Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 74 (2014) 360 – 367

Procedia Engineering

www.elsevier.com/locate/procedia

XVII International Colloquium on Mechanical Fatigue of Metals (ICMFM17)

Numerical calculation of crack parameters for propagation assessment in a complex component with residual stresses

M. Fossati[a], M. Pagani[a], S. Miccoli[a], A. Manes[a]*, M. Giglio[a]

[a]*Politecnico di Milano, Dipartimento di Meccanica, via la Masa 1, Milano, Italy*

Abstract

Fatigue is an actual issue, especially in the structural assessment of highly critical components where failure may provoke the loss of the whole system they belong to. This is especially true in a damage tolerant scenario. In this case the use of a component is permitted even after damage has occurred, but extensive knowledge of the damage status and the effect of the damage on the reliability of the component is required. An experimental approach to damage tolerance is often very complex both for the damage creation and for the subsequent step of fatigue tests. A reliable and efficient methodology for fatigue simulation and crack propagation is therefore of interest both from a theoretical and an industrial point of view. The aim of the paper is to investigate crack parameters in a transmission shaft subjected to ballistic impact damage; cracks can nucleate from the point of damage and propagate due the application of service loads Starting from the outcome of a FEM simulation of a ballistic impact, different strategies for numerical crack propagation under service loads are presented and assessed. Specifically the damage under investigation is composed of two holes separated by a septum. Experiments reveal that the crack that nucleates from one of the two holes is strongly influenced by residual stresses. The complexity and the variety of the phenomena involved (complex damage, presence of residual stresses, shape of the component) make the numerical simulation of the phenomenon not trivial and give grounds for a thorough investigation of different modelling approaches.

Selection and peer-review under responsibility of the Politecnico di Milano, Dipartimento di Meccanica

Keywords: ballistic damage; fatigue crack, XFEM, numerical simulation

* Corresponding author. Tel.:+39 0223998630 ; fax:+39 02 23998263.
E-mail address: andrea.manes@polimi.it

Selection and peer-review under responsibility of the Politecnico di Milano, Dipartimento di Meccanica
doi:10.1016/j.proeng.2014.06.280

1. Introduction

Nowadays projectile impacts play an increasingly important role in the design of modern military aeronautic frames and systems. For instance a low altitude flight, typical of a helicopter flight envelope, can be critical for the structural integrity of the flying system due to the extensive exposure to ballistic threat. It is therefore crucial to evaluate the damage of highly critical components after a ballistic impact and guarantee the survivability of the system, even in the worst condition. Regarding helicopters, the tail rotor power transmission has two important and critical features:

- it is a fundamental part in ensuring a flight: a significant damage of the tail rotor system can lead to a fast and drastic reduction in the control and stability of the machine, i.e. a catastrophic loss of the machine and probably also of the crew;
- the tail rotor transmission, and in particular the shaft, extents over the whole tail boom, and is therefore very exposed especially during hovering manoeuvres.

The last point addresses an unavoidable feature of typical helicopter flight mechanics. Therefore, research to investigate the ballistic impact and the subsequent residual structural capability of such components is clearly required.

At present, experimental tests have a primary role in improving the understanding of impact events and in verifying the residual strength of the damaged components after being hit by a real bullet. However a numerical method for the reproduction of the whole procedure is of interest for several aspects. With this aim numerical models of the impact stage have been developed and validated [1]. Experimental programmes of ballistic impact have been carried out considering a critical condition for the component (tangential hit with a 45° angle of obliquity). Such peculiar tests have been reproduced by means of Finite Element Models with a focus on the mechanical behaviour of the involved material and the numerical features of the analyses (mesh type and dimension, contact, erosion, etc). Such models have been validated with respect to experimental data both with macroscopical results (residual velocity, type of the damage, extension of the damage) and by the use of micromechanical features (residual stresses). A preliminary investigation on the fatigue simulation stage has been carried out exploiting the models obtained from the impact test stage. The capability to use the results obtained from the impact simulation in the subsequent step of crack propagation is a key feature that ensures reliable results. This stage has been performed also experimentally using a multiaxial testing device and the comparison between the experimental propagation and the results obtained by numerical models shows encouraging results [2]. It's worth mentioning that crack propagations have to be simulated in a very complex pattern that includes the shape of the impact damage (with petalling features) and an articulated pattern of residual stresses. However, such numerical methodology showed several drawbacks mainly related to the use of classical remeshing techniques in order to obtain the fracture parameters (Stress Intensity factors, SIF). Once the SIF values have been obtained they are processed in the NASGRO propagation law [3] in order to obtain the propagation of the crack versus the cycles.

