ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XI – SHIPBUILDING. ISSN 1221-4620
2015

NUMERICAL SIMULATION OF A PROJECTILE THAT RUNS THROUGH A PLANE STRUCTURE STRENGHTENED WITH ONE-WAY STIFFENERS

Mădălina Prodan
"Dunărea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:madalinaprodan_mp@yahoo.com

Vlăduţ-Alexandru Vicol
"Dunărea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:vladutalexandruvicol@yahoo.com

Costel Iulian Mocanu
"Dunărea de Jos" University of Galati,
Faculty of Naval Architecture, Galati,
47 Domnească Street, 800008, Romania,
E-mail:costel.mocanu@ugal.ro

ABSTRACT

Because of the naval incidents in recent years resulting in loss of human lives and materials, an outstanding importance is now given to the study of phenomena which occur in ship structures upon impact with other structures or in the case of military applications with projectiles. The objective of the present work is to study the impact loading of a projectile on steel plane plates using FEA (Finite Element Analysis) software ABAQUS.

Keywords: FEM, impact, Johnson-Cook.

1. INTRODUCTION

In shipbuilding, numerical analyses are essential to determine the structural resistance of the vessel. The main purpose is to secure safety at sea. According to the maritime accidents statistics, the collisions between ships and/or one ship and shore represent the highest percentage. Therefore, designing new structures that can ensure the safety of the vessel in these cases becomes one of the research themes. The problem of impact is extremely complex.

There are many situations where collision may occur: the impact with the quay or an iceberg, two ships hitting or armour striking of navy vessels. Thus, realistic simulations of the phenomena are required.

The case of impact involves highly non - linear structural deformations until fracture, elastic and plastic instability, material behaviour under high strain rates and the failure criterion which addresses the failing elements.

2. THEORETICAL BACKGROUND

The Finite Element Method can capture very well the material behaviour as far as the point of necking. Most of the time, beyond the onset of necking, results cannot describe the real phenomenon.

The complete control of material characterization is very important for numerical methods. Since the experimental tests are expensive, simulation methods were performed in order to facilitate the study of these problems.

The Johnson-Cook model [1], [2] is a viscous-plastic model for ductile metals that consider the strain rate effects on material behaviour and fracture. This model is frequently used because its parameters can be easily found in open literature for different materials.

A particular situation for dynamic applications is known as "Ductile Plastic Damage". Figure 1 depicts the damage and its effect on plastic instability (negative stiffness).

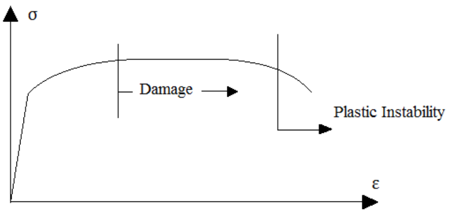

Fig.1. Damage and plastic instability

Johnson and Cook express the equivalent stress as a function of plastic strain, strain rate and temperature. In this way, it was easier to determine the associated parameters. The relation for the flow stress defined by Johnson and Cook is empirical and it is represented as:

$$\sigma_{flow} = \left[A + B\varepsilon^n\right]\left[1 + C\ln\dot{\varepsilon}^*\right]\left[1 - \left(T^*\right)^m\right]$$

where A, B, C, n and m are the material parameters measured at or below the transition temperature T^*, ε is the plastic strain, $\dot{\varepsilon}^* = \dot{\varepsilon}/\dot{\varepsilon}_0$ is the dimensionless plastic strain rate for $\dot{\varepsilon}_0 = 1s^{-1}$, and the non-dimensional temperature T^* is defined as:

$$T^* = \frac{T - T_{room}}{T_{melt} - T_{room}} .$$

The Johnson-Cook damage model is a cumulative-damage fracture model that takes into account the loading history, which is represented by the strain to fracture. The strain to fracture is expressed as a function of the strain rate, temperature, and pressure. It is also an instantaneous failure model, which means that no strength or stiffness remains after erosion (failure) of an element, at least in tension. ([3])

