UDK: 623.56
COSATI: 19-01

Possibility of the FEM Application for a Stress Condition Analysis of an Artillery Projectile Body

Danilo Đurđevac, MSc (Eng)[1)]

This article is dealing with a theoretical analysis of the stress distribution that appears in the body of an artillery projectile during its movement through the barrel. It considers a possibility of application of three-dimensional and axisymmetrical finite elements. Particular attention is paid to mesh quality and its influence on calculation quality. The calculation is based on the 105 mm HE projectile M1. The I-DEAS software package is used for this analysis.

Key words: **artillery projectile, HE projectile, shell body, 105mm caliber, stress, stress analysis, structural analysis, FEM.**

Abbreviations

Distortion – finite element distortion
F_{na} – axial component of the driving band force
L_{max} – FE largest diagonal; longest side of the triangle
L_{min} – FE smallest diagonal
p – propellant gas pressure
p_u – pressure of the fuse
R – radius of the inscribed circle of the triangle
R_{eH} – upper yield strength
Stretch – finite element stretch
V – finite element volume
V_1 – finite element volume translated to the local coordinate system
(x,y) – global coordinates
(ξ,η) – local coordinates
σ_{eq} – von Mises stress
ω – angular acceleration

Introduction

ENGINEERING of artillery projectiles is accompanied by dealing with numerous issues that need to provide:

- safety during exploitation, primarily during projectile travel through the barrel,
- appropriate effects on target,
- achieving a maximum possible range,
- low production cost,
- maximum productivity,
- application of existing production equipment and technological processes, and
- maximum possible unification.

Fulfilling all of these requests is often contradictory, so it is necessary to find some kind of compromise. However, the first one is the most important, and has to be met, even if it means neglecting some of the others.

During its movement through the barrel, the projectile is under influence of forces and momentum, as a result of the propellant gas pressure. The parts of the projectile, primarily the body and the driving band, have to be designed so that stresses that appear during the firing process do not produce neither an unacceptable level of deformation nor an explosion in the barrel or on the trajectory. In order to perform the safety analysis, it is necessary to determine the state of stress in the projectile body. The analytical calculation methods developed earlier [1, 2, 3], are unable to satisfy calculation accuracy. This article, based on a research made in the Military Technological Institute in Belgrade [4], considers the application of the FEM in order to simulate the stress state in the body of an artillery projectile. As an example, the 105 mm HE projectile M1 is used.

Load and boundary conditions

Fig.1 shows the model used for calculations.

On the outer surface of the projectile, behind the driving band a static pressure of p = 300 MPa is applied. This pressure simulates the pressure of the propellant gases. On the frontal surface of the projectile a static pressure of p_u = 178,8 MPa is applied simulating the existence of the fuse. In the direction of the axis of symmetry a value of angular acceleration of $d^2\omega/dt^2$ = 461097 s^{-2} is applied. An axial force component of F_{na}=71280 N is applied on the cylindrical surface of the driving band.

The kinematic degree of freedom in the axial direction is assigned to the central point on the bottom of the shell. Movement restraint in the radial direction is assigned to the cylindrical surface of the fuse opening. The presence of the fuse and the movement restraint of the shell as a rigid body (except in the axial direction) is thus simulated. The outer cylindrical surface of the driving band is restrained from movement in the tangent direction, which simulates the transition of the torque on the driving band. This model assumes an ideal position of the shell inside the barrel, i.e.

1) Military Technical Institute (VTI), Ratka Resanovića 1, 11132 Belgrade, SERBIA

the interraction between the barrel and the shell body is neglected. It can be noticed that the influence of the barrel upon the driving band is also neglected – the cylindrical surface of the driving band is free to move in the radial direction. At first, this assumption may seem illogical, although there is a physical explanation. During the process of cutting in and transformation of the driving band, it assumes the shape of the inner wall of the barrel. This process is characterized by pressing out and flowing of the driving band material. There is a groove behind the driving band the purpose of which is to channel material in excess. During the further motion of the shell inside the barrel, the increase of the driving band diameter due to the shell body deformation is followed by the removal of layers of the driving band material by abrasion. As a result of the previous analysis, it is acceptable to say that the contact surface between the inner wall of the barrel and the driving band is not subjected to pressure during the shell movement through the barrel [5].