At present a quite novel approach has been developed to overcome the remeshing technique issue: the extended finite element method [4-6]. Exploiting this method the crack growth can be modelled directly because the crack is implicitly considered in the approximation of the displacement field. This is possible thanks to the enrichment of the classical function used inside the numerical solver for the displacement calculation thus without a modification of the topology of the mesh. XFEM has been recently used for crack propagation also in presence of fatigue load and residual stress [7-9] showing a good agreement with experimental relieves. XFEM methodology has herein been applied to the model of the impacted shaft. In section 2 a very brief description of the shaft model is reported with the focus on the impact stage; in section 3 the model used for propagation is described as well as the result obtained. The conclusions are reported in section 4.

2. Numerical modelling of the impact stage

Numerical simulations of the impact of the projectiles were carried out using Abaqus/Explicit with a 3D simulation of the impact. The impact configuration is almost tangential, 45° degree oriented, see Fig 1a. The numerical modelling approach is described in [1] although in this case the bullet is slightly different: a 12,7 mm calibre with a brass jacket and a rigid core has been considered in the present work. Focusing on the shaft, different dimensions of the element were considered. A dense mesh was used in the impact area and a sparse mesh was used far from it. A mesh refinement analysis was done in order to obtain the correct dimension of the element [1]. The constraint was made by an "encastré" boundary condition applied to the end surfaces of the shaft. The same constraint was used during the ballistic experimental tests.

No friction was considered between the projectile and the shaft surfaces. The shafts are made of Al 6061-T6. The Johnson-Cook constitutive relation has been chosen to represent the plastic behaviour of the material in the numerical model. The temperature increment due to adiabatic heating is calculated during the analyses using a Taylor-Quinney coefficient of 0.9 (proportion of plastic work converted into heat). As far as the failure criterion is concerned, the Bao-Wierzbicki (B-W) [10] criterion has been selected [1]. Brass jacket of the bullet has been modelled as deformable using data from [11]. As already showed in [1], results are very good both in terms of damage shape, as shown in Figure 1b, residual velocity (less than 1% of error) and residual stress pattern.

.

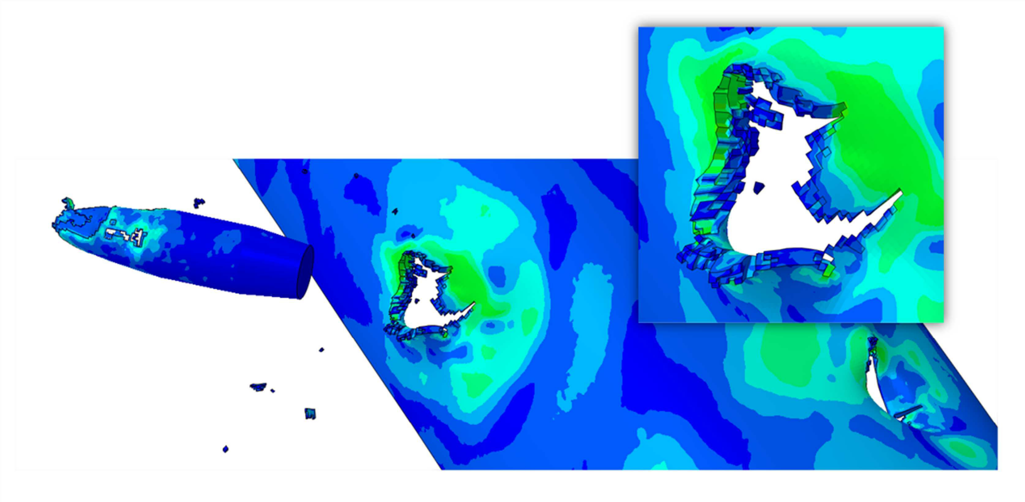

Fig. 1 – (a) Double-hole impact configuration: numerical entry and exit holes; enlargement of the exit hole

