INNOVATIVE MECHANICAL ENGINEERING **ISSN 2812-9229 (Online)**
University of Niš, Faculty of Mechanical Engineering
VOL. 3, NO 1, 2024, PP. 79 - 89

Original scientific paper*

FEM-BASED STUDY OF A BULLET IMPACT ON A COMPOSITE PLATE

Jovica Tasić[1], Dragan Marinković[1,2]

[1]University of Niš, Faculty of Mechanical Engineering, Serbia
[2]Berlin Institute of Technology, Department of Structural Analysis, Germany

Abstract. *This paper investigates numerically the resistance of a composite plate exposed to the bullet impact load. A 9 mm caliber bullet hits a three-layer plate made of Kevlar 49 material. For this case of impact analysis, a numerical simulation is performed using the finite element method. The simulation is realized in Abaqus software. As a high-velocity impact is analyzed with significant nonlinearities involved, an explicit time-integration scheme is selected as a suitable option. The prediction of plate damage is based on the Hashin criteria defined in a VUMAT subroutine, which is written in FORTRAN programming language. The objective of the work is to analyze how the selected sequence of layers influences the results of the bullet impact. The obtained results are of great significance in the ballistic vests industry, which develops products for passive protection of the user.*

Key words: *Finite element analysis (FEA), Hashin failure criteria, Modeling, High-velocity impact, Numerical simulation, Composite plate*

1. INTRODUCTION

Composite materials [1] are increasingly used as modern materials that enable light and strong structural designs. They are most often used in the form of laminates, obtained by bonding together several layers, each made of a composite material. Such a material design offers numerous options for tailoring material properties by changing the number and thickness of the layers, orientation of the fibers, proportion, form and arrangement of constitutive materials, as well as the constitutive materials of individual layers. In this manner, a composite laminate can be easily adopted to a wide range of requirements from the structural design. As a consequence, they are used in automotive, aviation, aerospace, aeronautical, defense, sport and other industries. A number of research works, such as [2, 3, 4, 5] investigated numerically and experimentally various properties of composite materials.

*Received: January 19, 2024 / Accepted April 25, 2024**
Corresponding author: Jovica Tasić
Faculty of Mechanical Engineering, University of Niš, Serbia
E-mail: jovicatsc93@gmail.com

Some specific applications of composite materials demand knowledge of the limit values of certain properties. Those may be obtained experimentally, but also numerically by performing adequate simulations. As composite laminated structures belong to thin-walled structures, their efficient FE-modeling and simulation are typically done by means of suitable shell elements [6, 7] as well as by means of the isogeometric FE approach [8]. Regarding dynamical behavior, the simulation can be sped up by several order of magnitudes by using model order reduction techniques, such as the modal superposition method. Most of those techniques are applicable in the realm of linear structural behavior, although some recent research works allow their extension to cover modest nonlinearities [9, 10]. In cases where highly localized mechanical behavior is of interest, such as impact analysis, 3D elements are needed to recover the mechanical behavior and the induced stresses in details.

In this paper, the composite plate was numerically tested for impact load, in a similar fashion as it has been done in the work by Narayanamurthy et al. [11]. The simulation was realized in Abaqus software, using the finite element method (FEM). The determined stresses and the resulting failure coefficients were used to predict the plate damage. There are several criteria according to which it is possible to make a prediction: Tsai-Hill [12], Azzi-Tsai-Hill [13], Tsai-Wu [14], Hashin [15]. The Hashin Failure Criterion was used in the paper. In the main program of the software, the subroutine VUMAT for the explicit, dynamic solver is implemented, where the material law is applied. As a part of the VUMAT subroutine, another subprogram was used, written on the basis of the Hashin criteria, which is intended for monitoring the values of the failure coefficients. The visualization of the simulation results showed an image of the plate stress, and the subprogram provided the values of the Hashin coefficients. The analysis of the results served to evaluate the state of the proposed solution as well as to consider the possibility of improvement.

2. Failure Criteria

Failure criteria are commonly used in finite element analysis (FEA) to predict failure events in composite structures [16]. The most commonly used are strength-based failure criteria. They are:
Maximum stress criterion, Maximum strain criterion, Truncated maximum strain criterion, Interacting failure criterion.