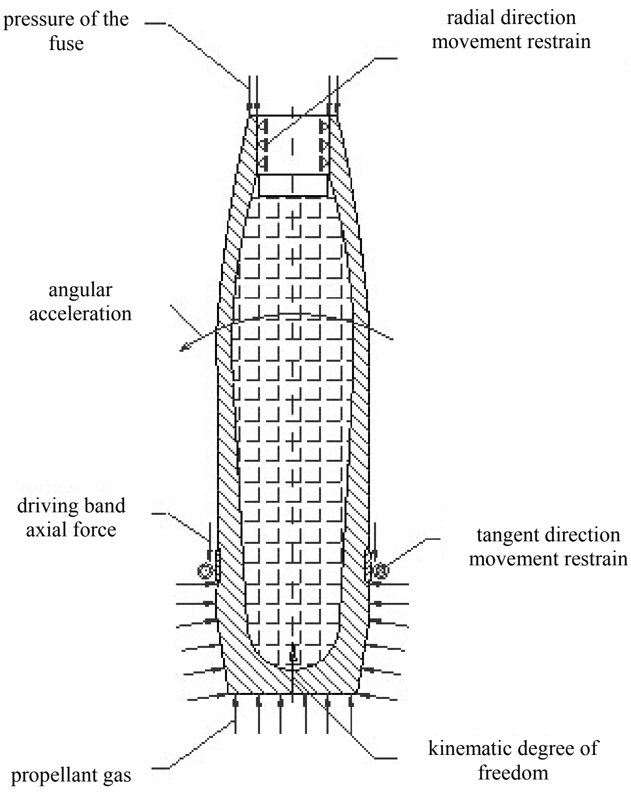

Figure 1. Load and boundary conditions

Mesh quality indicators

In addition to the correct assignment of loads and boundary conditions, the creation of an appropriate mesh of finite elements is also of utmost importance for computation. It is well known that a result of computation considerably depends on the mesh density, i.e. on the size of the used elements. In principle, the result will converge to the exact solution with the increase in number of mesh elements [6].

Distortion and stretch of finite elements are very important mesh quality indicators, and they are very significant in regard of computation results. The definition and the meaning of these concepts are taken from their formulation in the I-DEAS software [7].

For a 3D finite element in the shape of a hexahedron the stretch is defined as:

$$\text{Stretch} = \left(L_{\min}/L_{\max}\right)\cdot\sqrt{2}\,, \tag{1}$$

where:

L_{min} – FE smallest diagonal,

L_{max} – FE largest diagonal.

The stretch is defined in a similar way for a 2D element in the shape of a quadrangle (Fig. 2):

$$\text{Stretch} = \left(L_{\min}/L_{\max}\right)\cdot\sqrt{3}\,. \tag{2}$$

For a 2D triangular element, this parameter is calculated according to the expression:

$$\text{Stretch} = \left(R/L_{\max}\right)\cdot\sqrt{12}\,, \tag{3}$$

where:

R – radius of the inscribed circle of the triangle,

L_{max} – longest side of the triangle

For the definition of the distortion concept of the transformation, the Jacobian is used for translating elements from the global coordinate system to the local one:

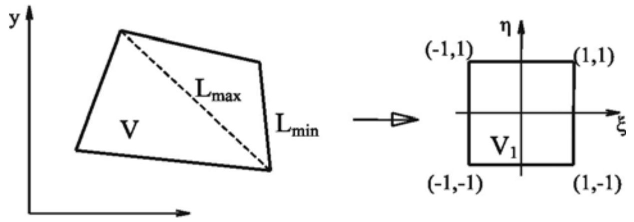

Figure 2. Stretch and distortion of a quadrilateral finite element

$$\text{Distortion} = \left[\left(\det J\right)_{\min}/V\right]\cdot V_1\,, \tag{4}$$

where:

$(\det J)$ – determinant of the Jacobian,

V – finite element volume

V_1 – finite element volume translated to the local coordinate system.

In the case of a 2D finite element, the finite element surfaces figure instead of its volumes (Fig.2).

In an ideal case, the stretch and the distortion of all elements would be equal to one. In practice, this is very hard to achieve, i.e. there is no exact criterion which defines acceptable boundaries of the stretch and the distortion of finite elements.