3. Numerical modelling of the fatigue propagation stage

For the crack propagation analysis step, the undamaged and not completely failed elements were exported from the last step of the impact explicit simulation. Position, node displacements and internal stresses were imported in the Abaqus/standard environment; during this process, all the failed and excessively distorted elements were eliminated from the parts, while the bullet was completely neglected. The final model for the standard analysis consists in an orphan mesh part, reproducing exactly the deformed mesh structure of the previous step. Due to the extremely distorted edge of the resulting damage holes (Fig 1), very few highly distorted elements, located mainly around the borders, were deleted. It's worth underlining that residual stresses were imported as the initial state of the new analysis

According to the experiments, two oblique cracks are expected to nucleate around the exit hole; from an FE point, when dealing with a crack propagation problem, the points of crack initiation and the direction of the crack propagation have to be addressed. The double cracks, which affect each other in terms of propagation velocity speed, make this task more challenging. The standard FE approach to the problem consists in the identification of the starting point generally by means of a stress analysis performed on the un-cracked specimen. Subsequently the mesh in the desired zone must be elaborated in order to take the discontinuity due to presence of the crack into account. The stress intensity factors and advancing direction are therefore calculated step by step. The use of automatic remeshing algorithms has grown in the last decade but the peculiar element conformation around the tip and, specifically for this work, the passage from the Abaqus/explicit model to the Abaqus/standard, creates some issues in the use of such techniques. Addressing these issues, two approaches have been considered in this work: a standard approach with a manual step-by-step crack opening and an XFEM approach. The results can be assessed by comparing the advantages of one approach versus the other.

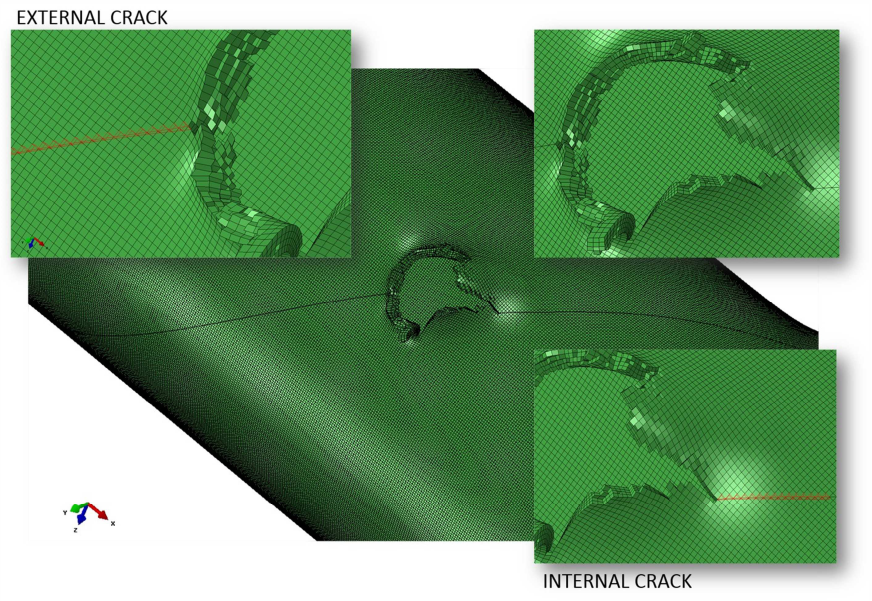

Fig. 2 – Crack path generated on the standard model by splitting two lines of hexahedral elements across the diagonal. Wedge elements are obtained.

3.1. Crack propagation: standard approach

As mentioned above, the usual approach to the simulation is an Abaqus/standard simulation in which the damaged tube is imported from the previous analysis and the residual stresses field is subsequently imposed at the start of the computation, in a dedicated relaxation step. The elements used are almost the hexahedral elements with reduced integration. However elements along the crack propagation paths were cut across the diagonal, forming couples of wedge elements (Fig 2). Using this technique, two 45° angled (respect to the axis of the tube) crack paths were created around the exit hole (in agreement with the experiments) and a tie interaction between the two sides of the cracks were used in order to efficiently run several analyses with cracks at the desired lengths. A 45° degree angle was chosen because experimental relieves showed that cracks follow this direction. However, a more complex approach should be considered calculating the propagation direction at each step and creating a dedicated mesh. This approach is very time consuming requiring several efforts for manual remeshing. Finally, one side of the tube is fully constrained while, on the other side, only the axial displacement and the axial rotation are unconstrained and the torsional load is applied. At the end of each calculation, a special sub-model, assembled with focused mesh elements around the center of the crack tip, following equally spaced round contours, is used to acquire the SIF factors for the desired crack length and for each load case. The material used in the global model is identical to the impact analysis JC material while, for the sub-model, a simpler linear material is sufficient. After the simulations, the SIFs can be obtained by using a displacement spline interpolation. Considering the dissymmetry of the problem (mutual interfering cracks), a set of different crack lengths was tested. A predefined set of discrete crack lengths are proposed for each of the two cracks across the exit hole, and, consequently, all the combinations between them are processed filling a matrix. Using a MATLAB routine, the SIF values (in terms of J-integral, KI, KII and KIII) in function of the length of the two cracks can be obtained.