The failure criterion is defined by means of failure index given as:

$$I_F = \frac{stress}{strength} \tag{1}$$

A failure occurs when $I_F \geq 1$. The strength ratio is the inverse of the failure index

$$R = \frac{1}{I_F} = \frac{strength}{stress} \tag{2}$$

Failure is predicted when $R \leq 1$.

2.1 Hashin Failure Criterion

The Hashin Failure Criterion (HFC) proposes four separate modes of failure:
Fiber tension, Fiber compression, Matrix tension, Matrix compression.

These are defined by the following four equations, as follows:

$$I_{Fft}^2 = \left(\frac{\sigma_1}{F_{1t}}\right)^2 + \alpha\left(\frac{\sigma_6}{F_6}\right)^2 \text{ if } \sigma_1 \geq 0 \tag{3}$$

$$I_{Ffc}^2 = \left(\frac{\sigma_1}{F_{1c}}\right)^2 \text{ if } \sigma_1 < 0 \tag{4}$$

$$I_{Fmt}^2 = \left(\frac{\sigma_2}{F_{2t}}\right)^2 + \left(\frac{\sigma_6}{F_6}\right)^2 \text{ if } \sigma_2 \geq 0 \tag{5}$$

$$I_{Fmc}^2 = \left(\frac{\sigma_2}{2F_4}\right)^2 + \left[\left(\frac{F_{2c}}{2F_4}\right)^2 - 1\right]\frac{\sigma_2}{F_{2c}} + \left(\frac{\sigma_6}{F_6}\right)^2 \text{ if } \sigma_2 < 0 \tag{6}$$

where α is a weight factor to give more or less emphasis to the influence of shear on fiber failure.

In the above equations, according to Abaqus Analysis User's Manual [17]:
F_{1t} denotes the longitudinal tensile strength, F_{1c} denotes the longitudinal compressive strength, F_{2t} denotes the transverse tensile strength, F_{2c} denotes the transverse compressive strength, F_4 denotes the transverse shear strength, F_6 denotes the longitudinal shear strength, σ_1, σ_2, and σ_6 are components of the effective stress tensor, σ, that is used to evaluate the initation criteria and which is computed from:

$$\sigma = \boldsymbol{M\sigma} \tag{7}$$

where $\boldsymbol{\sigma}$ is the true stress and $\boldsymbol{M}$ is the damage operator:

$$M = \begin{bmatrix} \frac{1}{(1-d_f)} & 0 & 0 \\ 0 & \frac{1}{(1-d_m)} & 0 \\ 0 & 0 & \frac{1}{(1-d_s)} \end{bmatrix} \tag{8}$$

Here, d_f, d_m, and d_s are internal (damage) variables that characterize fiber, matrix, and shear damage, respectively. They are derived from damage variables d^t_f, d^c_f, d^t_m, and d^c_m, corresponding to the four modes previously discussed, as follows:

$$d_f = \begin{cases} d_f^t \text{ if } \sigma_1 \geq 0 \\ d_f^c \text{ if } \sigma_1 < 0 \end{cases} \tag{9}$$

$$d_m = \begin{cases} d_m^t \text{ if } \sigma_2 \geq 0 \\ d_m^c \text{ if } \sigma_2 < 0 \end{cases} \tag{10}$$

$$d_s = 1 - \left(1 - d_f^t\right)\left(1 - d_f^c\right)\left(1 - d_m^t\right)\left(1 - d_m^c\right) \tag{11}$$

3. FINITE ELEMENT MODELING

3.1 Creation of models

To create conditions for the simulation, the analysis object was modeled in Abaqus software. Since it is the contact analysis of the impact between two bodies, a 9 [mm] bullet grain model [18] and a three-layer composite plate model were created.