Computation using three-dimensional finite elements

A model of finite elements based on curved tetrahedral elements with ten nods (Model a) is shown in Fig.3. The nominal size of the element (element edge length) is 5mm. It was shown that this size of elements is optimal from the aspect of calculation time.

The stress condition based on this model is shown in Fig.4. The maximum value of von Mises stresses $\sigma_{eq} = 653\,\text{N/mm}^2$ is obtained. In the following part of this article it can be seen that by accepting this value, a significant error would be introduced.

The elements in which stress significantly differs from the surrounding elements can be noticed by a more detailed analysis (Fig.5). The finite elements distortion field is shown in Fig.6. The elements the distortion of which is less than 0.65 are represented in green. It can be noticed that the highest stress values occur in these elements.

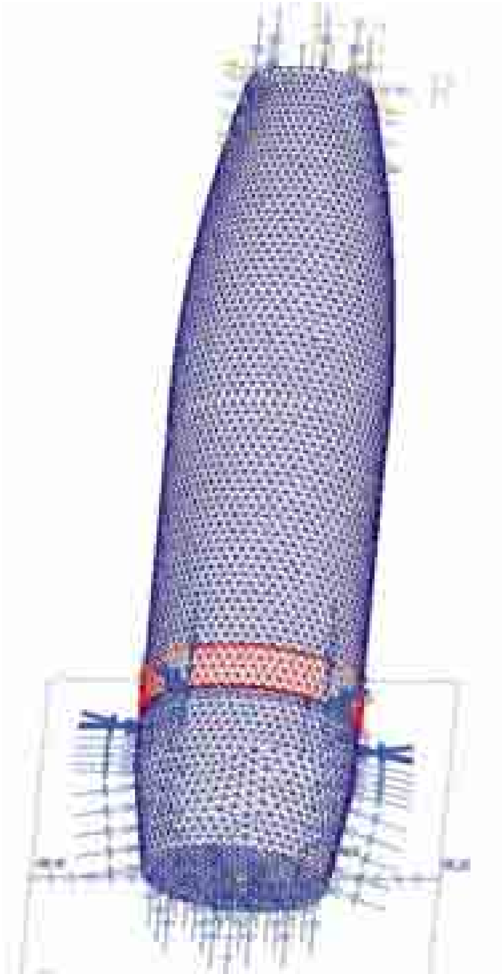

Figure 3. Model *a* of finite elements - curved tetrahedral elements -

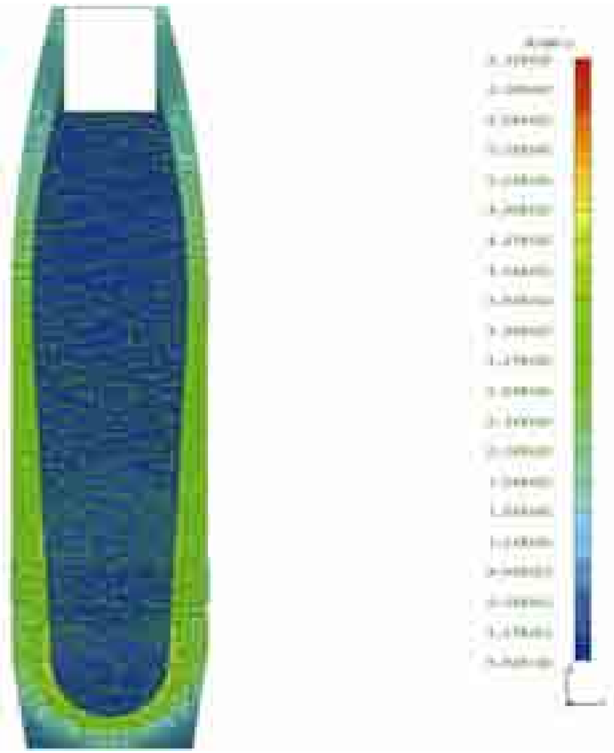

Figure 4. Stress condition of Model *a*

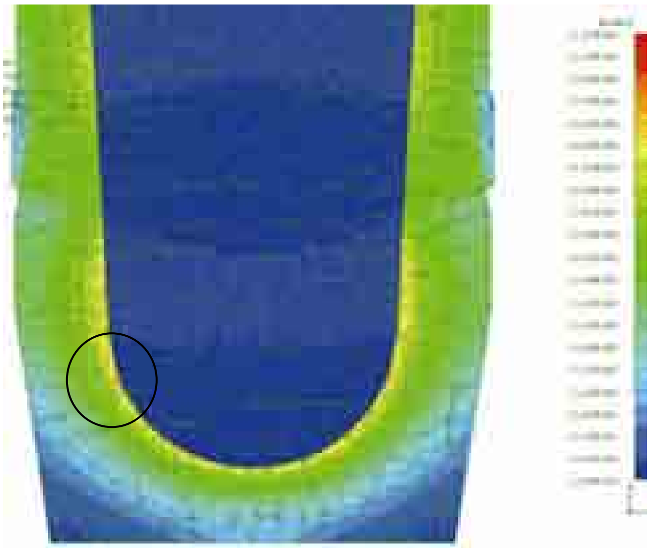

Figure 5. Stress distribution of Model *a*

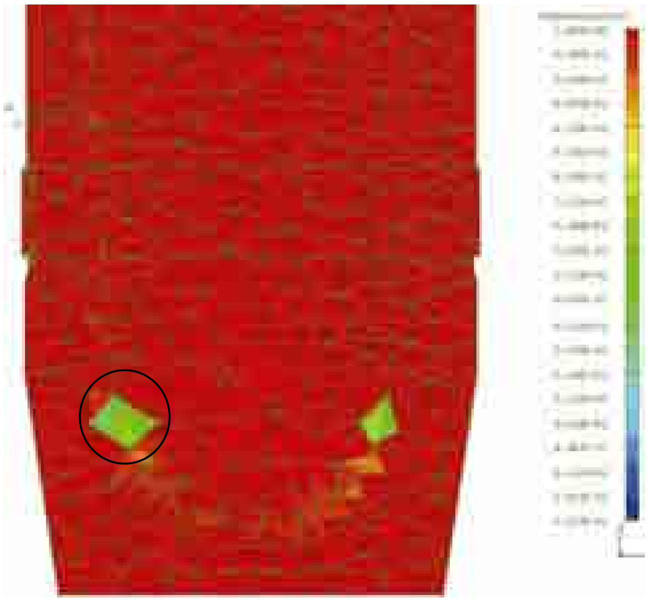

Figure 6. Distortion field of the Model *a* of finite elements

The previous proves that the obtained results do not represent the actual stress distribution well and that they cannot be taken as acceptable. Therefore, it is necessary to carry out certain modifications of the finite elements mesh in order to improve its quality, in the particular case, to increase the distortion of elements. The easiest way to achieve this is by using rectilinear finite elements. A model based on such finite elements is shown in Fig.7 (Model *b*). Visually, it differs very little from the previous model. The distortion field of these elements is given in Fig.8. It can be seen that the distortion value of all elements in this model is higher than 0.95, thus making this mesh almost ideal from this aspect.

The stress distribution obtained by this model is shown in Fig.9. By comparing the stress state of Model *a* (Fig.5) and Model *b* (Fig.8), a significant difference in the maximum stress values can be noticed (560 N/mm^2 compared to the previous value of 653 N/mm^2). Also, Model *b* stress distribution is sufficiently “smooth”, which also confirms a good quality of the mesh regarding the size of finite elements, as well as their distortion and stretch.

