32nd Congress of the International Council of the Aeronautical Sciences
September 6-10, 2021 Pudong Shangri-La, Shanghai, China

BIRD STRIKE SIMULATIONS IN MSC.SOFTWARE: CORRELATION BETWEEN THE EXPERIMENTS AND SIMULATIONS

V. Symonov, J. Juračka

Institute of Aerospace Engineering, Brno University of Technology, Technická street 2896/2, 616 69, Brno, Czech Republic

Abstract

A partial parametric study of FE simulations of bird strike tests of CFRP panel is presented. The simulation were performed with help of MSC.Dytran and MSC.Nastran software, where two different simulation methods were used: Lagrangian-Eulerian (LE) and Smooth Particles Hydrodynamics (SPH). During the study the simulation results were compared to test ones. In particular the maximum deflection profile and failure extent and position were compared. The main objective of the study was to find both the parameters of FE models, which significantly influence the simulation results, and choose the software and simulation method, which give the best correlation with experiment. Based on correlation of panel maximum deflection and failure extent and position conclusions about influence of the investigated parameters on simulation results and about advantages and disadvantages of different simulation software and methods are made.

Keywords: bird strike, finite element simulation, MSC.Software, Lagrangian-Eulerian, smooth particles hydrodynamic.

1. Introduction

According to the modern airworthiness regulations the aircraft structures, such as leading and trailing edges, air intakes, nacelles, nose cowls, etc., must proof sufficient crashworthiness against hail, bird and debris impacts. The most precise and effective method to do it, of course, is the impact test. In case of debris impacts, since the low impact energy and initial velocity, the tests are quite simple and cheap, however the bird strike/hail impact test are quite complex and expansive. This is the reason why and where FE simulations can help to prepare experiment more effectively and reduce the number of tests, thus the experiments budget, time and human resources can be reduced dramatically.

There exist many explicit codes, which can be used for simulation of such dynamic problems like bird strike – LS-Dyna, PamCrash, ABAQUS, MSC.Dytran, MSC.Nastran, etc. The correlation of simulated and experimental results highly depends on the used simulation methods [1, 2], FE modeling techniques and model complexity [3, 4, 5]. However, industrial problems require as simple models as possible and usually shell FE models are used, which have known limitations especially in case of out-of-plane loading and composite structures.

The full-scale structural tests and simulations of these tests should be preceded by material tests and validated by tests of simple characteristic structural elements like panel, spar, etc. All these stages should be accompanied with FE simulations, where customization/tuning of material models and other parameters, which influence and control the simulation process, should be done. The material models can be customized based on material tests and the simulation control parameters – based on tests of characteristic structural elements.

Nowadays, the bird strike simulation problem is widely investigated with help of different FE analysis codes for different structures, materials, impact energies and velocities (e.g. [1]), however in most cases the studies are revealing the influence of FE model and simulations parameters in general or in particular case for particular structure, material, impact energies and velocities. There does not exist any fundamental work on this topic. This is why any investigation in this area would be worthwhile.

This work includes a series of bird strike tests simulations of such a characteristic structural element as CFRP panel using MSC.Dytran and MSC.Nastran software and two different simulation methods: LE and SPH. During these simulations an attempt to tune the FE model parameters was made in order to improve the correlation between the tests and simulations results. Therefore, a partial parametric study of FE models, simulation methods and software was performed. The simulation results were compared to each other and to bird strike tests results.

2. Bird strike tests and FE simulations

2.1 Bird strike tests description

The bird strike tests were performed in the framework of the TE02000032 project (see Acknowledgement section). Their detailed description and results are given in [6], however for clear imagination the specimen geometry is shown in Figure 1, test frame and jig necessary for specimen fixation are shown in Figure 2. The fresh dead 1 kg chickens were used as projectiles during the tests. The ply properties of the composite material used for specimens manufacturing are listed in Table 1.

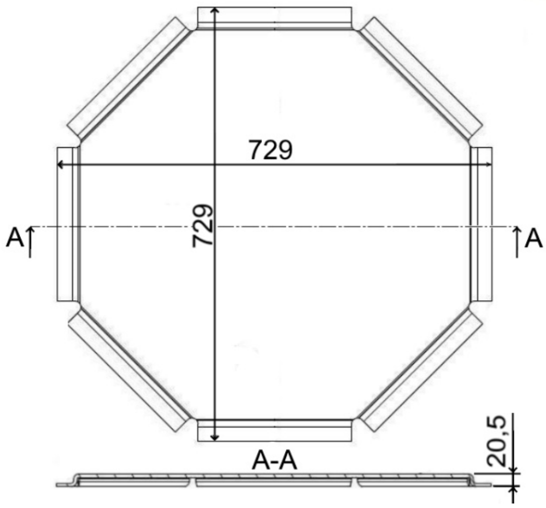


Figure 1 - Geometry of specimen

Table 1 - Hexply 8552 material properties [7]

Property	Unit	Value
$E_{11} = E_{22}$	MPa	60000
G_{12}	MPa	4500
$G_{13} = G_{23}$	MPa	3000
μ_{12}	-	0,38
$X_t = Y_t$	MPa	625
$X_c = Y_c$	MPa	738
S	MPa	100
ILSS	MPa	66
t	mm	0,21

The main parameters of the tests (projectile velocity and impact angle) and the test panels' thickness are provided in Table 2.