The general expression for strain at fracture is given by:

$$\varepsilon^f = \left[D_1 + D_2 \exp D_3\sigma^*\right]\left[1 + D_4 \ln\dot{\varepsilon}^*\right]\left[1 + D_5 T^*\right]$$

where $D_i = \overline{1,5}$ is represented by damage constants of the model: D_1, D_2, D_3 are constants for the triaxial state of stress, D_4 is the strain parameter, D_5 the temperature parameter for constant values of the variables ($\sigma^*, \dot{\varepsilon}^*$ and T^*) and for $\sigma^* \leq 1.5$. The dimensionless pressure / stress ratio is a measure of triaxiality of the stress state and is defined as:

$$\sigma^* = \frac{\sigma_H}{\sigma_{eq}}$$

where σ_H is the hydrostatic stress,

$$\sigma_{eq} = \sqrt{\frac{2}{3}\sigma_{ij}^D\sigma_{ij}^D}$$

the von Mises equivalent stress. [3]

3. NUMERICAL SIMULATION

The main objective of this analysis is to study the impact loading of a projectile on steel plate using FEA (Finite Element Analysis) software Abaqus (explicit dynamic modulus) manufactured by the company Dassault Systems. An element deletion criterion is defined and, in this way, the projectile passes through the plate.

The plane structure represents the side shell of one vessel's midship section. It is strengthened with steel flat bar profiles. The main dimensions are presented in Table 1.

Table 1 Main dimensions

Plate			Profiles	Projectile	
Length [mm]	Breadth [mm]	Thickness [mm]	Type [mm] x [mm]	Length [mm]	Diameter [mm]
1400	1400	10	B 200x1	400	100

The 3D-CAD model is developed in Abaqus, using volume parts and it is shown in Figure 2. Creating mesh involves using three-dimensional continuum elements (hexahedral elements). They are also known as brick-shaped elements (eight corners, twelve edges, six faces). The elements mesh size of the model is 30 mm, but in the collision area, it is 15 mm. The 3D-FEM mesh detail of the model is shown in Figure 3.

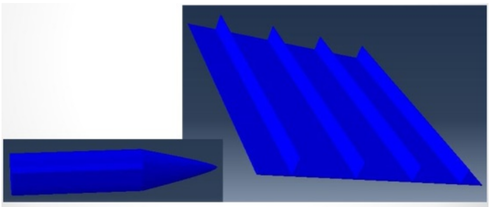

Fig.2. 3D-CAD model

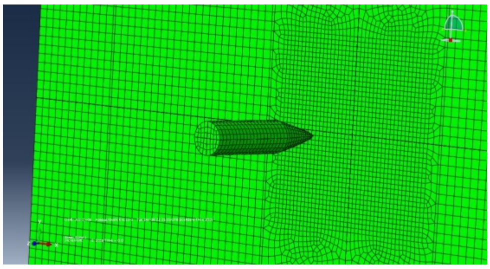

Fig.3. 3D-FEM mesh detail

The 3D-FEM plane structure has naval steel grade AISI 4340 and the projectile is considered to be a linear material with the modulus (E), Poisson's ratio (ν) and density (ρ) of steel (Table 2a):

Table 2a Material characteristics

	Steel (plate)	**Steel (projectile)**
E (MPa)	205000	202000
ν	0.3	0.3
ρ (t/mm^3)	7.85E-9	8.6E-9

Table 2b Johnson-Cook parameters

	Steel (plate)	**Steel (projectile)**
A (MPa)	792	1430
B (MPa)	510	2545
n	0.26	0.7
m	1.03	1.03
D_1	0.05	-
D_2	3.44	-
D_3	-2.12	-
D_4	0.002	-
D_5	0.61	-

In Table 2b, the Johnson - Cook parameters are also presented for both materials, which are initial yield (A), strain hardening (B), strain hardening exponent (n), thermal softening (m), constants for the triaxial state of stress (D_1, D_2, D_3), strain parameter (D_4), temperature parameter (D_5). The value of the displacement at failure is 0.9 mm.

Another stage which needs to be considered is to set the boundary conditions. The plate is fixed at the edges in all degrees of freedom. Moreover, the model has two planes of symmetry: XOZ, YOZ. The projectile is given an initial velocity: v_3 = 600000 mm/s.