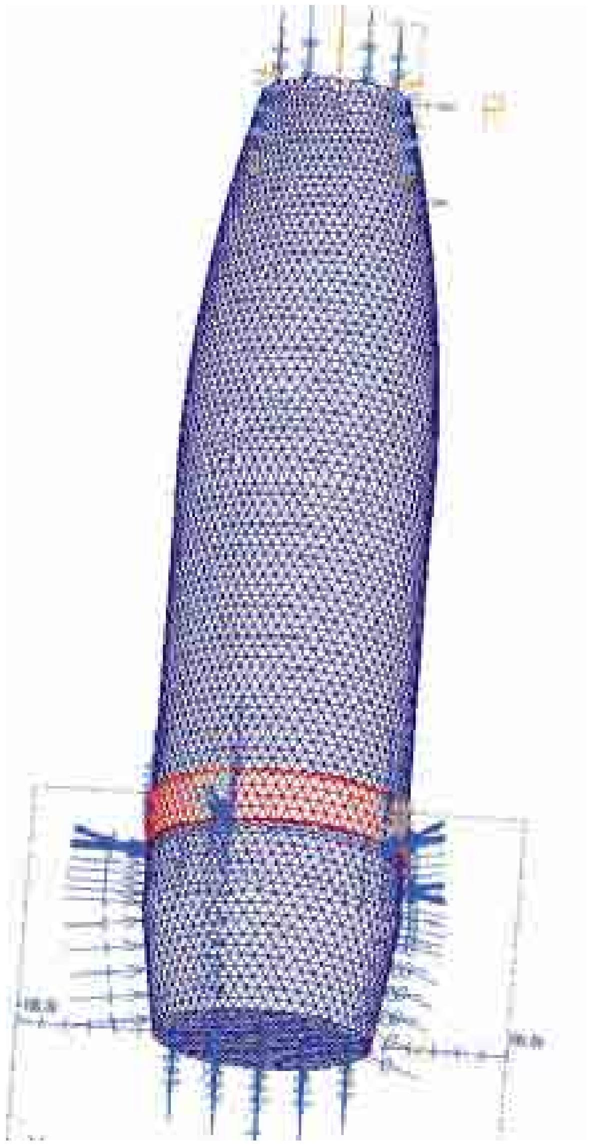

Figure 7. Model *b* of finite elements - rectilinear tetrahedral elements -

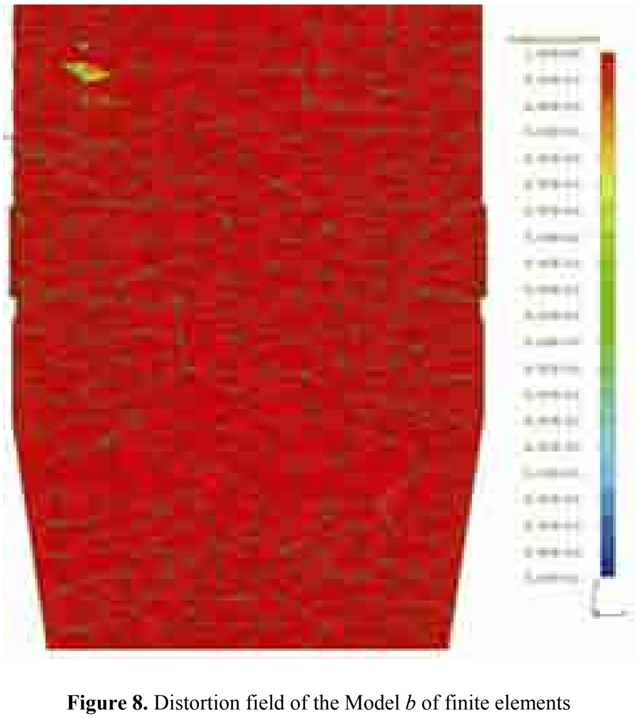

Figure 8. Distortion field of the Model *b* of finite elements

Computation with axisymmetrical finite elements

Model *b* of finite elements was used to examine the influence of the torque, due to the angularly accelerated rotation of the shell around its axis of symmetry. The stress distribution obtained after neglecting the shell rotational acceleration values in the Model is shown in Fig.10.