The grain of the bullet is modeled in an ellipsoidal shape using the technique of rotation around the y-axis, according to the dimensions in accordance with the technical description, where the semi-major axis is 10.5 [mm] and the minor semi-axis is 4.5 [mm], Fig. 1. The center of mass, m = 8 [g], is located at the top of the bullet in the form of a reference point. The bullet is placed as a rigid body ("Discrete rigid"), without the possibility of deformation. It is discretized with finite elements of the type "R3D4 A 4-node linear quadrilateral elements" and "R3D3 A 3-node linear triangular elements", with a total of 2900 elements. The mesh is more finely divided near the top of the bullet, Fig. 1.

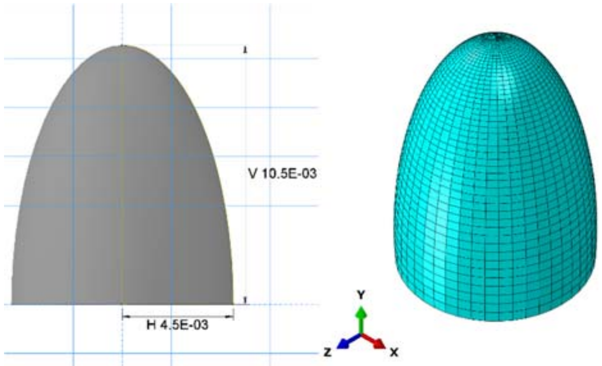

Fig. 1 Contactor model, bullet grain 9 [mm]

The plate is modeled in a three-dimensional square-shaped space with dimensions 100 x 100 [mm], Fig. 2.

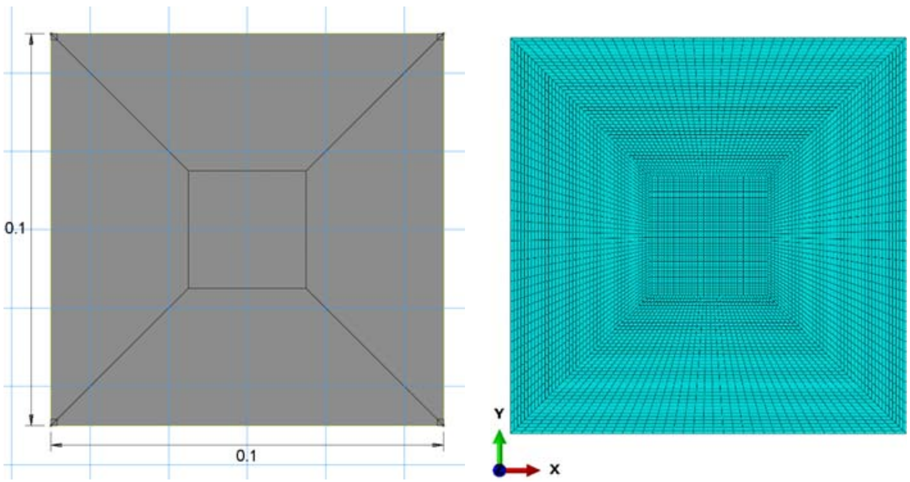

Fig. 2 Target model, composite plate

The structure of the plate is in the form of a 3D deformable plate. It consists of three layers, which make up the total thickness of the plate of 9 [mm]. The orientation of the layers is as follows: [45/90/-45]. It is discretized with elements of the type "C3D8R A 8-node linear hexahedral elements". A total of 465660 elements were used. In the area where the highest stresses are expected, the discretization mesh is of a finer division.

3.2 Material

Based on the criterion that the material should be resistant to impact loads, a design for damage tolerance is recommended. As Kevlar fibers and tough matrices can provide the necessary impact and damage propagation resistance, the type of Kevlar 49 material was chosen. Kevlar composites have low density, high tensile strength, and excellent toughness and impact resistance.