3.2. Crack propagation: XFEM

The XFEM model is set up from the Abaqus/explicit impact model in the same way as the standard model using exactly the same elements imported from the previous analysis for each part of the tube. However some major differences between the two models must be pointed out: firstly the initial explicit simulation was done using a meshed tube modeled with only two elements in the thickness (Fig. 3) instead of the previously used 4 elements. This modification became necessary to reduce the computational time of the XFEM analysis. However, all of the other variables are the same and finally the damage shape is almost equivalent. The second difference is that, in the zones near the initial cracks and along the predicted propagation path, the elements are enriched with new degrees of freedom (XFEM). The added degrees of freedom permit a modification in the overall stiffness of the single element if crossed by the crack: the stiffness matrix of the element is modified differently for each direction, in order to consider the damage shape and length. Moreover, the software makes a calculation of the energy release rate, with respect to the load applied. The energy release is then used, providing a direct cycle step with an alternate load, along with the Paris law: thus the propagation speed of the crack is predicted. Depending on the load application modality, on the strength and on the frequency, the software should be able to calculate the crack propagation speed, the direction and, so, the final crack paths. The initial cracks were generated introducing one surface for each defect (Fig. 3), crossing the full thickness of the tube and cutting the enriched elements to the desired length. The torsional load is applied in the same way as the previous standard analysis, but, in this case, an alternate sinusoidal amplitude law is used to shift the loads from a minimum to a maximum in a unique step: a direct cyclic analysis method calculates the number of load cycles between two consecutive advancement of the cracks. Each advance means a new (fully automated) step with an updated geometry. Residual stresses were imported at the start of the simulation and an initial relaxation step was set up.

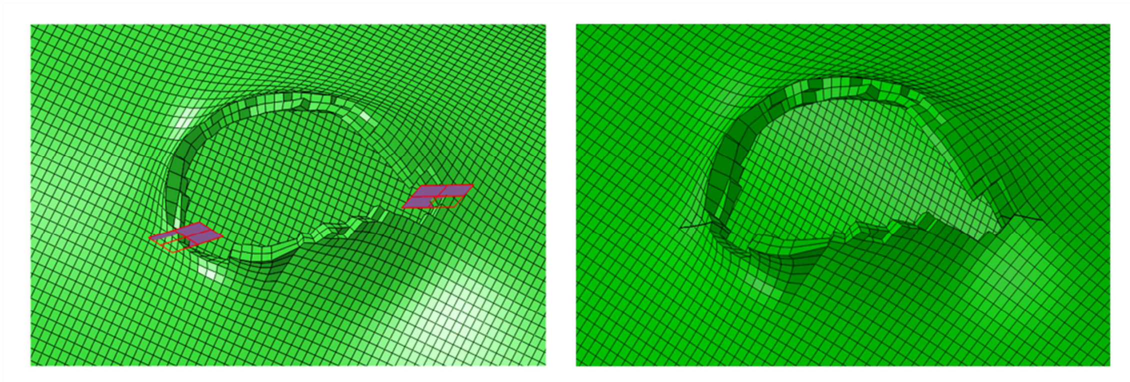

Fig.3– Generation of two cracks in the XFEM model

4. Discussion of the results

Numerical results are now discussed. Please note that the load case applied to both models, the standard approach and the XFEM approach, are the same: a static constant torsion plus an alternate torsion of near ±50% of the static one.