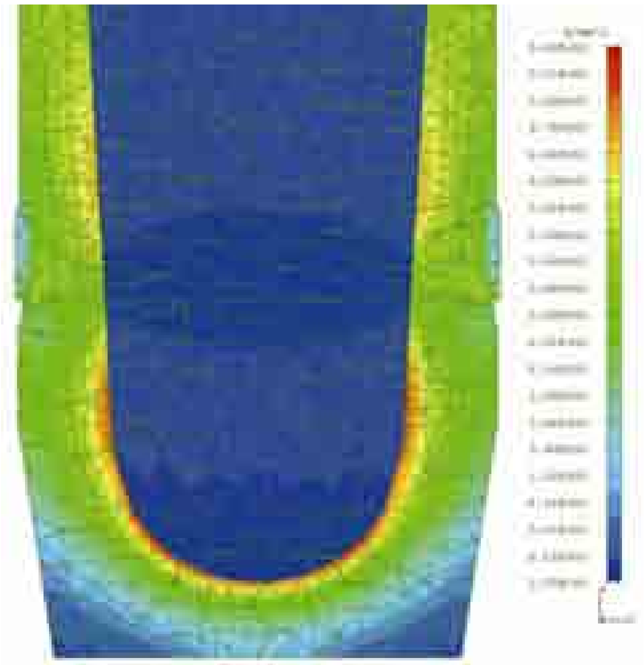

Figure 9. Model *b* stress distribution

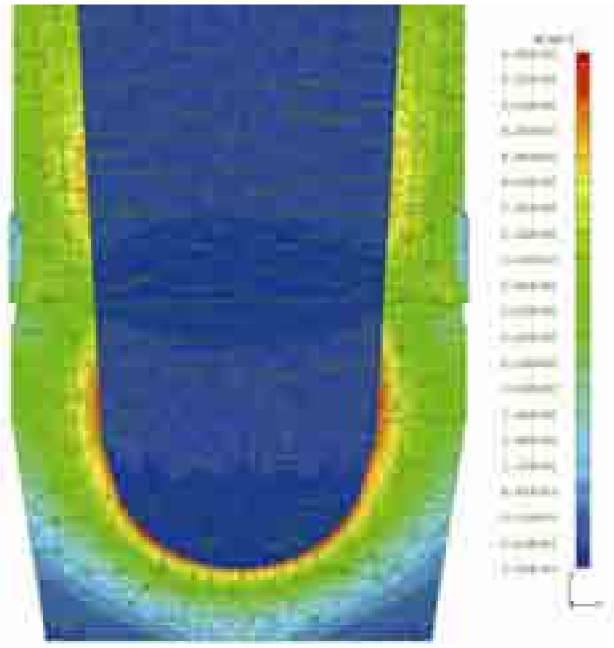

Figure 10. Model *b* stress distribution obtained by neglecting the torque

As it can be seen, in the case of this projectile the torque has almost no influence on the stress distribution. By neglecting the twisting, the problem is reduced to an axisymmetrical one, which creates the conditions for the application of axisymmetrical finite elements. A model based on the axisymmetrical triangular elements with quadratic interpolation and a nominal size of elements of 3mm (Model *c*) is shown in Fig.11.

The stress distribution obtained by this model is shown in Fig.12. A comparison of this stress distribution with the stress distribution from Fig.9, shows minimal differences. On the other hand, the advantages of using axisymmetrical finite elements, especially from the aspect of time required for computation, are significant.

As it can be seen, the values of the stresses inside the shell body in the moment of the maximum pressure of propellant gases are higher than the value of the upper yield strength of the shell body material, which is R_{eH} = 490 N/mm^2. It does not necessarilly mean that the safety of the projectile during its travel through the barrel is endangered, and there are two reasons for this. Firstly, maximum stresses occur in a very narrow area on the inner side of the shell body, thus provoking no significant plastic deformation. Secondly, the dynamic character of the load acting upon a projectile should be taken into consideration. The projectile travel through the barrel lasts for 9 ms, with the maximum pressure of propellant gases reached after only 3 ms. It is well known that for the deformation rates like these there is some increase in steel yield strength [8]. It should be pointed out that this phenomenon largely depends on physical properties of materials, their chemical composition, heat treatment, etc.