Table 1 Properties of Kevlar 49 material

Property	Value	
Density, ρ	1380	[kg/m^3]
Longitudinal modulus, E_1	80	[GPa]
Transverse in-plane modulus, E_2	5.5	[GPa]
Transverse out-of-plane modulus, E_3	5.5	[GPa]
In-plane shear modulus, G_1	2.2	[GPa]
Out-of-plane shear modulus, G_2	1.8	[GPa]
Out-of-plane shear modulus, G_3	2.2	[GPa]
Major in-plane Poisson's ratio, υ_{12}	0.34	
Out-of-plane Poisson's ratio, υ_{23}	0.40	
Out-of-plane Poisson's ratio, υ_{13}	0.34	
Longitudinal tensile strength, F_{1t}	1400	[MPa]
Transverse tensile strength, F_{2t}	30	[MPa]
Out-of-plane tensile strength, F_{3t}	30	[MPa]
Longitudinal compressive strength, F_{1c}	335	[MPa]
Transverse compressive strength, F_{2c}	158	[MPa]
Out-of-plane compressive strength, F_{3c}	158	[MPa]
Out-of-plane shear strength, F_4		[MPa]
Out-of-plane shear strength, F_5	37	[MPa]
In-plane shear strength, F_6	49	[MPa]
Damping parameter	10^{-9}	

In accordance with the properties of the selected material, according to Issac [19], density ρ, modulus of elasticity E_1, E_2, E_3, and shear modulus G_1, G_2, G_3 in the longitudinal and transverse directions, and Poisson's coefficients υ_{12}, υ_{23}, υ_{13} were defined, with the values shown in Table 1. Also, the strength coefficients are defined as follows: tensile strengths F_{1t}, F_{2t}, F_{3t}, compressive strengths F_{1c}, F_{2c}, F_{3c} along the principal ply direction, and shear strengths F_4, F_5, and F_6, as shown in Table 1.

3.3 Initial and boundary conditions

Initial and boundary conditions determine the movement of the bullet and how the plate is supported, respectively. The bullet is defined to move in a straight line along the negative z-direction V_3 = -100 [m/s] with the effect of rotation around the z-axis counter- clockwise VR_3 = 3500 [rad/s]. The plate is supported at all ends ($U_1=U_2=U_3=UR_1=UR_2=UR_3=0$), as shown in Fig. 3. The bullet hits the central part of the plate at a right angle producing a load. As the load on the plate is greatest when the bullet strikes at an angle of 90 degrees, the worst possible scenario is observed.

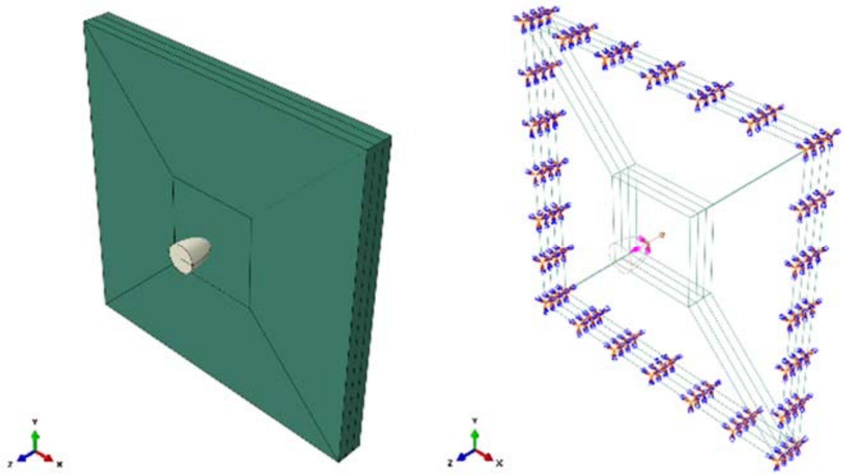

Fig. 3 Initial position and constraint conditions

4. SIMULATION

The simulation was realized in Abaqus based on the previously prepared models and defined initial conditions. It is a nonlinear contact analysis in which the contactor is the grain of the bullet and the target is a composite plate. Since the impact is instantaneous, the explicit dynamic solver of the mentioned software was used. The duration of the simulation is $t = 5 \cdot 10^{-5}$ [s] and it was realized in 89273 integration steps. The correct choice of duration and a sufficient number of steps was a necessary condition for the stability of the simulation. Since the software supports the possibility of implementing subprograms, an algorithm was written to control the flow of the simulation. The main software program is called the VUMAT subroutine, written in the FORTRAN programming language [20], according to instructions in Abaqus User Subroutine Reference Manual [21]. The subroutine code is used to solve the problem. The algorithm flow is the following: reading material properties according to Table 1, calculating the stiffness matrix, updating the total strains and stresses at each integration step, failure evaluation, and integrating the internal specific energy. In the damage monitoring part, another subprogram is called within the VUMAT subroutine, which is written according to the material laws of the Hashin criteria.

