnected

Fig. 10. Impact into the plate – the shank hits into the plate

Apart from the comparison of projectile deformation process during impact into the plate, ballistic cap deformation after impact was compared. Fig. 11 presents a real ballistic cap, which has a visible longitudinal cracks starting at the initial point, of the contact between the projectile and the plate. Cracks edges, as a result of impact, were wrapped to the inside of the projectile. Additionally, it can be observed that the surface of this element is ground. In the numerical model, (Fig. 12) identical cracks and edge wrapping occur. However, there are no visible effects of grinding on it since the model was built of shell elements.

Fig. 11. Front part of the projectile after impact into the rod

Fig. 12. Numerical simulation of projectile impact into the plate - maximal destruction of the ballistic cap

The other test, aiming at validation of the numerical model, was projectile impact into two steel rods. Fig. 13 and 14 present the initial and final stages of interaction of those elements recorded using a fast camera. In the initial impact stage, it is possible to observe that only the central part of rods is deformed. As a result, a small lobe is formed. This lobe, in the next stages of impact, propagates to rest of the rods. In the final stage, it is observed that the rods, extend to a larger spacing than the projectile diameter, what results from the received impulse.

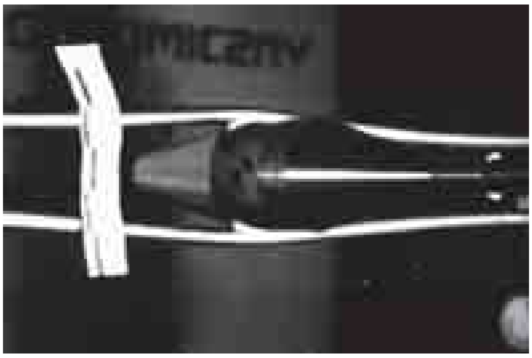

Fig. 13. Projectile impact into steel rods –the initial stage of the process

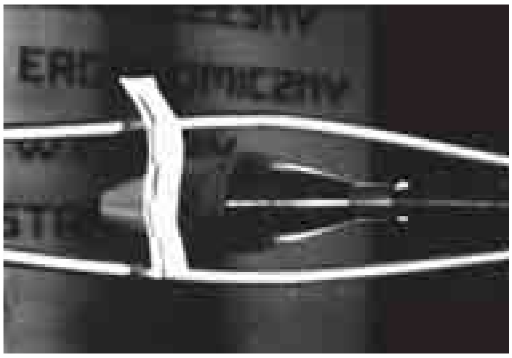

Fig. 14. Projectile impact into steel rods – the final stage of the process

An identical effect can be observed during numerical simulation (Fig. 15-16). The projectile is less deformed during impact.

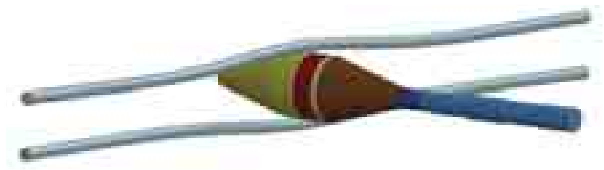

Fig. 15. Numerical analysis of projectile impact into rods – the initial stage

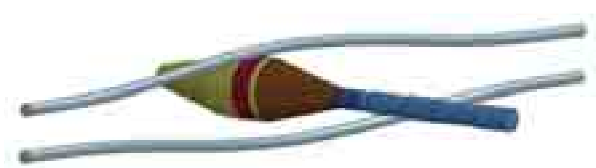

Fig. 16. Numerical analysis of projectile impact into rods – the final stage

6. Summary

In the paper, the results of the experimental test and numerical simulation of projectile impact into the rigid plate and elastic rods are presented. The deformation process of stands and the projectile indicates high correlation between the numerical model and the real projectile. Behaviour of the structure and the time of their characteristic effects occurrence are the same in both cases. It warrants the statement that the developed numerical model is reliable, and the chosen method of description of material, finite elements and contacts is correct.

References

[1] Sybilski, K., Panowicz, R., Niezgoda, T., Barnat, W., *Analiza numeryczna uderzenia pocisku z głowicą kumulacyjną w pancerz wykonany z kątowników*, Mechanik, Nr 1, 2011.

[2] Niezgoda, T., Panowicz, R., Sybilski, K., Barnat, W., *Numerical analysis of missile impact being shot by rocket propelled grenades with rod armour*, WIT Transaction on Modelling and Simulation, 51/2011.

[3] Sybilski, K., Panowicz, R., Kołodziejczyk, D., Niezgoda, T., Barnat, W., *Opracowanie i badania porównawcze modelu zapalnika WP-7*, Mechanik, Nr 2, 2011.

[4] Panowicz, R., Sybilski, K., Gieleta, R., Kupidura, P., Bazela, R., Magier, M., *Badania eksperymentalne wybranego pancerza prętowego*, Biuletyn Naukowy Wojskowego Instytutu Technicznego Uzbrojenia, T. 2, 2011.

[5] Hallaquist, J. O., *LS-Dyna, Theoretical manual*, Livemore Software Technology Corporation, California 2005.