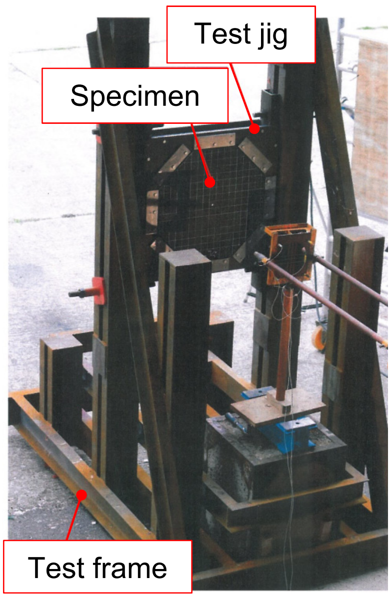


Figure 2 - Test frame and jig

Table 2 - Main parameters of the tests and specimens

#	Panel thickness, mm	Initial bird velocity, km/h	Impact angle, deg
1	3.6-3.7	227	90
2	3.6-3.7	333	90
3	3.6-3.75	350	30
4	3.6-3.7	429	30
5	3.6-3.7	487	30
6	3.7-3.8	493	30
7	3.6-3.7	520	30
8	3.7-3.8	604	30

2.2 Bird strike FE simulations

For bird strike modeling and simulation MSC.Dytran software and LE method were chosen. Modeling of test panels was done with help of 2D Belytschko-Tsay shell elements CQUAD4 and CTRIA3 with

average size of 15 mm (see Figure 3). The panel geometry corresponds to Figure 1. The panel's boundary conditions were simplified to fixation of all nodes translations (UX, UY, UZ) at the panel rim, since it was proved such a simplification did not have significant influence on the simulation results in the central zone of the panel. The shape of the bird in accordance with standard world practice (e.g. see [1]) is a cylinder with hemispheric ends. The length to diameter aspect ratio is 2:1. The volume and dimensions are calculated taking into account the weight of 1 kg.
The air and the impactor were modeled by 3D Euler elements MM/Hydro (PEULER1) (see yellow elements in Figure 3). The material model used for the air was Idial Gas (DMAT, EOSGAM) and for the impactor - LinFluid (DMAT, EOSPOL). The air density was set to 1,1848 kg/m^3, and the adiabatic exponent to 1.4. The water bulk modulus was set to 2200 MPa. For the interaction between Euler and Lagrangian elements the General Coupling (COUPLE card) was chosen. The initial position and shape of the impactor was defined with help of Dummy shell elements (see red elements in Figure 3). The composite material, which the specimens were made of, was defined by the MAT8A material model and the laminate material with PCOMPG card. The in-plane shear plasticity factor ALFA for MAT8A card was set to 5x10^{-7}.

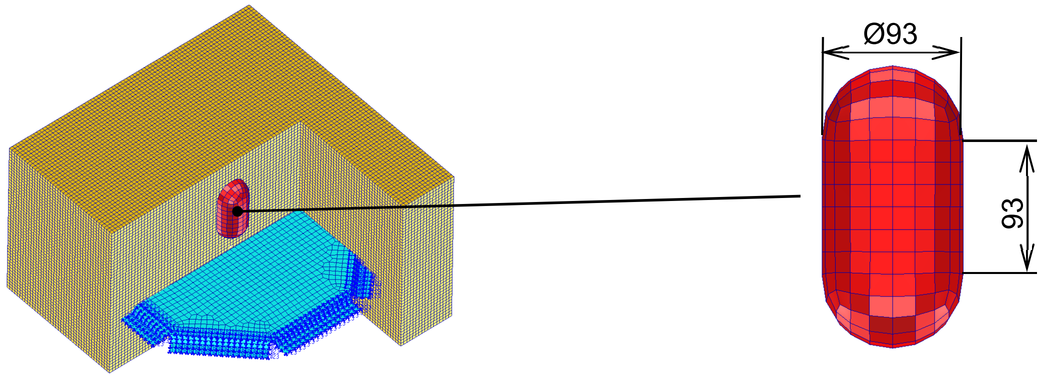


Figure 3 – MSC.Dytran FE model for bird strike simulations

3. Comparison of bird strike tests and FE simulations results

First, the profile of maximum deflection in the A-A cross section of the panel (see Figure 1) was compared for all cases from Table 2 (see Figure 4 and Table 3). The best correlation of the deflection occurred at askew impact (30°) in the cases 5 and 8 and the worst at perpendicular impact in the cases 1 and 2.

