Numerical Simulation of Elastic-Plastic Deformation of Aircraft Fuel Tank Access Cover Impacted by Tyre Fragment

Alexander A. Ryabov, Vladimir I. Romanov, Sergey S. Kukanov, Anatoliy K. Botvinkin

Sarov Engineering Center

Abstract

All new-designed passenger aircrafts have to meet strict national and international safety requirements in accidents. One of the possible accidents is pneumatic tire damage, which can become the reason of the tire tread fragmenting. Some fragments of the tire tread with different mass and velocity can impact and break different plane elements. According to the safety requirements the designing company has to prove that the new plane withstands the tire fragment impact.
The development tests to meet this requirement are obviously difficult and costly experiments, so it is very cost-effective to use a numerical simulation in the design of a plane to solve the problem. In this case the simulations results should be verified by the model experiments.
The results of LS-DYNA® numerical simulation of an aircraft fuel tank access cover elastic-plastic deformation are presented in the paper. The results are compared to experimental data, obtained for the covers subjected to tire fragment impacts at different angles with the speed of 110 m/s. Comparison of the cover residual deflection and strain time-histories shows that the simulation results are in a good agreement with the experimental data.

1 Introduction

According to the national and international safety rules [1, 2], the fuel tank access covers must prevent loss of hazardous quantities of fuel in the normal and accident conditions. All the covers located in an area where experience or analysis indicates a strike is possible, must be shown by analysis or tests to minimise penetration and deformation by tyre fragments, low energy engine debris, or other likely debris.
Nowadays the numerical simulation rather then the full-scale experiment is used to confirm that the fuel tank access covers meet this requirement. In this case, considering the complexity of the problem, the numerical simulation results should be verified by the model experiments.
One of the examples of such verification is presented in the paper [3], where the comparison of the numerical and experimental investigation results of gas flow impact on the plate and tyre piece impact on the pipe problems is shown.
This paper presents the results of the numerical and experimental investigations of a fuel tank access cover, subjected to a tyre fragment impact. These investigations are also carried out to verify the accuracy of the numerical simulation methodology, which is used to confirm the safety of the passenger aircrafts in the possible accident conditions.

2 Statement of the experiments

A number of the experiments with a tyre fragment model impacting a fuel tank access cover installing on a wing panel model was carried out to verify the numerical simulation methodology. A view of the wing panel model is shown in Fig. 1. The tyre fragment model of given sizes and mass is cut out from the real passenger aircraft tyre (Fig. 2).

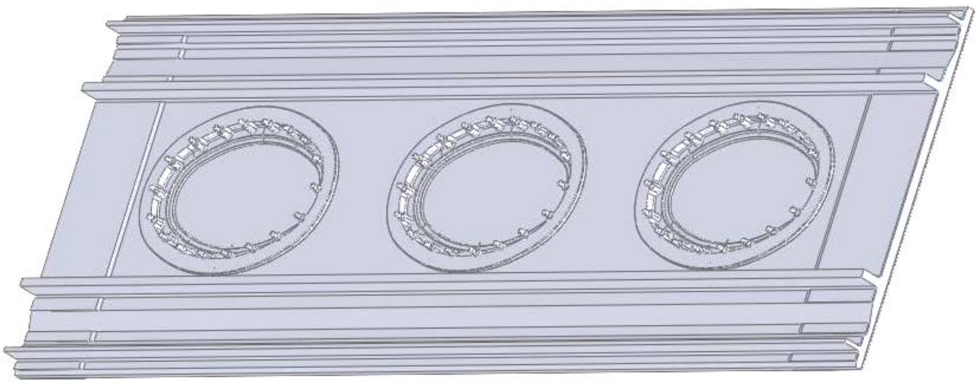


Fig. 1: Wing panel model

Fig. 2: Tyre fragment model (a - view from the tread side, b - view from the opposite side)

The five tests with the different slope of the wing panel, velocity of the tire fragment and location of the first impact point were carried out. A list of the experiments with the description of the main parameters is represented in Table 1. As a result of the experimental investigations, the time histories of the strains from the strain gages, installed on the cover's inner surface (Fig. 3) and the cover's residual deflection are obtained.