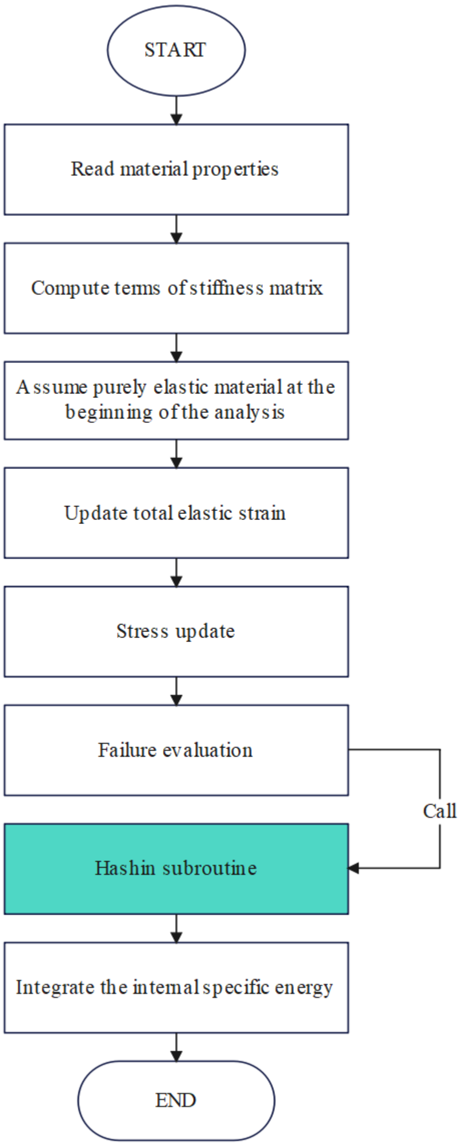

Fig. 4 Algorithm, VUMAT subroutine

Hashin's subprogram uses two *"If"* conditions of inequality in the form of normal stresses σ_1, σ_2, which control the further course of determining the coefficients of failure, according to the algorithm shown in Fig. 5. Stress σ_1 determines the coefficient of fiber tension or compression (Eqs. (3) and (4)), while σ_2 determines the coefficient of matrix tension or compression (Eqs. (5) and (6)). If condition $\sigma_1 \geq 0$ is satisfied, Eq. (3) is applied, otherwise Eq. (4) is applied. If condition $\sigma_2 \geq 0$ is satisfied, Eq. (5) is applied, otherwise Eq. (6) is valid. Eqs. (3) to (6) are used to calculate the failure index I_F, or the strength ratio

R, Eq. (2). If they satisfy the condition that the failure index $I_F \geq 1$, that is the strength ratio $R \leq 1$, the plate is predicted to be damaged.