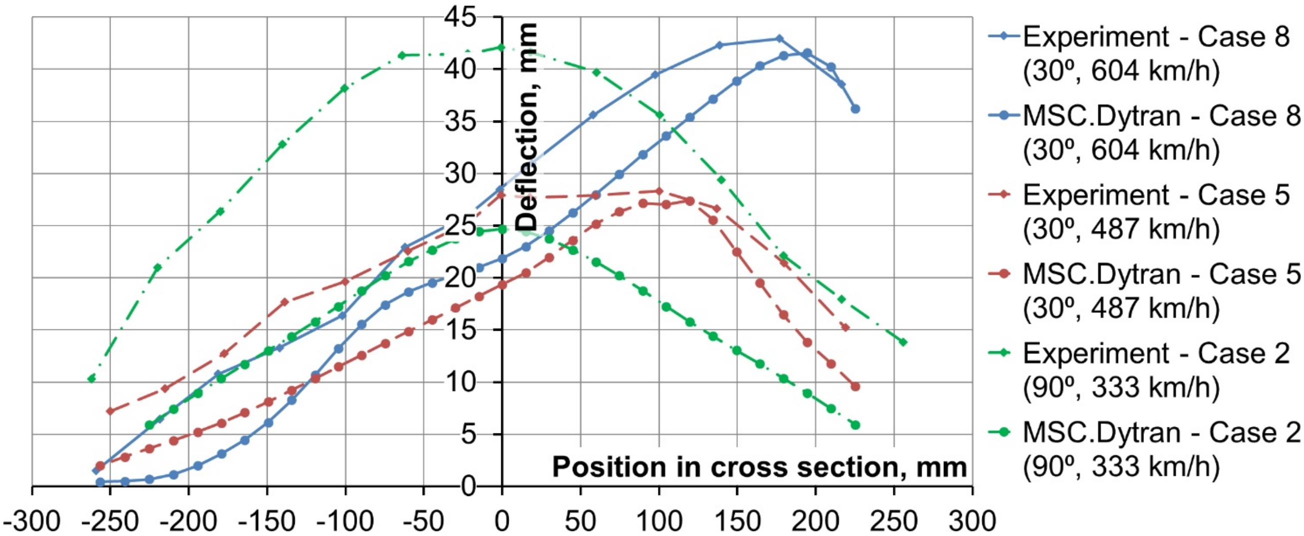


Figure 4 – Comparison of maximum deflection profile of the panel

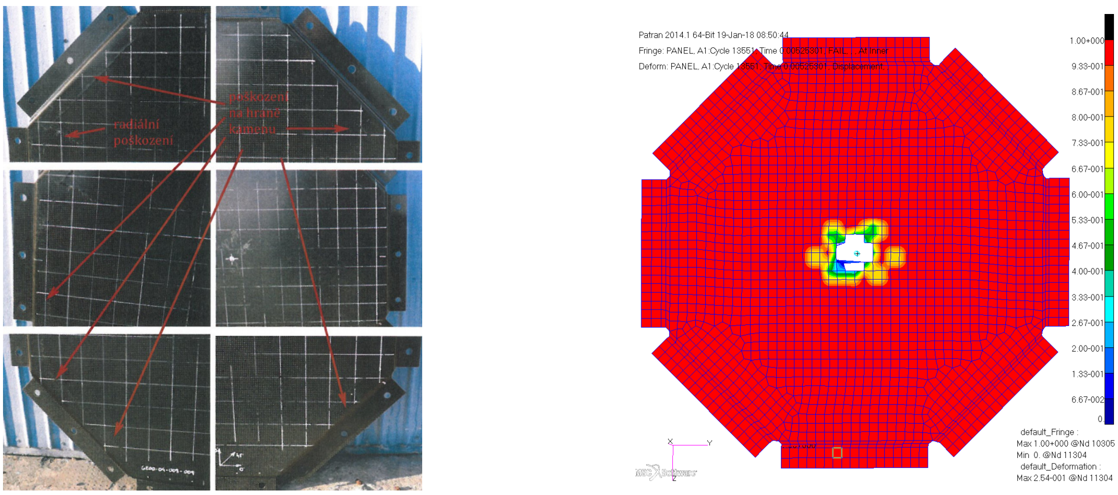
Then, the failure extent and position were compared (see Figures 5 - 6 and Table 3). The most severe failure was occurred in the case 8 at askew impact with initial bird velocity of 604 km/h (see Figure 6).

Table 3 – Comparison of maximum deflection and failure of specimens during tests and simulations

#	Velocity, km/h	Impact angle, deg	Max deflection, mm		Error*	Failure	
			experiment	simulation		experiment	simulation
1	227	90	34,3	20,62	40%	no	no
2	333	90	42,1	24,7	41%	significant	small local, penetration
3	350	30	31,0	19,1	39%	no	no
4	429	30	32,0	23,9	25%	small local	no
5	487	30	28,3	27,4	3%	significant	significant
6	493	30	49,1	32,1	19%	complete, penetration	complete, penetration
7	520	30	33,3	29,3	12%	significant	significant
8	604	30	43,0	41,6	3%	complete, penetration	significant, penetration

*the error was calculated as the difference between the upper most points of curves in Figure 4.

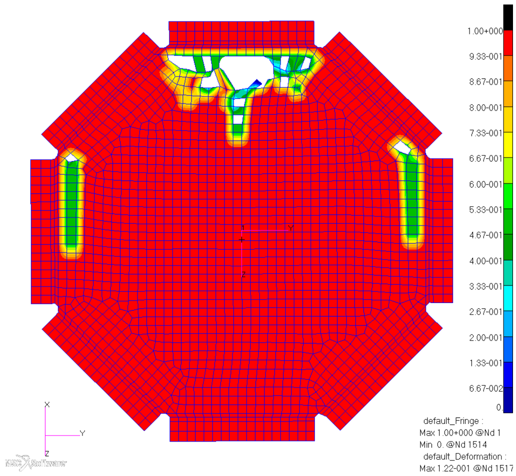
The failure correlation between the experiment and simulations is quite poor in all its aspects (extent and position) in most of cases from Table 3. As can be seen from the figures, corresponding to the experiment, the macroscopic cracks always started to appear right close to the edges of the supporting jig. However, the simulations show, the failure starts in place, where the impactor touches the plate. Moreover, the simulations in cases 2 – 6 show partial penetration of the impactor through the panel, while the experiment shows such a penetration only in the case 6. Even in the case 8 (see Figure 6), where the best correlation can be seen, the simulation shows much less severe failure extent. The cracks can be seen only on the left and right sides of the octangular panel, while they should be along almost all sides according to the experiment.



