Advances in Engineering Software 123 (2018) 93–103

Contents lists available at ScienceDirect

Advances in Engineering Software

journal homepage: www.elsevier.com/locate/advengsoft

Research paper

Numerical simulation of missile warhead operation

G. Martynenko[b], M. Chernobryvko[a,b], K. Avramov[a,b,*], V. Martynenko[b], A. Tonkonozhenko[c], V. Kozharin[c], D. Klymenko[c]

[a] Podgorny Institute for Mechanical Engineering, National Academy of Science of Ukraine, Department of Vibrations, 2/10 Pozharskogo St., 61046 Kharkiv, Ukraine
[b] National Technical University "Kharkiv Polytechnic Institute", Department of Dynamics and Strength of Machines, NTU "KhPI", 2, Kyrpychova str, 61002 Kharkiv, Ukraine
[c] Yangel Yuzhnoye State Design Office, Dnipro, Ukraine

ARTICLE INFO

Keywords:
Missile warhead
Impact pressure
Dynamic fracture
Dynamic stress-strain state
Tension buckle

ABSTRACT

Numerical simulation of a missile warhead dynamic fracture is considered. The design of the warhead basic units is treated. The approach for simulations of the warhead fracture in software ANSYS is proposed. This approach is split on three stages. (I). The analysis of the static stress-strain state of the warhead, which is arisen owing to its assembling. (II). Calculations of the dynamic stress state of the structure. (III). Analysis of dynamic fracture of the most loaded units. The parameters of the warhead are chosen in order to a fracture takes place in the structure specified area.

1. Introduction

The subject of warhead dynamic fracture has been interested in the military field. Therefore, a lot of efforts were made to study this problem. Numerical computer model for simulation of the fragmentation parameters of explosively driven shell is developed by Gold [1]. The warhead, which is treated in [2], consists of the composite casing and high explosive, which can greatly reduce the damage to objects outside the damage range. In order to evaluate its blasting damage, three types of charges were analyzed experimentally. The paper [3] presents a design for a protective bulkhead in the form of a multi-layered composite structure, which consists of a front plate, front aerogel felt, anti-penetration layer and buck plate. Hemispherical-nosed warhead is manufactured to simulate the warhead of anti-ship missiles. The protective capability of the multi-layered composite structure is analyzed experimentally. The use of sintered metal compositions for the shells of explosive warhead is treated in [4]. The analysis is based on the simulations with different numerical approaches and the models for the launch and the impact phase. The new formula is proposed to predict the initial fragment velocity of the warhead with hollow core in [5]. The spatial dispersion of fragment generator warhead is analyzed experimentally in [6]. This paper presents the explanation of the fragment dispersion phenomenon using one dimensional shock theory. The experimental results of the paper [7] show, that the acceleration process of the warhead can be divided into two distinct phases: initial acceleration under shock waves and further acceleration under the load of detonation products. The simulation was developed to predict damage from closely spaced tumbling rods in [8]. This simulation predicts the synergistic effects from any collateral damage against submunition and bomblet payloads. A projectile system is proposed to improve efficiency and effectiveness of damage done by anti-tank weapon system on its target by designing a ballistic projectile that can split into multiple warheads and engage a target at the same time [9]. A three dimensional integrated guidance and control law with impact angle constraint is developed for the bank to turn missile attacking a ground fixed target in the presence of input saturation and actuator failure in the paper [10]. Numerical simulations are implemented to demonstrate the effectiveness and robustness of the integrated guidance and control law. Vulnerability assessment methodology, which is treated in [11], consists of calculation of the dispersion of the fragments, determination of the hit locations, penetration calculations and probability of kill calculations for the aircraft utilizing the fault tree established for the particular aircraft studied. Based on the developed methodology, vulnerability assessment and survivability analysis of a generic aircraft is performed. In the paper [12], the response of buried shelters to blast loadings due to conventional weapon detonation has been investigated using the finite element method. The finite element analysis was carried out using a commercial finite element software package, ABAQUS. The validity of finite element model parameters adopted was established by comparisons with existing empirical formulae. The paper [13] is devoted to the simulation of penetration of steel clamping into aluminum sheet. The calculations were performed using the Lsdyna finite element