To verify the numerical simulation methodology, tests 1 and 5 were chosen. The choice of these two experiments is based on the following criteria:

- the given wing panel slope angles of 22° and 29°;
- the highest fragment velocities;
- the first impact point is located on the access cover;
- the highest covers' residual deflections.

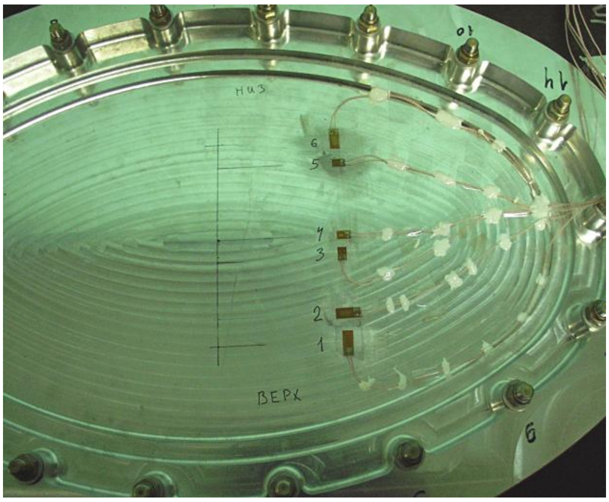


Fig. 3: Location of the strain gages

3 Description of the computer model

The finite element models of the wing panel with the access covers and the tire fragment were created to perform the simulations. The total size of the models is about 1,100,000 solid finite elements (1,450,000 nodes). Views of the wing panel, the access cover and the tyre fragment finite element meshes are shown in Fig. 4 and 5.
The tyre fragment is modelled as a solid piece of rubber without any reinforcement. MAT_OGDEN_RUBBER and the experimental stress-strain curves are used to describe the tyre fragment deformation. The curves are obtained from the uniaxial compression of the cylindrical specimens, which were cut out from the tyre.
The Friction coefficient for the tyre fragment to the wing panel contact interaction is set to 0.5.