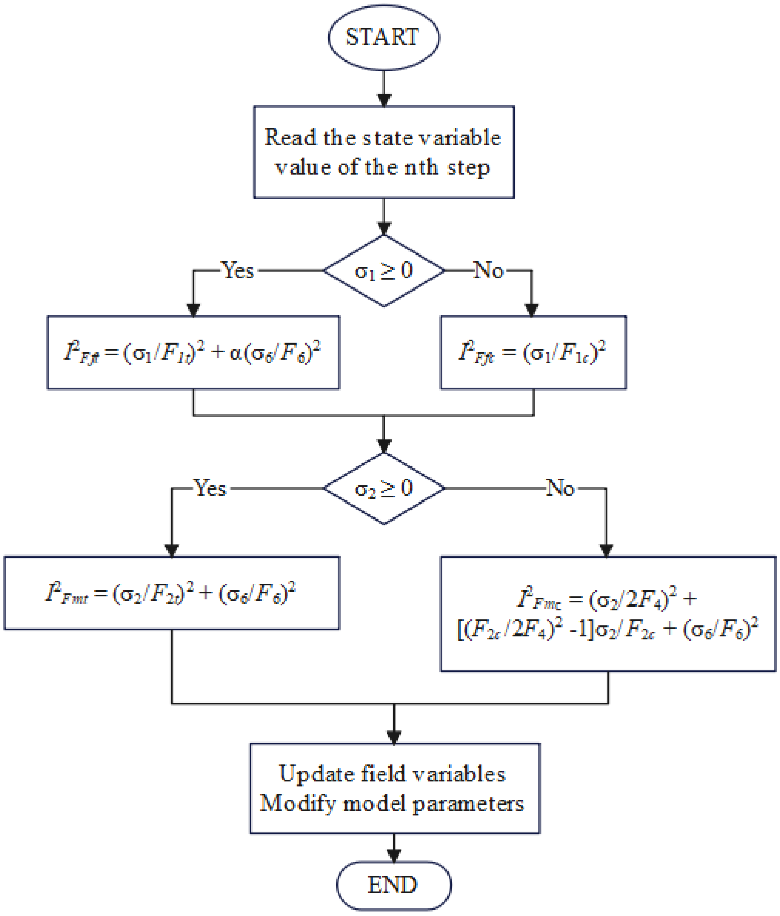

Fig. 5 Hashin subroutine

4. RESULTS AND DISCUSSION

The results of the simulation are shown in 100 images and enable the monitoring of the stress distribution during the realization. The bullet impacts the plate with a certain velocity and rotation. The maximum stresses increase according to the transfer of energy from the bullet to the plate. The visualization of the results showed that the specified stresses were sufficient to cause damage to the plate in the form of a crack. A sufficient number of integration steps allowed the monitoring of the crack formation in the impact zone, Fig. 6. The installation of a three-layer composite plate, with the correct choice of materials and orientation of the layers, prevented the crack from spreading above the impact zone, as well as the failure of the complete plate.

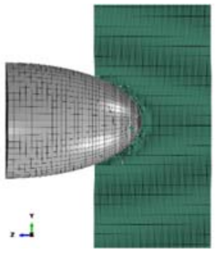

Fig. 6 Display of plate cracks in the y-z plane

The final values of the monitored variables were recorded as relevant. In Fig. 7, the von Mises stress is shown as *S* with the maximal value of σ=1.292·10^9 [Pa]). Figs. 8 and 9 show the normal stresses S_{11} (maximal value σ_1=1.301·10^9 [Pa]) and S_{22} (maximal value σ_2=2.088·10^7 [Pa]).

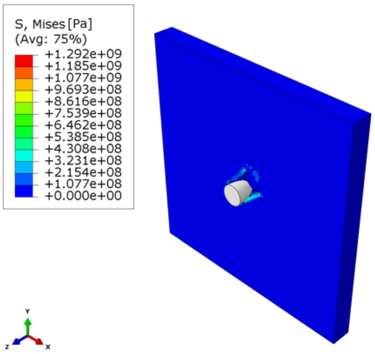

Fig. 7 Von Mises Stress, σ [Pa]

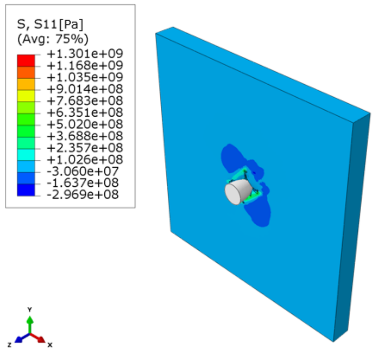

Fig. 8 Normal Stress S_{11}, σ_1 [Pa]

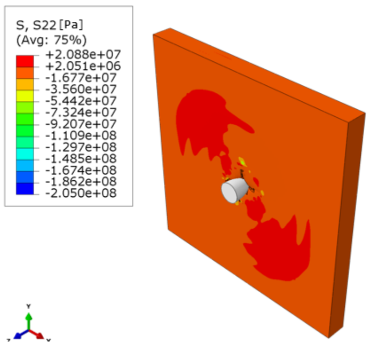

Fig. 9 Normal Stress S_{22}, σ_2 [Pa]

During the realization of the simulation, the failure coefficients were simultaneously recorded in a separate (*txt*) file. The coefficients are presented in the form of a graph, where the x-axis indicates the number of data, while the value of the coefficient is given on the y-axis, Fig. 10.