* Corresponding author.
E-mail addresses: gmartynenko@ukr.net (G. Martynenko), kvavr@kharkov.ua (K. Avramov).

https://doi.org/10.1016/j.advengsoft.2018.07.001
Received 14 April 2018; Received in revised form 31 May 2018; Accepted 4 July 2018
0965-9978/

commercial code. An experimental and numerical study on the influence of biaxial preloading on the low velocity impact performance of laminated composite plates was conducted in [14]. Finite element analysis was developed, using Hashin failure criteria for the composite material. The high-strain-rate of the composite casing of solid propellant motor under the action of impact loads is treated in the paper [15]. Development of finite element numerical solution combined with the digital material representation approach to simulate brittle fracture is the subject of the paper [16]. A multi scale concept was used to describe numerically material behavior at various length scales from microstructure to macro sample. In the paper [17], the implementation of the method in Abaqus/Explicit using its co-simulation features to couple two separate Abaqus/Explicit analyses is presented. The approach is illustrated in the case of the simulation of delamination under high velocity impact. The article [18] presents an inverse method for predicting the reference geometry of the plastically deformed body. The reference configuration is found by solving an elastic-plastic boundary value problem. Rate-type elastoplastic constitutive laws are employed in the inverse analysis. The numerical technique for wave propagation problems has been applied to the accurate modeling of stresses in the vicinity of crack tips and the dynamic stress intensity factor for stationary cracks in the paper [19]. The numerical technique includes the linear finite elements with reduced dispersion as well as the two-stage time-integration approach.

The paper [20] considers a shape optimization using an elastic contact problem. Redesign of the contact surface so, that the distribution of the force between contact surfaces of elastic bodies is uniform, is performed.

The monograph [21] is the compendium of engineering models used in high-speed penetration mechanics. The book presents a very detailed summary of the engineering models used for the analysis of high-rate penetration of rigid projectiles into metals. The dynamic fracture mechanics is treated in the books [22] and [23].

The schematic diagram of the missile warhead, which is designed in design office “Yuzhnoye” (Ukraine), is discussed in this paper. The method for the calculations of the warhead dynamic fracture under the action of the impact pressure is suggested on the basis of the software ANSYS. The data of the dynamic fracture calculations and optimization of the warhead parameters, which offers to obtain the preset fracture time and the locus of the structure fracture, is treated.

2. The problem formulation

Fig. 1 shows schematically the considered part of the missile warhead. It consists of the throwing elements, which are arranged in two circles. The throwing elements of the first circle are denoted by the number 1 (Fig. 1). They are arranged on the supports and they are anchored to the shell by means of the clips and the bolted joints. The throwing elements of the second circle are arranged on the supports. They are anchored by two bands with the tension buckle outside the warhead. These throwing elements are denoted by the number 2. At the end of the missile flight, the starting mechanism is actuated, which cause to the scattering of the throwing elements.

The throwing elements of the first circle are arranged on the supports 3, which are welded on the shell. The supports of the second circle 4 are arranged between the throwing elements of the first and the second circles. The supports of the first and the second circles are joined by the couplings 5, which consist of the bolt, the washer and the clamp (Fig. 2). All the above-considered structure is tightened by the unit 6, which consists of four sets of the tension buckles. Two tension bands are installed (Fig. 1). The values of the bend tension and the forces, acting on the throwing elements of the second circle, are governed by the twisting of the bolts. The static state of the structure is brought about by the band tension. The charge, which generates the impact pressure, is installed inside the cylindrical shell 7 (Fig. 1). This impact pressure acts on the throwing elements of both circles. The curves 1 and 2 (Fig. 3)