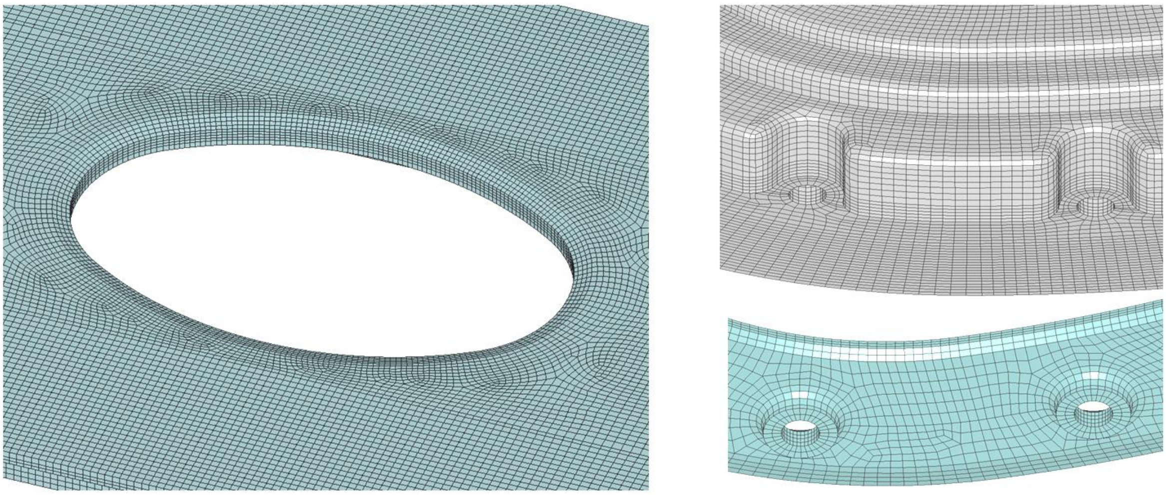


Fig. 4: Finite element meshes of the wing panel and the access cover

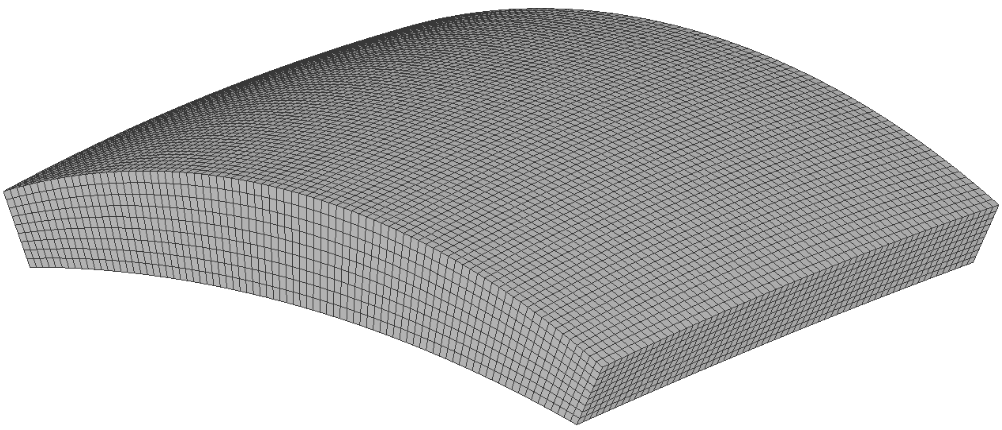


Fig. 5: Finite element mesh of the tyre fragment

4 Results of numerical and experimental investigations

Fig. 6 shows the experimental and simulated sequential views of the tyre fragment and the wing panel interaction process in test 5. The comparison of the results shows that the simulated motion and deformation of the tyre fragment are in a good agreement with the experimental ones.
The experimental and simulated deformed shapes of the access cover (for test 5) are shown in Fig. 7. As it can be seen from the figure, the simulated deformed shape of the cover is quite close to the experimental one.
Fig. 8 shows simulated and experimental diagrams of the access cover deflection along axis X-X (major axis of the cover elliptic shape) for test 1. Fig. 9 shows simulated and experimental diagrams of the access cover deflection along axis Y-Y (minor axis of the ellipse) for the test 5. Comparison of the

simulated and experimental diagrams shows that they are in a good agreement. Difference of calculated and experimental maximum deflections of the cover for test 1 is about 1% and for test 5 - about 3 %