The results are presented in the following frame rates:

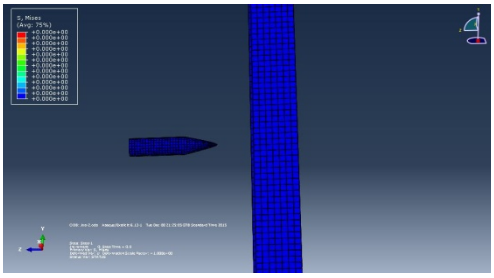

Fig.4.a t0

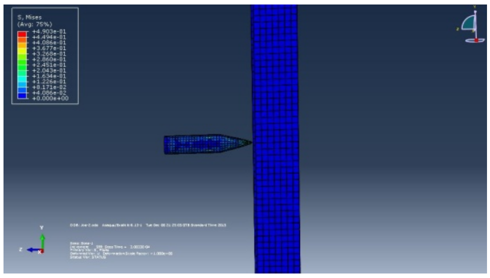

Fig.4.b t1

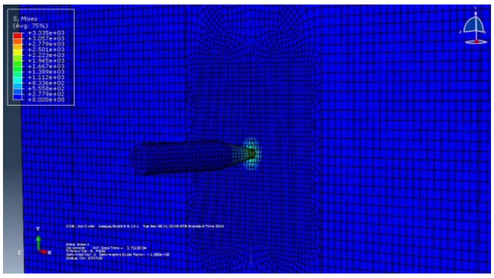

Fig.4.c. t2 (front)

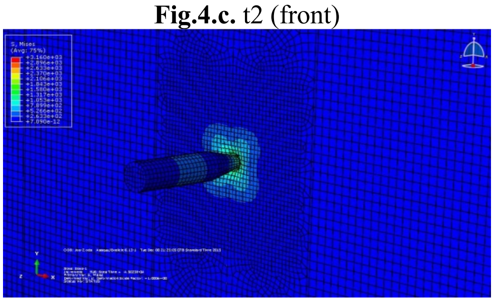

Fig.4.d. t3 (front)

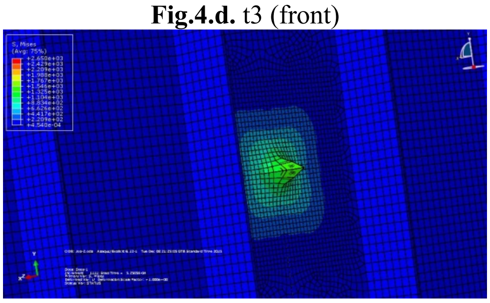

Fig.4.e. t4 (back)

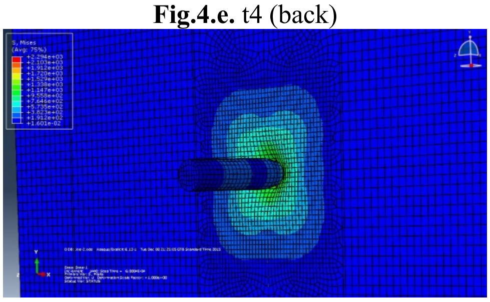

Fig.4.f. t5 (front)

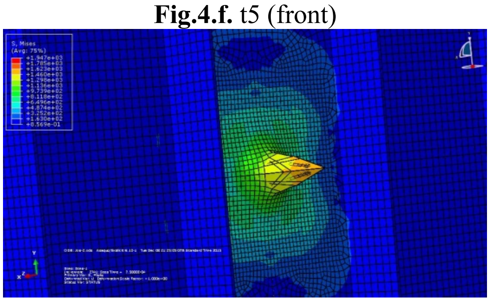

Fig.4.g. t6 (back)

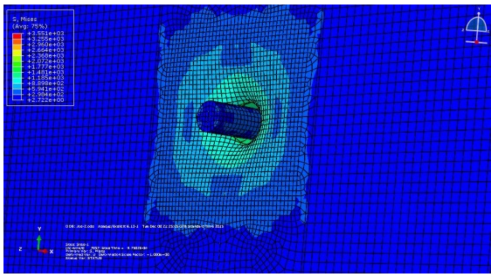

Fig.4.h. t7 (front)

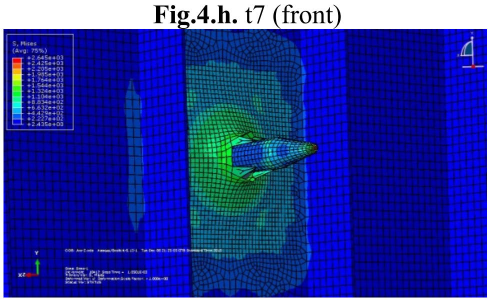

Fig.4.i t8 (back)