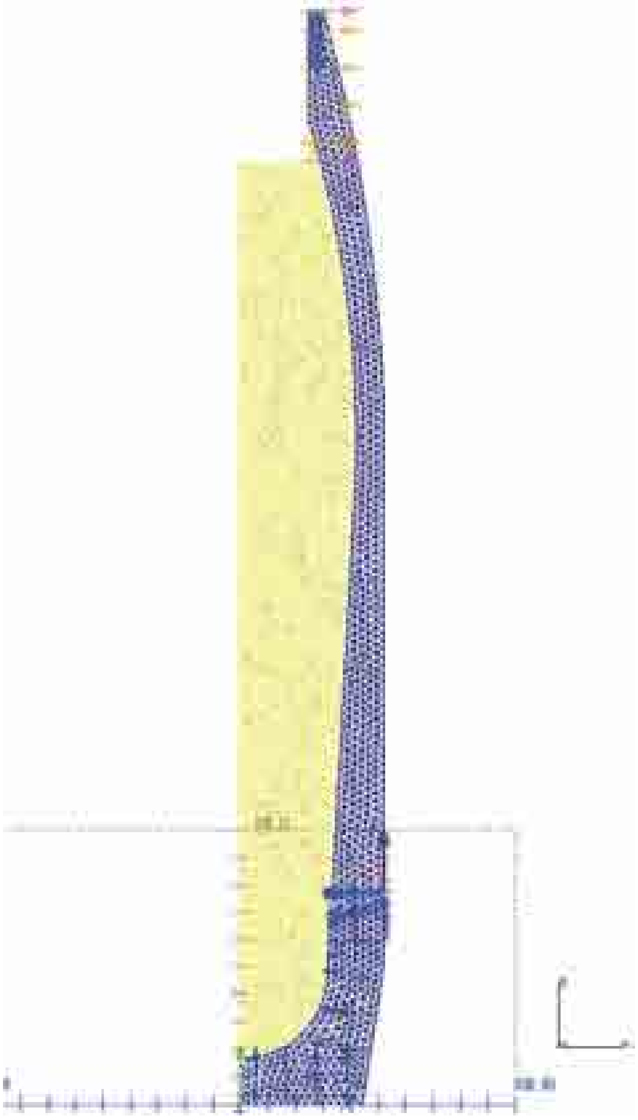

Figure 11. Model *c* of finite elements - axisymmetrical elements –

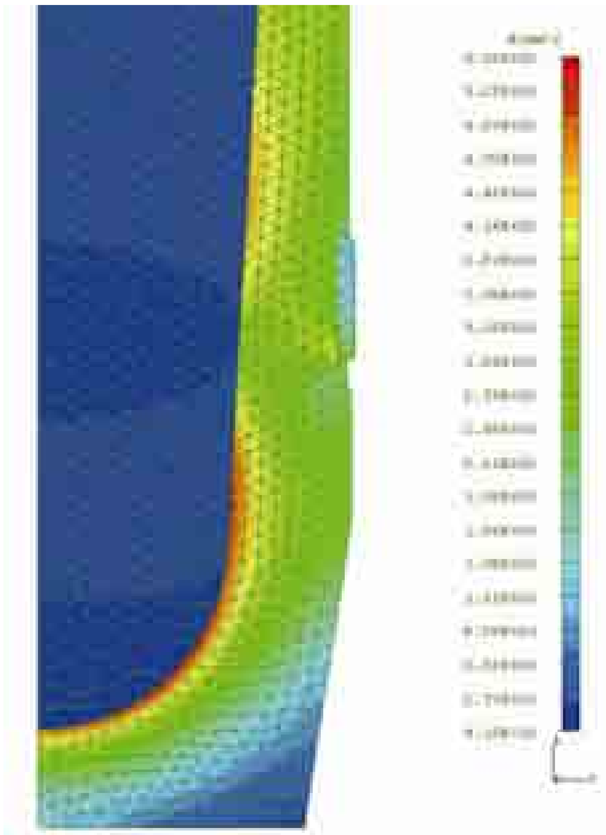

Figure 12. Model *c* stress distribution

Conclusion

It was shown that the FEM can be applied in determining stress distribution inside the body of an artillery projectile during its travel through the barrel. Giving the nature of this phenomenon, real values of stresses inside the shell body are not possible to be measured, so the FEM results are taken as referent values. A special care was taken to minimise the computation errors during the FEM application. The quality of the FEM mesh proved to be very significant for the computation. This is especially true for three-dimensional finite elements, where too low values of distortion could cause unrealistically high values of stress. The application of an inadequate mesh resulted in a 20% higher value of maximal stress, when compared to the adopted referent value. When neglecting the influence of the shell body twisting on the stress distribution, as in this case, it is acceptable to use axisymmetrical finite elements. These elements allow for considerable saving of computation time. Another advantage of these elements is smaller influence of the mesh quality indicators on computation precision.

Another important conclusion of this paper is that the degree of safety based on the upper yield strength of the shell body material is not an authoritative criterion for evaluating the safety of an artillery projectile during its travel through the barrel. Local plastic deformations in a narrow area of the shell body do not cause any significant residual plastic deformations of the shell body after the firing [4][1]. For a more detailed analysis of this problem it is necessary to use methods which would enable an analysis of stress and deformation of the shell body in dynamic

[1] Stress-strain state of the shell body is studied in detail in this paper.

conditions of material plastic yielding, as well as an experimental verification of computational results. By using a commercially available software it is possible to define, with a satisfying precision, the critical areas on the shell body and to evaluate its reliability during the travel of the projectile through the barrel.