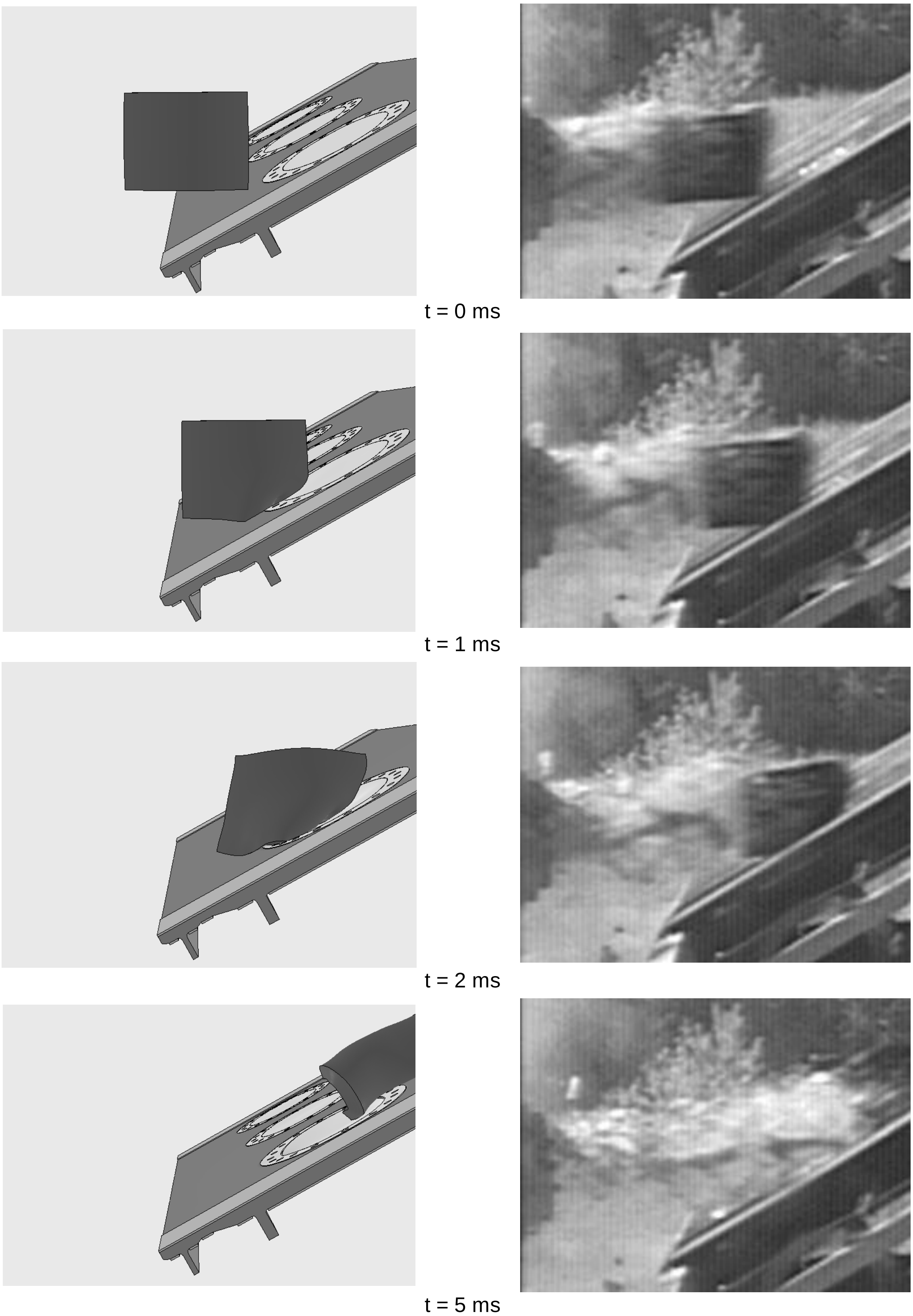


Fig. 6: Sequential views of the tyre fragment and wing panel interaction process (test 5)

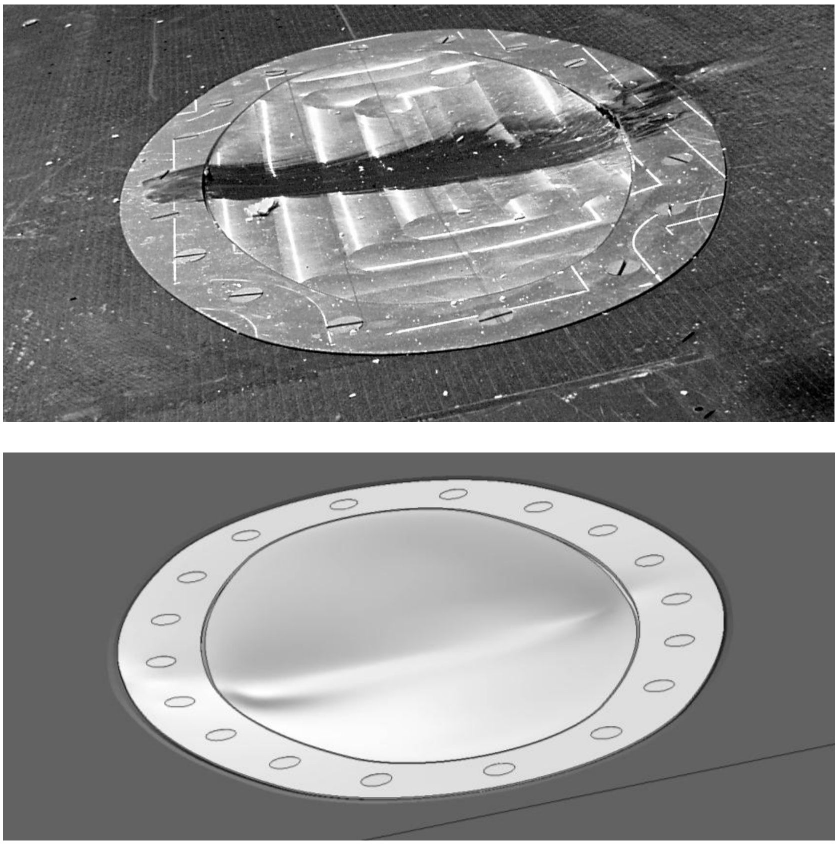


Fig. 7: Deformed shape of the access cover (test 5)