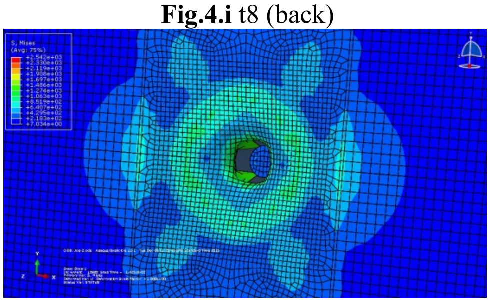

Fig.4.j t9 (front)

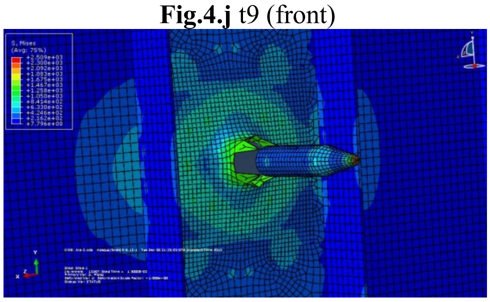

Fig.4.k t10 (back)

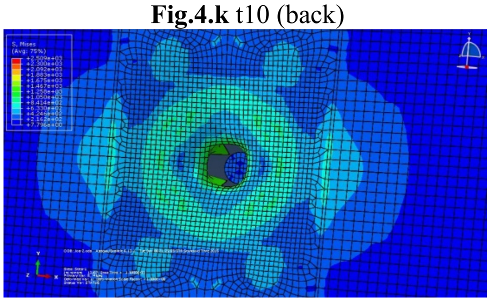

Fig.4.l t11 (front)

It is essential to have additional insights about the behaviour of the plate under this type of loading. The plate behaviour can be obtained by plotting the kinetic energy (it is present in the model as a result of the motion of the mass) and the strain energy (it is present as a result of the displacement of the structure).

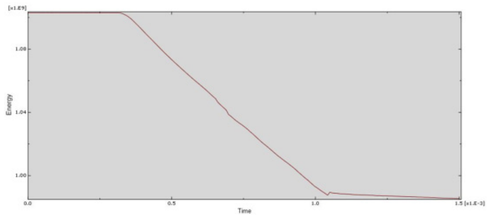

Fig. 5 Kinetic energy

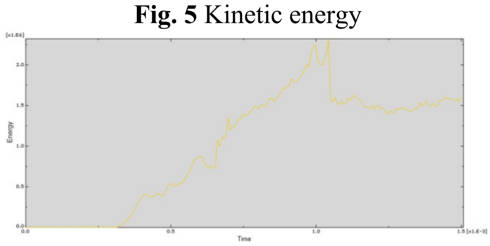

Fig. 6 Strain energy

In Figure 5 and 6 the kinetic energy and the strain energy are represented as functions of time for the whole model.

4. CONCLUSIONS

Based on the numerical results, we can see that the kinetic energy (Fig. 5) of projectile decreases and a part of it is transformed into internal (strain) energy. This decrease corresponds to the increase in strain energy (Fig. 6). Therefore, it can be concluded that the methodology designed for the study of the impact loading of a projectile on steel plate is correct.

This paper represents the beginning of the research process which aims to establish the methodologies for verifying the maritime structures and it does not only concern the impact case.

REFERENCES

[1]. **G.R. Johnson, W.H. Cook**, "*A constitutive model and data for metals subjected to large strains, high temperatures, in Proceedings of the seventh International Symposium on Ballistic"*, The Netherlands, The Hague, 1983;

[2]. **G.R. Johnson, W.H. Cook**, "*Fracture characteristics of three metals subjected to various strains, strain rates, temperatures and pressures"*, Engineering Fracture Mechanics Vol. 21, No. 1, 1985;

[3]. **Maxime Jutras**, "*Improvement of the characterisation method of the Johnson-Cook model"*, 2008;

[4]. ***Abaqus*** *6.12 Documentation*, Dassault Systèmes, 2012.

Paper received on December 31st, 2015