Experiment MSC.Dytran

Figure 5 – Failure comparison, case 2 (impact angle – 90º, initial bird velocity – 333 km/h)

Experiment



MSC.Dytran

Figure 6 – Failure comparison, case 8 (impact angle – 30º, initial bird velocity – 604 km/h)

4. Partial parametric study of FE model, used methods and software

4.1 Description of used FE models, methods and software

For the parametric study, two different software MSC.Dytran and MSC.Nastran (SOL700) were used. In case of MSC.Nastran two different methods LE and SPH were used.
The MSC.Dytran FE model was described in the subsection 2.2.

Table 4 – Parameters of the MATD058 material model

Property	Unit	Value
TAU1	MPa	50
GAMMA 1	-	0,011
SLIMT1	-	10^{-12}
SLIMC1	-	10^{-12}
SLIMT2	-	10^{-12}
SLIMC2	-	10^{-12}
SLIMS	-	1
AOPT	-	0
TSIZE	s	10^{-8}
ERODS	-	0,35
SOFT	-	0,55
FS	-	-1
E11C	-	0,0123
E11T	-	0,01
E22C	-	0,0123
E22T	-	0,01
GMS	-	0,25

In the MSC.Nastran (SOL700) FE model in case of LE method the material model used for the air and impactor was MATDEUL. The composite material was modeled by MATD058 material model and PCOMPG card for laminate definition. The material properties for the MATD058 were taken from Table 1, the other parameters were set according to Table 4. The other was the same as for the MSC.Dytran FE model.
In case of SPH simulation of the impact the impactor was modeled with 0D mass elements CSPH. The mass of the elements was calculated in accordance with their number within predefined volume. The volume and the shape of the impactor was the same as in Figure 3.
The material model taken for the SPH particles was MATD010 with the next parameters: Shear Modulus = 0, Yield Stress = 0, Hardening Modulus = 0 a Cutoff Pressure = -10 Pa. In some cases the MATD009 was also used.
For the interaction between the particles the Gruneisen Equation Of State (EOS) was taken. It was defined with help of EOSGRUN card. The parameters set in the card are listed in Table 5.
The initial velocity of the particles was defined with the TCID card (Transient Analysis Initial Conditions with Increment Options), which allows to define the velocity of taken nodes in a selected coordinate system.
The contact between the panel and particles was modeled with BCTABLE card. The panel was set as "MASTER" defined with BCBODY, BSURF. The impactor was set as "SLAVE" defined with BCBODY, BSGRID.
Definition of parameters for particles computation is

important also. It is defined with help of SPHDEF card. The number of cycles between particles rearrangement (NCBS = 10) and time for death of the particles, DT (equal or less than the full time of simulation). In addition, the parameter for the initial number of neighboring particles, MEMORY, was defined – it is equal to the entire number of particles.
MSC.Nastran (SOL700) impact simulations were performed only for cases 2 and 6 by LE method and for cases 5, 7 and 8 by SPH method (see Table 2).

Table 5 – Parameters of the EOSGRUN card

Parameter	Constant, C	Constant, S1	Constant, S2	Constant, S3	Gruneisen Gamma, GAMMA0	First Order Volume Correction, A	Initial Internal Energy, E0	Initial Relative Volume, V0
Unit	-	-	-	-	-	-	J/m³	-
Value	1647	1,921	-0,096	0	0,35	0	289500	-

4.2 Parametric study results and discussion

The study was performed in order to understand the influence of FE model parameters and used simulation software and methods on the simulation results and to find the good way to tailor the FE model, which will give relevant simulation results.
The Table 6 shows subcases of FE model parameters study for MSC.Dytran LE simulations. The parametric study of MSC.Dytran FE model was performed for the case 1 from Table 2. According to Figure 7 the FE model parameters change listed in Table 6 does not have significant influence on the simulation results, in particular on deflection and failure of the panel.