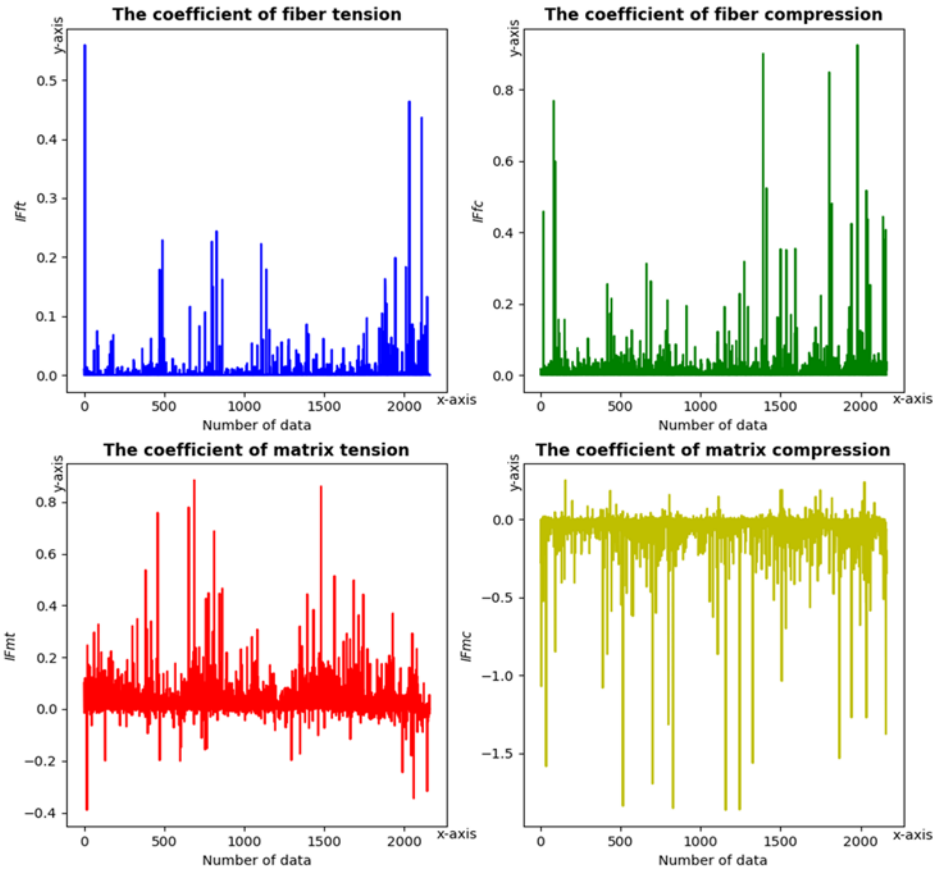

Fig. 10 The failure coefficients

From the analysis of the graphs of all coefficients, it can be concluded that the compression matrix coefficient I_{Fmc} has the greatest influence, since it satisfies the condition of being greater than one $I_F \geq 1$, according to the given criteria. The results of the failure index agreed with the simulation results for stress. In this way, the correctness of the setting and implementation of the simulation in a customized software according to the given problem was confirmed.

6. CONCLUSIONS

This paper was concerned with the numerical simulation of the ballistic impact of a bullet on a composite plate using a nonlinear explicit solver available in Abaqus. The bullet and plate models are discretized with finite elements. A 9 [mm] bullet is defined as a rigid body. The three-layer plate is in the form of a deformable shape and is made of Kevlar 49.

We used the finite element method to test the resistance of the composite material to impact loading numerically. For this purpose, a VUMAT subroutine was implemented in in Abaqus. The subroutine was written in the FORTRAN programming language with Hashin's failure criteria. Hashin's subroutine allowed tracking of the values of the failure coefficients. The results of the coefficients agreed with the simulation results for stresses. Failure criteria were shown to be successful in predicting damage caused by impact loading.

The result of the numerical analysis reveals that the plate would not break and it can be concluded that this plate set meets the requirements for user safety. Further work will focus on experimental verification of the obtained numerical results.