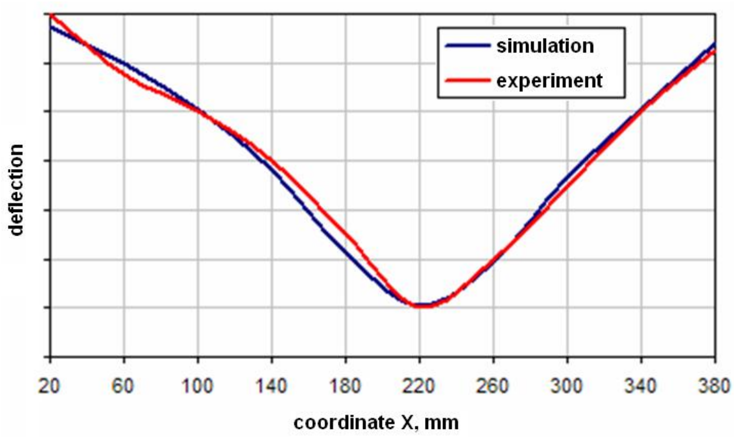


Fig. 8: Diagram of the access cover residual deflection along axis X-X (test 1)

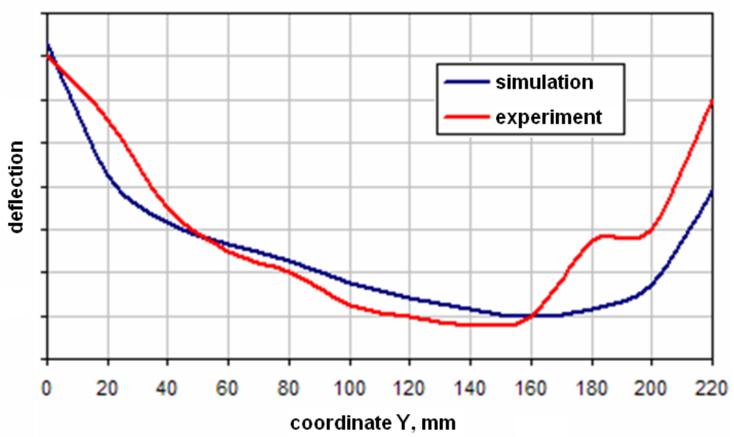
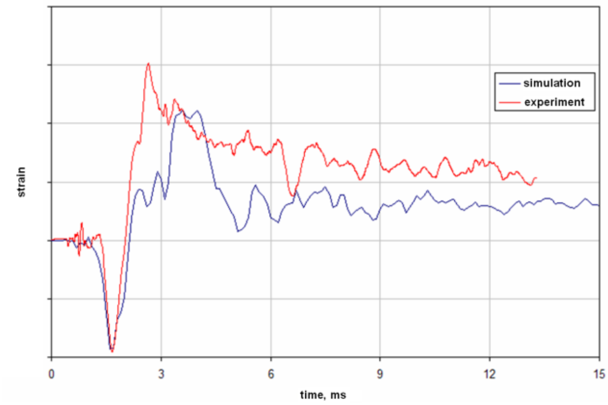


Fig. 9: Diagram of the access cover residual deflection along axis Y-Y (test 5)