Table 6 – Subcases of MSC.Dytran FE model parametric study

Subcase	**Impactor density, kg/m³**	**Contact thickness, mm**	**Young's moduli, GPa**	**Interlaminar shear moduli, GPa**	**Strength (tens., compr., shear), %**
Baseline	950	$2{,}34 \times 10^{-3}$	60	3	100
Impactor 900	900	$2{,}34 \times 10^{-3}$	60	3	100
Impactor 800	800	$2{,}34 \times 10^{-3}$	60	3	100
Contact thickness	950	0	60	3	100
Young's moduli	950	0	55	3	100
Strength 80%	950	0	60	3	80
Interlaminar	950	0	60	1	100

The Figures 8 - 9 show comparison of the maximum deflection profile of the panel between experiment and MSC.Dytran and MSC.Nastran (SOL700) LE simulations for cases 6 and 7 from Table 2. It can be concluded according to the figures, the difference in maximum deflection between these two simulations is not significant. From Figures 10 - 11 it can be seen MSC.Nastran (SOL700) LE simulations does not allow to predict the failure extent with good probability. However, in the two first cases the experiment showed significant failure of the panel, simulations did not showed any failure. In the third case the simulation result shows much severe failure of the panel than it was obtained from the experiment.
Unfortunately, MSC.Nastran (SOL700) does not have an ability to obtain the failure indices for laminate and its layers from simulation results, when the material model MATD058 is used for composite material modeling.

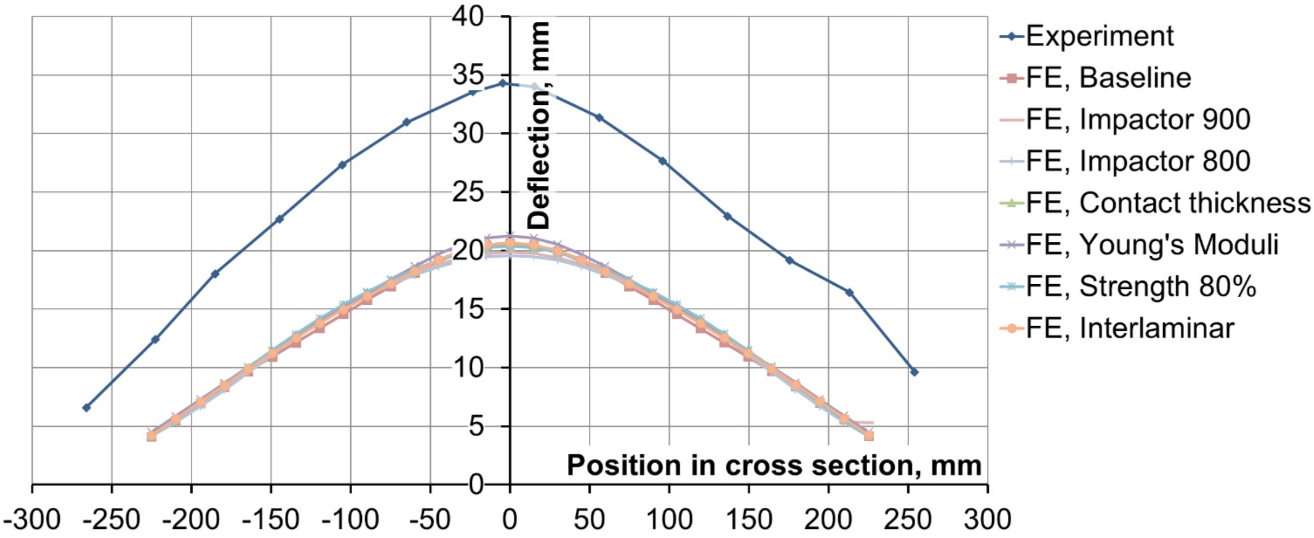


Figure 7 – Comparison of the panel's maximum deflection profile between experiment and FE MSC.Dytran simulations, case 1 (impact angle – 90º, initial bird velocity – 227 km/h)

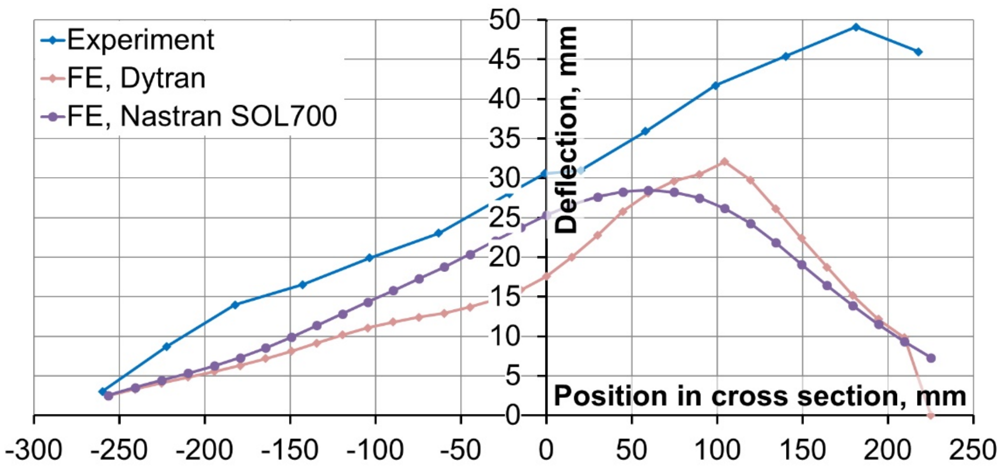


Figure 8 – Comparison of the panel's maximum deflection profile between experiment and MSC.Dytran and Nastran LE simulations, case 6 (impact angle – 30º, initial bird velocity – 493 km/h)