REFERENCES

1. Barbero, E., 2018, *Introduction to composite materials design*, CRC Press, Boca Raton, 535 p.
2. Ha, G.X., Zehn, M.W., Marinkovic, D., Fragassa, C., 2019, *Dealing with nap-core sandwich composites: How to predict the effect of symmetry*, Materials, 12(6), 874.
3. Li, S., Liu, W., Mao, Y., Hou, S., 2023, *Numerical cross-scale optimization of homogenized composite laminates under impact loading*, International Journal of Mechanical Sciences, 245, 108108.
4. Ha G.X., Marinkovic D., Zehn M.W., 2019, *Parametric investigations of mechanical properties of nap-core sandwich composites*, Composites Part B: Engineering, 161, pp. 427-438.
5. Nastos, C., Komninos, P., Zarouchas, D., 2023, *Non-destructive strength prediction of composite laminates utilizing deep learning and the stochastic finite element methods*, Composite Structures, 311, 116815.
6. Rama, G., Mairnkovic, D., Zehn, M., 2017, *Linear shell elements for active piezoelectric laminates*, Smart Structures and Systems, 20(6), pp. 729-737.
7. Rama, G., Marinkovic, D., Zehn, M., 2018, *High performance 3-node shell element for linear and geometrically nonlinear analysis of composite laminates*, Composites Part B: Engineering, 151, pp. 118-126.
8. Milić, P., Marinković, D., Klinge, S., Ćojbašić, Ž., 2023, *Reissner-Mindlin Based Isogeometric Finite Element Formulation for Piezoelectric Active Laminated Shells*, Tehnicki Vjesnik, 30(2), pp. 416-425.
9. Marinković, D., Zehn, M.W., 2010, *Geometric stiffness matrix in modal space for multibody analysis of flexible bodies with moderate deformations*, Proceedings of ISMA 2010 - International Conference on Noise and Vibration Engineering, Leuven, Belgium, pp. 3013-3019.
10. Marinković, D., Zehn, M., Marinković, Z., 2012, *Modal-space based solutions including geometric nonlinearities for flexible multi-body systems*, Civil-Comp Proceedings, 8th International Conference on Engineering Computational Technology, ECT, Vol. 4.-7- September 2012, Valencia, Spain.
11. Narayanamurthy, V., Rao, C., Rao, B., 2014, *Numerical Simulation of Ballistic Impact on Armour Plate with a Simple Plasticity Model*, Defence Science Journal, 64(1), pp. 55-61.
12. Hill, R., 1948, *A Theory of the Yielding and Plastic Flow of Anisotropic Metals*, Proceedings of the Royal Society, 193.
13. Azzi, V., Tsai, S. 1965, *Anisotropic Strength of Composites*, Experimental Mechanics, 5(9), pp. 283-288.
14. Tsai, S., Wu, M., 1971, *A General Theory of Strength for Anisotropic Materials*, Journal of Composite Materials, 5, pp. 58-80.
15. Hashin, Z., 1980, *Failure Criteria for Undirectional Fiber Composites*, Journal of Applied Mechanics, 47(1), pp. 329-334.
16. Barbero, E., 2013, *Finite Element Analysis of Composite Materials Using Abaqus*, CRC Press, Boca Raton, 406 p.
17. Abaqus Analysis User's Manual, Materials, 2010, Dassault Systèmes Simulia Corp, RI, USA, 3.
18. Cacoilo, A., Mourao, R., Teixeira-Dias, F., Azevedo, A., Coghe, F., 2021, *Modelling Ballistic Impact on Military Helmets: The Relevance of Projectile Plasticity*, Defence Technology, 17, pp 1699-1711.
19. Isaac, D., 2006, *Engineering mechanics of composite materials*, Oxford university press, New York, 396 p.
20. Nyhoff, L., Lesstma, S. 1995, *Fortran 77 and Numerical Methods for Engineers and Scientists*, A Simon & Schuster Company, New Jersey, 764 p.
21. Abaqus User Subroutines Reference Manual, 2010, Dassault Systèmes Simulia Corp, RI, USA.