The results of the standard approach analysis were calculated using the Paris Law based on the Stress Intensity Factors (SIFs). Paris equation parameters were obtained from the literature, [3]. In Fig. 4, the K mode 1 for the internal crack (i.e. the one that propagates within the septum between the two holes, red line) and the external crack (the one that propagate outside the septum, blue line) are shown as a function of crack length. The ΔK obtained from the analysis without the residual stresses are higher than the one obtained considering them. As a consequence, the propagation speed, as shown in Fig.5, is much faster. For a comparison, the numerical propagation results are summarized in Table 1 with a percentage error with respect to the experimental data: 81%, neglecting the residual stresses, and 21% considering them. The residual stresses play a fundamental role in the crack propagation behaviour considering the typology of the problem here exposed. Particularly, considering the internal crack, the experimental data show that the propagation from the internal side starts when the external crack is well extended and the specimen is going to fail. On the contrary, the internal crack, neglecting the residual stresses, propagates from the beginning: the evaluation of the acquired data points out that, in this case, the propagation speed of the external crack is influenced by the evolution of the internal one. Besides, the extremely fast predicted propagation behaviour cannot be explained only with the simultaneous propagation of the two defects but additionally with an over estimation of the ΔK for each propagation step. In conclusion, the introduction of the residual stresses is very important because it strongly contributes to the correct prediction of the crack's initial propagation behaviour.

Considering the XFEM analysis, the results show some discrepancy with respect to the experimental tests. According to Table 1, considering the residual stresses, the life prediction is nearly three times higher than the experimentally obtained value, whereas, while neglecting the residual stresses, the life prediction is more realistic but obviously this result is not reasonable. Moreover, regarding the final crack propagation path, it must be specified that the internal crack failed to propagate, regardless whether the residual stresses were considered or not.

Table 1. Results comparison.

Numerical methods	Error Percentage
Standard model without Residual stresses	-81%
Standard model with Residual stresses	-23%
XFEM model without Residual stresses	14%
XFEM model with residual stresses	286%

5. Conclusion

The simulations carried out for this work confirmed the good agreement between the standard FE models for the crack propagation and the experimental results. Some weak points of this type of simulation have to be pointed out: first of all, the crack direction is almost fixed (especially if the cracks to be simulated are more than one) and the manual work needed to set up the full analysis process is considerable. On the contrary, the XFEM simulation requires less manual intervention during the propagation process, however the results obtained have to be carefully evaluated. Moreover another issue concerning the use of XFEM is the very high computation time.

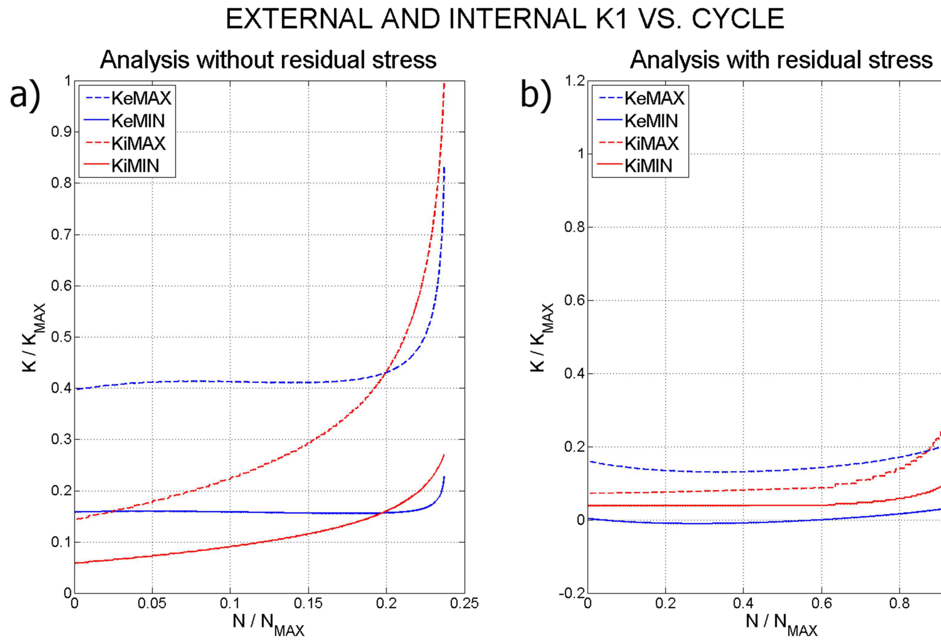

Figure 4 – (a) KI without residual stresses calculated for the external and the internal crack. (b) KI with residual stresses calculated for the external and the internal crack (values have been normalized)