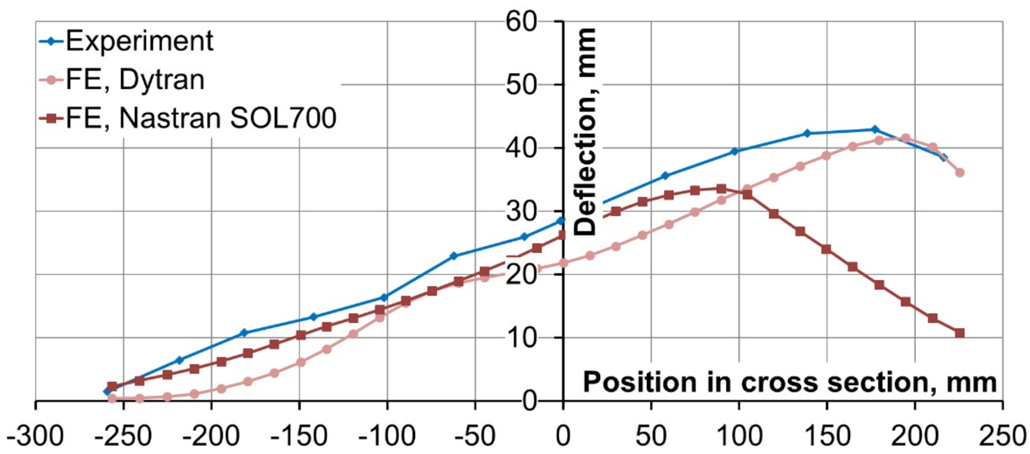


Figure 9 – Comparison of the panel's maximum deflection profile between experiment and MSC.Dytran and Nastran LE simulations, case 8 (impact angle – 30º, initial bird velocity – 604 km/h)

The next part of the study is concerning the simulations with help of SPH method, which were realized with help of MSC.Nastran SOL700. The simulations were performed for cases 1 and 2 form Table 2. The variation of FE model parameters is shown in Table 7.
It can be seen from Figure 12, the variation of particles number has slight influence on the maximum deflection profile. However, the other investigated parameters of the SPH FE model do not have significant influence on it (see Figure 13). The maximum deflection is 5 mm deeper, than it was in case of MSC.Dytran LE simulations (see Figure 7), but the maximum correlation error is still quite high ~32%.

Table 7 – Subcases of MSC.Nastran (SOL700) SPH FE model parametric study

Subcase	**Impactor density, kg/m^3**	**Number of particles**	**Contact thickness, mm**	**Young's moduli, GPa**	**Interlam. shear moduli, GPa**	**Impactor material model**
Baseline	950	1964	0	60	3,0	MATD010
2234 particles	950	2234	0	60	3,0	MATD010
2710 particles	950	2710	0	60	3,0	MATD010
Impactor 850	850	2145	0	55	3,0	MATD010
Impactor 850, MATD009	850	2145	0	55	3,0	MATD009
Impactor 850, MATD009, Modulus	850	2145	0	55	3,0	MATD009
Impactor 850, Friction	850	2145	0.5	55	3.0	MATD010
Impactor 850, Interlam	850	2145	0	60	1.0	MATD010
Impactor 800	800	2314	0	60	3,0	MATD010
Impactor 800, Interlam	800	2314	0	60	1.0	MATD010

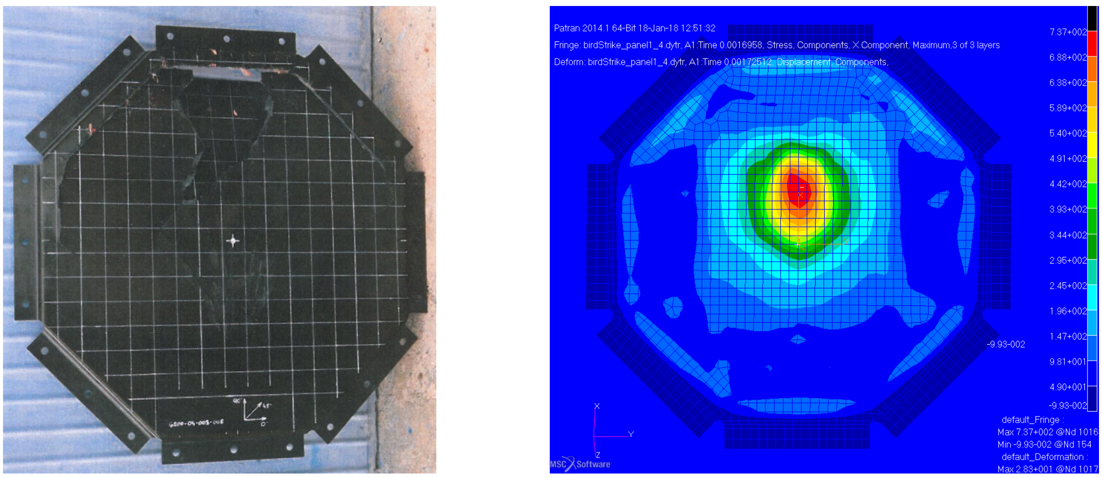


Figure 10 – Comparison of the panel's failure between experiment and MSC.Nastran LE simulations, case 6 (impact angle – 30º, initial bird velocity – 493 km/h)