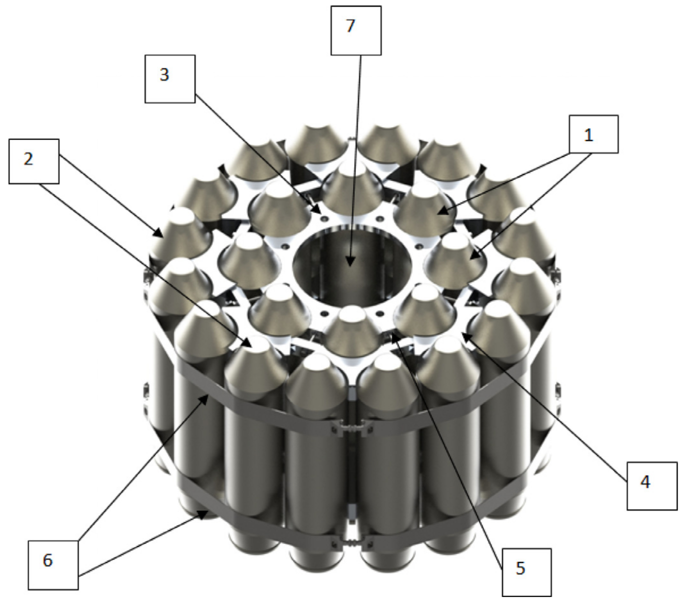

Fig. 1. Sketch of the missile warhead.

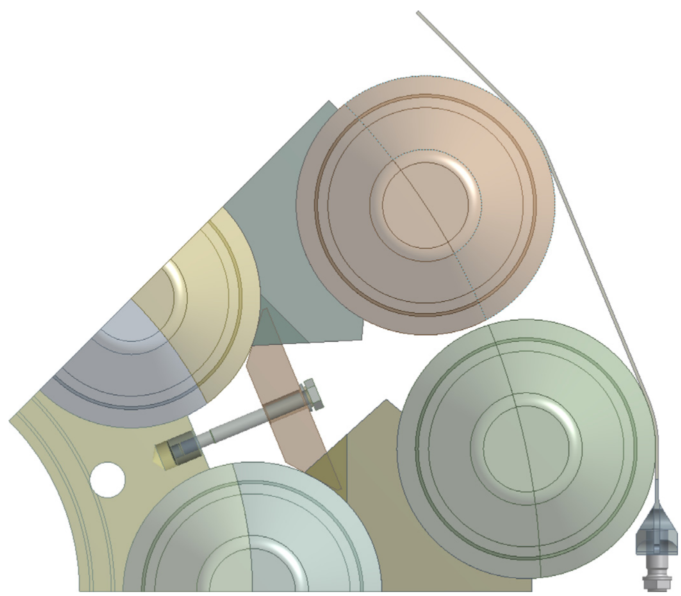

Fig. 2. Configuration of the clips and the tension buckle, which are a part of warhead.

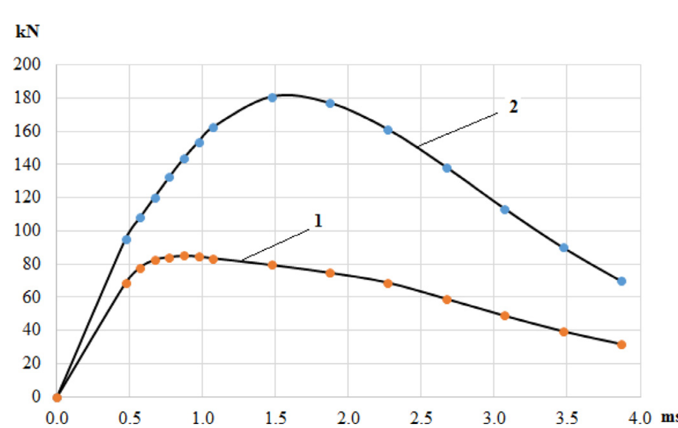

Fig. 3. The pressure resultants acting on the throwing elements.