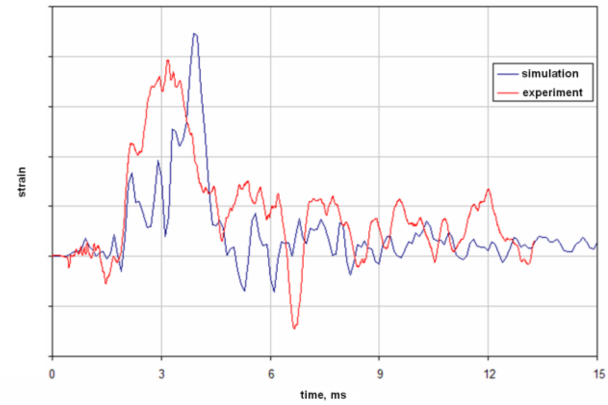
The simulated and experimental strain time histories for strain gages T1...T6 are shown in Fig. 10 (test 1) and Fig. 11 (test 5). As it can be seen from the figures, the numerical simulation quite correctly describes the process of the strain changing in time. Both the calculated and experimental time histories can be divided into three main stages:

- the compression in the early stage of the impact;
- the tension due to deflection of the cover in the second stage;
- the unloading after rebound of the tyre fragment, which is characterized by oscillation about a midrange.

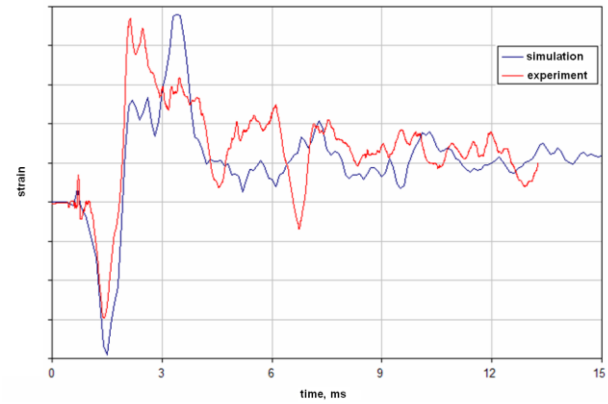
The quantitative comparison of the calculated and experimental strains shows that differences between maximum values are varying from 0% to 30%. Taking into account that the level of maximum strains is quite low (less than 1%), this difference is considered to be satisfactory. The possible cause of the differences is that reinforcement of the tyre fragment is not modelled in the simulation. This assumption can lead to slightly different deformation of the tyre fragment and, as a result, to different loading of the cover.



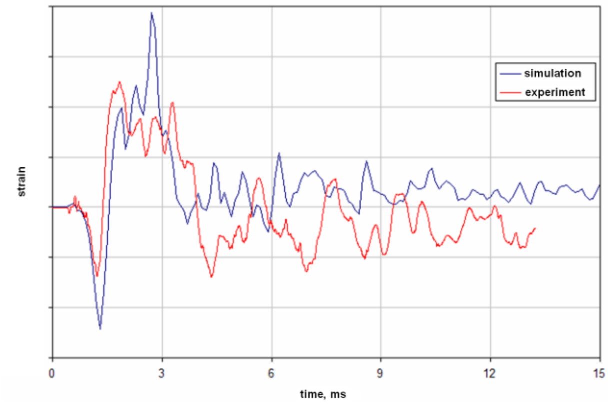
Gage T1



Gage T2



Gage T3



Gage T5

Fig. 10: Strain time histories (test 1)