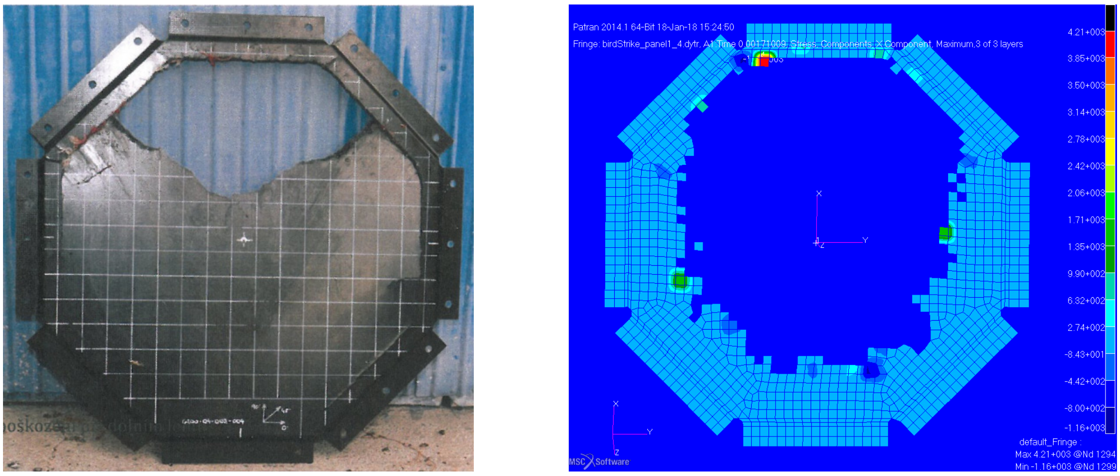


Figure 11 – Comparison of the panel's failure between experiment and MSC.Nastran LE simulations, case 8 (impact angle – 30º, initial bird velocity – 604 km/h)

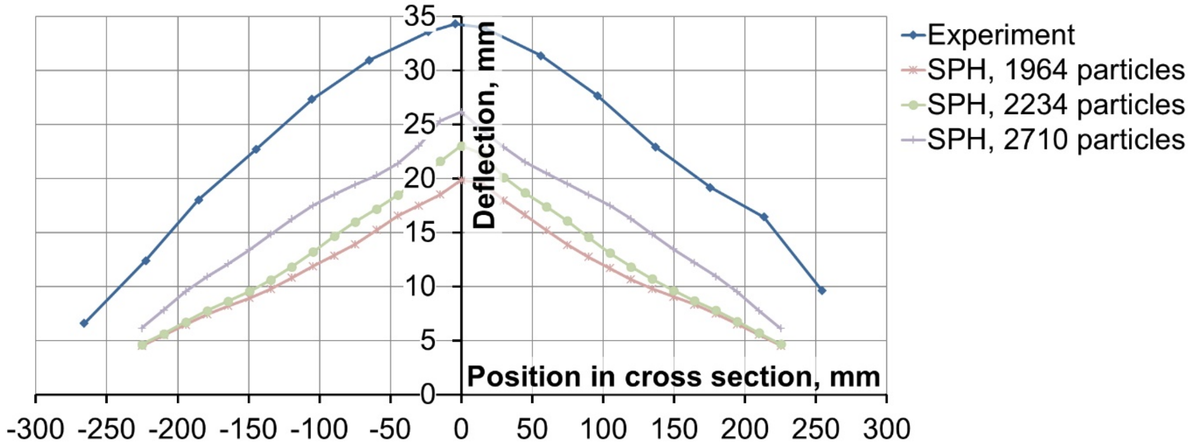


Figure 12 – Maximum deflection profile change at particles' number variation, case 1 (impact angle – 90º, initial bird velocity – 227 km/h)

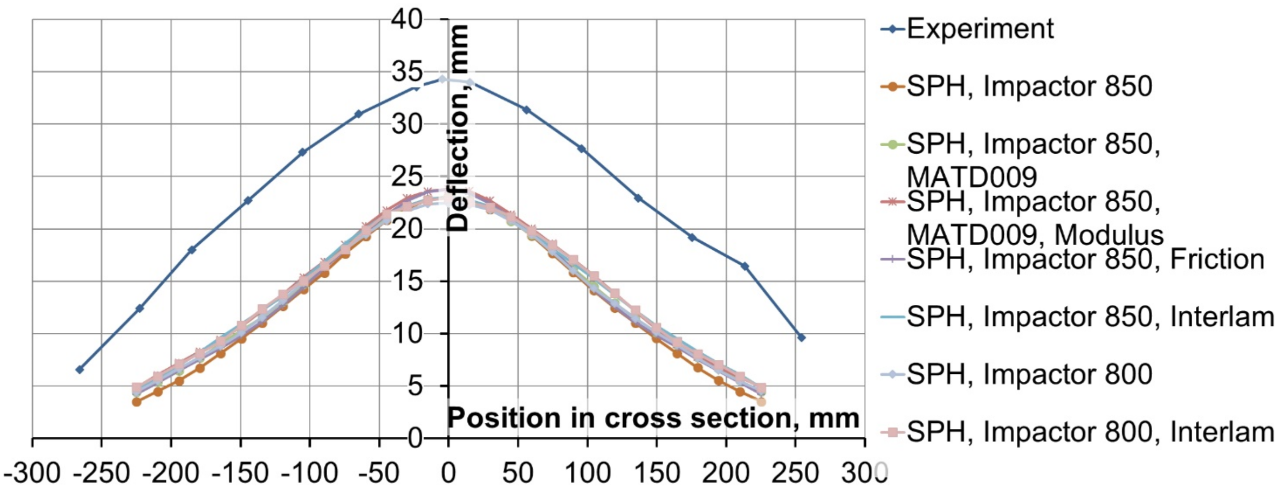


Figure 13 – Maximum deflection profile change at SPH model parameters variation, case 1 (impact angle – 90º, initial bird velocity – 227 km/h)