Literature

[1] SUTTERLIN,R.: *Cours de munitions,* Ecole nationale superieure de l'armament, Paris, 1963-1966

[2] STAMATOVIĆ,A.: *Konstruisanje projektila*, Ivexy p.o., Beograd, 1995

[3] MOMČILOVIĆ,V.: *Sigurnost projektila pri opaljenju i u toku kretanja u cevi oruđa*, VTI 02-01-0067, Vojnotehnički institut, Beograd, 1987

[4] ĐURĐEVAC,D.: *Analiza naponskog stanja košuljice artiljerijskog projektila - magistarski rad*, Mašinski fakultet Univerziteta u Beogradu, Beograd, 2008

[5] LISOV,M.: *Modeling wear mechanism of artillery projectiles rotating band using variable parametars of internal ballistic process*, Scientific Technical Review, Belgrade, 2006, Vol.56, No.2, pp.11-17,

[6] SEKULOVIĆ,M., *Metod konačnih elemenata*, IRO Građevinska knjiga, Beograd, 1988

[7] LAWRY,M., *I-DEAS Master Series - student guide*, Structural Dynamics Research Corporation, Milford, USA, 1997

[8] ОРЛЕНКО,Л.П., *Поведение материалов при интенсивных динамических нагрузках*, Издательство "Машиностроение", Москва, 1964

Received: 16.01.2008.

Mogućnost primene metode konačnih elemenata za analizu naponskog stanja košuljice razornog artiljerijskog projektila

Ovaj rad obuhvata teorijsku analizu strukture i raspodele napona koji se javljaju unutar košuljice artiljerijskog razornog projektila u fazi njegovog kretanja kroz cev oruđa. Razmotrena je mogućnost primene trodimenzionalnih i osnosimetričnih konačnih elemenata. Naročita pažnja je poklonjena analizi kvaliteta mreže konačnih elemenata i njegovom uticaju na tačnost proračuna. Proračun je izvršen na primeru projektila 105 mm TF M1. Za proračun je iskorišćen programski paket I-DEAS.

Ključne reči: **artiljerijski projektil, razorni projektil, košuljica projektila, kalibar 105 mm, naponsko stanje, analiza napona, strukturalna analiza, metoda konačnih elemenata.**

Возможность применения метода конечных элементов для анализа состояния напряжения рубашки (тела) артиллерийского снаряда

Настояшчая работа охватывает теоретический анализ структуры и распределения напряжений, являющихся внутри рубашки (тела) артиллерийского разорительного снаряда в фазе его движения через нарезной ствол артиллерийского орудия. Здесь рассматривана и возможность применения трехразмерных и осевосимметрических конечных элементов. Особое внимание уделено анализу качества сети конечных элементов и его влиянию на точность расчета. Расчет сделан на примере снаряда калибра 105 мм TFM1, а для расчета использована упаковка программного обеспечения I-DEAS.

Ключевые слова: **артиллерийский снаряд, разорительний снаряд, рубашка (тело) снаряда, калибр 105 мм, состояние напряжения, анализ напряжения, анализ структуры, метод конечных элементов.**

Possibilité d’application de la méthode des éléments finis pour analyser l’état de tension chez le corps de cartouche du projectile d’artillerie

Ce papier comprend l’analyse théorique de la structure et la distribution de tension qui apparaissent à l’intérieur du corps de cartouche chez le projectile d’artillerie puissant pendant son mouvement dans le canon. On a considéré la possibilité d’application des éléments finis à trois dimensions et symétriques axialement. Une attention particulière a été donnée à l’analyse de la qualité du réseau des éléments finis et de son influence à la précision du calcul. Le calcul a été fait chez le projectile de 105 mm TF M1. Le progiciel I-DEAS a été employé pour faire ce calcul.

Mots clés: **projectile d’artillerie, projectile puissant, corps de cartouche, calibre 105 mm, état de tension, analyse de la tension, analyse structurale, méthode des éléments finis.**