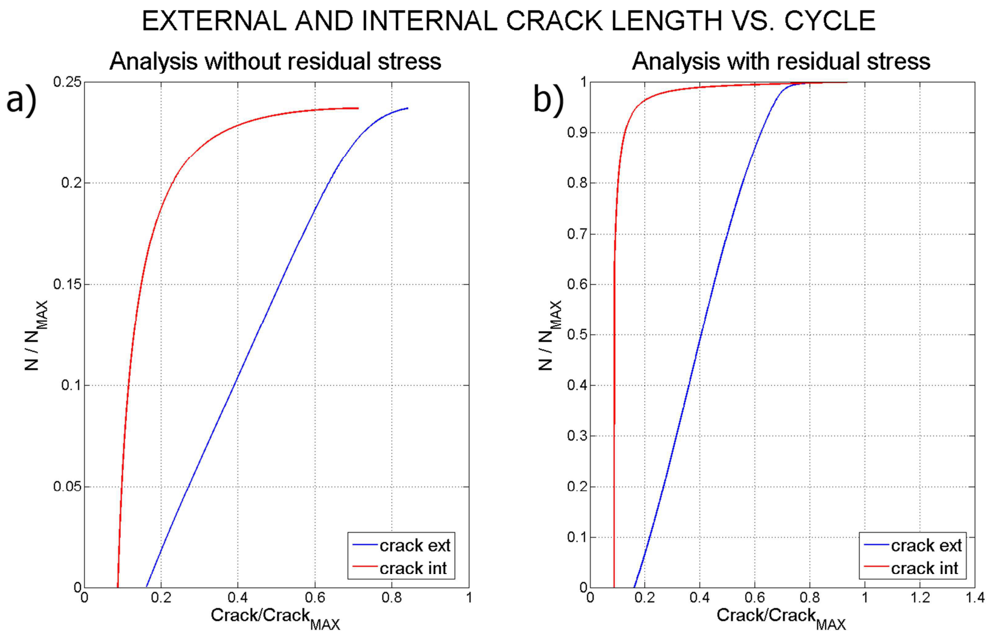

Fig. 5 – (a) Crack length without residual stresses calculated for the external and the internal crack. (b) Crack length with residual stresses calculated for the external and the internal crack (values have been normalized)

References

[1] Manes A, Lumassi D, Giudici L, Giglio M. An experimental-numerical investigation on aluminium tubes subjected to ballistic impact with soft core 7.62 ball projectiles. Thin-Walled Struct 2013;73:68-80.
[2] M. Giglio, A. Manes, Terminal ballistic effect on the crack growth assessment of a helicopter rotor drive, Eng. Fract. Mech. 78 (8), 1542 1554 (2011)
[3] NASGRO 4.11, Reference Manual, NASA Johnson Space Center; 2004.
[4] Melenk J, Babuska I. The partition of unity finite element method: basic theory and applications. Comput Methods Appl Mech Eng 1996;139:289–314.
[5] Belytschko T, Black T. Elastic crack growth in finite elements with minimal remeshing. Int J Numer Methods Eng 1999;45:601–20.
[6] Moes N, Dolbow J, Belytschko T. A finite element method for crack growth without remeshing. Int J Numer Methods Eng 1999;46:131–50.
[7] Feulvarch E., Fontaine M. b, Bergheau JM. XFEM investigation of a crack path in residual stresses resulting from quenching, Finite Elements in Analysis and Design 75 (2013) 62–70
[8] Singh I.V., Mishra B.K., Bhattacharya S., Patil R.U. The numerical simulation of fatigue crack growth using extended finite element method, International Journal of Fatigue 36 (2012) 109–119
[9] Fengmei X., Fuguo L., Jiang L., Min H., Zhanwei Y., Ruiting W., Numerical modeling crack propagation of sheet metal forming based on stress state parameters using XFEM method, Computational Materials Science 69 (2013) 311–326
[10] Y.Bao, T.Wierzbicki, A comparative study on various ductile crack formation criteria. J.Eng. Mater. Technol. 126(3), 314–324 (2004)
[11] M. Giglio, A. Gilioli, A. Manes, L. Peroni, M. Scapin, Investigation about the influence of the mechanical properties of lead core and brass jacket of a NATO 7.62 mm ball bullet in numerical simulations of ballistic impacts, Dymat 2012- 10th International Conference on the Mechanical and Physical Behaviour of Materials under Dynamic Loading, EPJ Web of Conferences, Volume 26, 04010, DOI: 10.1051/epjconf/20122604010