Comparison of failure extent between experiment and SPH simulations for case 1 showed large inconsistence. There was not any visual failure of the panel during the experiment, however the SPH simulation gave significant failure of the central portion of the panel with penetration of the impactor, when the impactor density was set to baseline value of 950 kg/m^3. After the density was corrected to 850 kg/m^3, the failure has not appeared.

5. Conclusions

The performed simulations have shown that in general the maximum deflection of the panel does not have good correlation with experiment. The correlation error of the deflection is in the range of 12 - 41% except two cases. It is almost the same for simulations performed by MSC.Dytran (LE) and MSC.Nastran (LE and SPH). Varying the FE model parameters such as impactor density, in-plane and interlaminar stiffness of the panel, contact thickness, laminate strength and friction coefficient do not influence the simulation results significantly.

The simulations in MSC.Dytran have a partial correlation with experiment in sense of failure prediction. In the most of cases, the simulation allows predicting the failure extent. On the other hand, in general, the simulations do not allow predicting the failure position. However, during the experiments failure started close to the supporting jig in all cases, in simulations it started where the impactor touched the panel.

MSC.Nastran SOL700 simulations using LE method give even worse results. They do not allow to predict the failure with good probability: in two investigated cases the experiments showed significant failure, while the simulations did not show any. In the third case the simulation showed much severe failure, than it was seen from experiment. Also, is was revealed the SOL700 does not allow investigating the laminate failure indices in case of impact simulations, when the laminate modeler and the MATD058 are used.

The most simple and fast for simulation of bird strike impact is the SPH method with help of MSC.Nastran. However, it also does not give good correlation with experiment. The failure extent can be tuned by varying the impactor model density, but varying of the other investigated parameters does not influence the precision of predicting the failure position and maximum deflection.

The most probable reason of such a poor correlation between experiment and simulations could be the simplified model. The used shell elements and material model cannot take into account delamination failure mode and through-the-thickness deformation and failure. The complex FE models ([8], [4], [9]) can take these phenomena into account, however such modelling techniques will lead to dramatic increasing in simulations cost and barely can be applied for real aero structures modeling. Also, the important parameter, which can improve the simulation results is the impactor porosity. Unfortunately, the MSC.Software do not have any means for modeling porosity in the impactor.

The authors in [6] used ABAQUS explicit solver with double precision for such simulations and compared the results with the same experiments. ABAQUS has given much better correlation with experiment.

Contact Author Email Address

Mailto: symonov@fme.vutbr.cz

Copyright Statement

Acknowledgement

This work was done with the financial support of the next projects: NETME Centre built in the frame NETME Centre “New Technologies for Mechanical Engineering” (Reg. No. CZ.1.05/2.1.00/01.0002) with the financial support from the Operational Program Research and Development for Innovations; TE02000032 “Advanced Aerostructures Research Centre” financed by the Technology Agency of the Czech Republic.

References

[1] Heimbs S. Computational methods for bird strike simulations: A review. *Computers and structures,* No. 89*,* pp 2093-2112, 2011.
[2] Georgiadis S, Gunnion A. J, Thomson R. S and Cartwright B. K. Bird-strike simulation for certification of the Boeing 787 composite moveable trailing edge. *Composite structures,* No. 86, pp. 258-268, 2008.
[3] Johnson A. F, Pickett A. K and Rozycki P. Computational methods for predicting impact damage in composite structures. *Composites science and technology,* No. 61, pp 2183-2192, 2001.
[4] Mocayo J. E. D, Wagner H, and Drechsler K. Benchmarks for composite delamination using LS-Dyna 971: low velocity impact. *LS-Dyna anwenderforum*, Frankenthal, Germany, pp D-I-13-D-I-22, 2007.
[5] Mališ M. Impact analysis of the rigid body on thin-walled aluminium structure with considering of the stochastic material properties. *Czech Aeronautic Proceedings,* No. 3, pp 65-67, 2007.
[6] Doubrava R, Oberthor M, Raška J, Bělský P, Doubrava K and Dvořák M. Verification and numerical simulation of advanced composite inlet in compliance of airworthiness impact requirements. *13th research and education in aircraft design conference*, Brno, Czech Republic, pp 7-14, 2018.
[7] HexPly® 8552. Product Data Sheet http://www.aerospares.hu/files/hexcel/hexply_8552.pdf, visited 20.11.2017.
[8] Iannucci L and Willows M. L. An energy based damage mechanics approach to modelling impact onto woven composite materials - Part I: Numerical models. *Composites: Part A,* No. 37, pp. 2041-2056, 2006.
[9] Menna C, Asprone D, Caprino G, Lopresto V and Prota A. Numerical simulation of impact tests on GFRP composite laminates. *International journal of impact engineering,* No. 38, pp. 677-685, 2011.