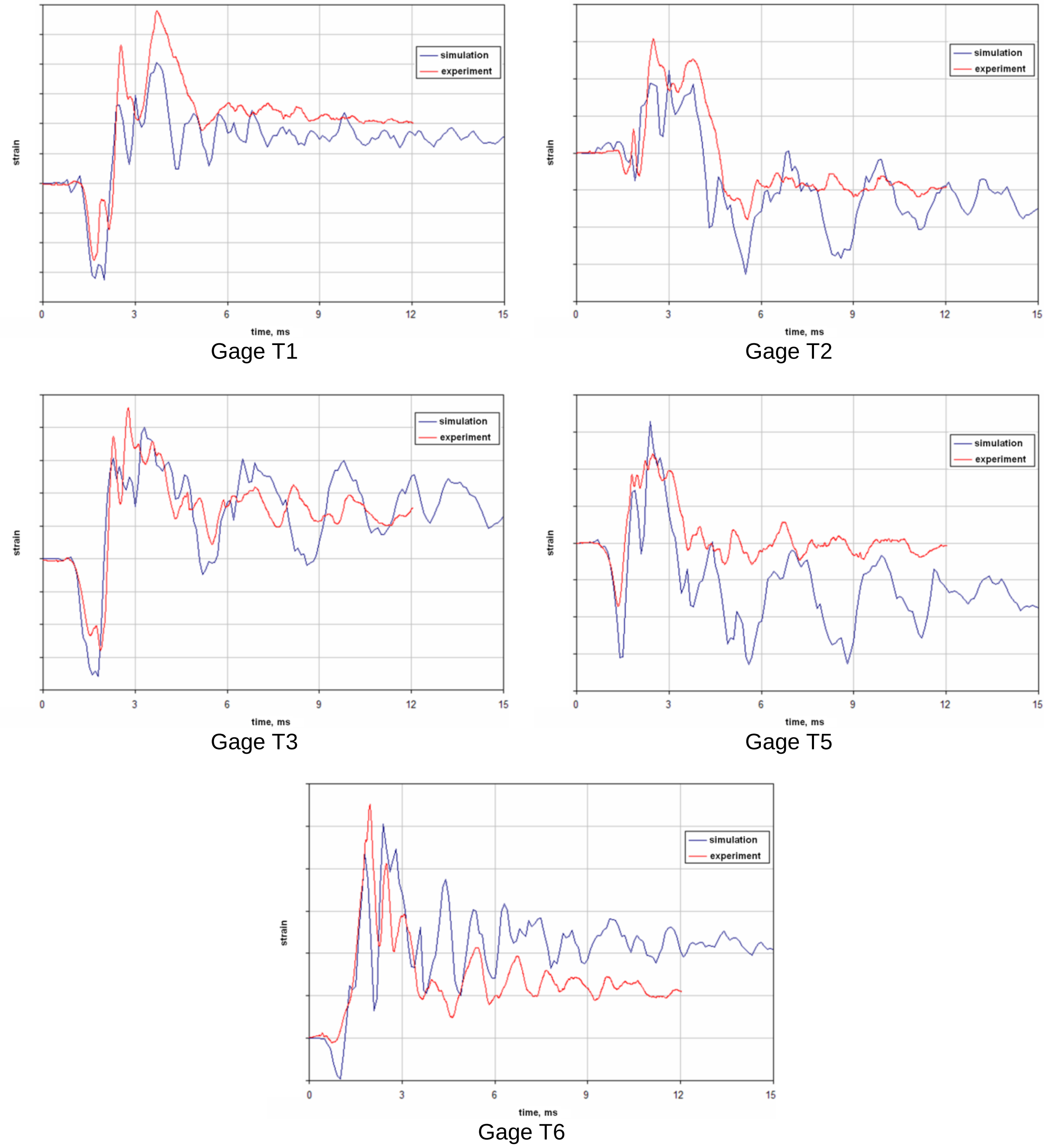


Fig. 11: Strain time histories (test 5)

5 Summary

The numerical and experimental investigations of the dynamic deformation of the wing panel with the fuel tank access covers subjected to the tyre fragment impact are carried out to verify the numerical simulation methodology.

Basing on the comparison of the simulation and experimental results it can be concluded that:

1. The used numerical simulation methodology allows correct describing of elastic-plastic deformation of the fuel tank access cover subjected to a high-speed impact of the tyre fragment.
2. The relative difference of the simulation and experimental maximum residual deflection of the cover is less than 3%. The differences between the maximum values of the strains, obtained for 9 strain gages, are varying from 0% to 30%. Taking into account that the level of maximum strains is quite low (less than 1%), this difference is considered to be satisfactory (or can be considered as satisfactory).

3. The verified numerical simulation methodology can be used to confirm that the wing panel with the access cover meets SC25.963(e) requirements.

6 References

[1] Aviation rules. Part 25. Intergovernmental aviation committee, 2004.
[2] Certification Specifications and Acceptable Means of Compliance for Large Aeroplanes. CS-25, Amendment 12, 13 July 2012.
[3] Ryabov A., Romanov V., Kukanov S. et al. Numerical Simulation of Consequences of Passenger Aircraft Tyre Damage. Proceedings of 8th European LS-DYNA Users Conference, Strasbourg - May 2011.