Impact damage and failure response of various aircraft structures under high velocity loading

Augustin Gakwaya, Jean-Charles F. Saint-Marc, Fatima Zohra El Bahri,

Department of Mechanical Engineering
Laval University, Quebec QC G1K 7P4, CANADA

Abstract: The paper present results obtained using Abaqus/Explicit software for predicting the impact damage and failure response of various aircraft structures under high velocity impact loadings. Two types of problems are considered. The first concern the crash simulation of a generic airplane fuselage section and the second concern study of bird strike on typical wing leading edge using both metallic and composite structure. In the case of an aircraft fuselage hitting the ground, both vertical crash test and steep glide slope crash conditions are considered. The main objective was then to develop a methodology for performing such fast transient dynamic simulation tests and to determine in each case the residual velocity as well the various deformation modes associated with particular loading and crash conditions. The second example is concerned with bird strike on airplane wing leading edge built from multilayered fibers-reinforced composite. Several features were used to optimize the results. A VUMAT subroutine containing all material properties and failure model equations describing the main damage mechanisms that can occur in the structure has been used. Contact definition is a critical issue in setting up an analysis definition of impact on a multilayered composite that can experience intra and interlaminar damage. Appropriate contact parameters have thus to be specified according to the model geometry and element type. A parametric study was considered by refining the model to study the convergence and establish accuracy of the process of modeling contact interaction between the wing leading edge and the projectile.

Keywords: Aircraft, Crashworthiness, Damage mechanics, Crack Propagation, Delamination, Failure, Design Optimization, Impact Dynamics, Shell Structures, Coupled Eulerian-Lagrangian analysis.

1. Introduction

High velocity impact modeling and simulation of aircraft structure is an active area of research. Currently, accurate and reliable results involve mastering of state of the art knowledge in material modeling and non linear computational mechanics technology [Fasanella, 2002]. Experimental tests for crashworthiness and bird strike development and certification are extremely expensive and time consuming. In order to reduce the number of costly prototype tests, a reliable analytical tool is necessary to accurately predict the structural responses/failures and to provide design guidance for aircraft subjected to the high-energy bird impact [Toussaint G. et al, 2004;].

The test results are useful in the appraisal of the structural performance under crash loading or high energy bird strikes, but do not reveal the various mechanisms that do and do not contribute to the overall performance of the structure [Hughes K et al, 2007]. Further, the performances of individual components may be influenced by the overall structural assembly. Hence to investigate the effectiveness of various components, numerical modeling would be more appropriate and less expensive. However, the predictions of the numerical models are dependent on the geometric definition of the structure, the material models, failure criteria, etc. The description of material behavior under dynamic loading is a key aspect of the numerical modeling of the crash scenarios.

The use of simulation thus provides the opportunity to cost-effectively evaluate numerous improved energy-absorbing structural design approaches that minimize structural weight and reduce the risk of not meeting civil or military aircraft bird strike design requirements. However, analysis techniques must be validated before they can be employed to accurately guide the design process.

Although a tremendous amount of knowledge has been gained and better understanding of the concepts relevant to impact simulation of metallic and laminated composite structure has been achieved in recent years, many topics, however, still require further investigation to allow for high level of confidence in the numerical modelling of high velocity impact on composite and metallic structures. While carrying out high velocity impact simulation, it is often noticed that existing failure modeling approximations in commercial explicit FE codes for both metals and composites have limitations that often make conclusive results difficult without development of testing. In fact, when erosion criterion is introduced as a solution to avoid excessive mesh distortion, it produced damages far from the evidences collected after real bird/debris impacts tests. Hence more systematic studies of factors such as element size, failure criteria, failure mode and computing expense that all interact to constrain the ultimate accuracy of explicit code predictions must be undertaken [Hughes K. et al , 2007; Chiara Bisagni, 2002].

In this paper, results obtained using Abaqus/Explicit package to predict the impact damage and failure response of some aircraft structures under high velocity loadings will be presented. Two types of problems are considered. The first concerns the crash simulation of a generic airplane fuselage section. Since only metallic structure is considered, it is taken as an entry level problem to allow one to set up a valid transient fast dynamics analysis and to lay up a methodology for further investigations involving composite materials in the Abaqus/Explicit environment. The second concerns the study of bird strike on typical wing leading edge using both metallic and composite structure. It is related to advanced composite design problem since it relies on the use of a VUMAT subroutine to study a progressive failure of a composite structure under high velocity loading causing damage and failure [Camanho, P.P et al, 2003; Pinho, S.T et al., 2005]. Both pure lagrangian and recent coupled eulerian-lagrangian(CEL) analyis procedures are applied.

2. Crash simulation of aircraft fuselage

Crashworthiness simulation study is related to occupant’s protection during crash situations. It is a relatively novel field which deals with how materials and structures deform, fail and absorb energy in a controlled manner during a crash event and have thus to be further explored. By controlled manner, we mean a deformation mode where the crushing force is kept to an approximately constant level during the collision, in such a way that a maximum amount of energy is absorbed at bearable levels of acceleration for the passengers [Hughes K. et al , 2007].

The crashworthiness of an airframe structure is thus measured in terms of its ability to maintain a survivable volume for the occupants and alleviate the loads transmitted to the occupants during potentially survivable accident scenarios per various FAR specifications. The occupant loads are minimized by dissipating the kinetic energy using an energy absorption device, while the structural integrity is maintained by accounting for the dynamic loads during the sizing of structural elements. The crashworthiness is in general evaluated for emergency landing on a hard surface, soft soil or water. The surface upon which the aircraft lands greatly influences the response of the structure and safety of the passenger. Typically, aircrafts are designed to land on a hard surface, which means that most of the loads are carried through the existent structure. However, when landing on soft soil or water, the skin absorbs most of the impact. Therefore structural integrity of the skin is critical, especially when ditching occurs in water and because of the complex mechanical behaviour of advanced composites. The existing analytical and numerical models to predict the crushing behaviour of composite materials are still of limited capability [Sareen, A. K et al ,2002].

The performance of an airframe under crash loads is thus dictated primarily by its geometry, structural arrangements, materials and energy absorption devices used to dissipate the energy, and the interaction of these variables. The energy dissipation in metallic airframes is primarily due to plastic deformation while in composite airframes it is due to synergistic sequence of various failure mechanisms. The limited number of dynamic and drop tests performed on fully composite fuselage structures have indicated differences in the crush patterns/failure modes, stiffness and other structural properties, compared with the traditional metallic fuselage structures. Our previous bird strike and crash simulations have been performed using mostly the Lsdyna software and we want to assess how Abaqus/Explicit can handle similar problems [Lavoie M.A et al, 2007].

The design goal here was to obtain a global knowledge of how an aircraft fuselage response to impact loading is critical in designing such structures. Even if little can be done in saving lives during a high velocity crash, it is nonetheless important to know which part of the structure to reinforce and optimize. So that, in the event of a catastrophic crash, the structure will provide maximum protection for its occupants. Thus it is useful to know how the fuselage will behave under certain types of solicitation in order to further optimize its overall design.

In order to give a clear idea of the basic behavior of the fuselage under crash loading and as a first approach to Abaqus/Explicit transient fast dynamics analysis, the problem has been simplified and reduced to an idealized model of a typical aircraft fuselage portion made of reinforced cylindrical shells and plates structures. The geometry of the fuselage will thus be created using beam (wire) and shell features.

2.1 Design methodology in Abaqus/Explicit

2.1.1 Geometrical definition of the model

For the geometrical definition of the fuselage section we used typical designs for the hoop-frame and floor which are given in figure 1.

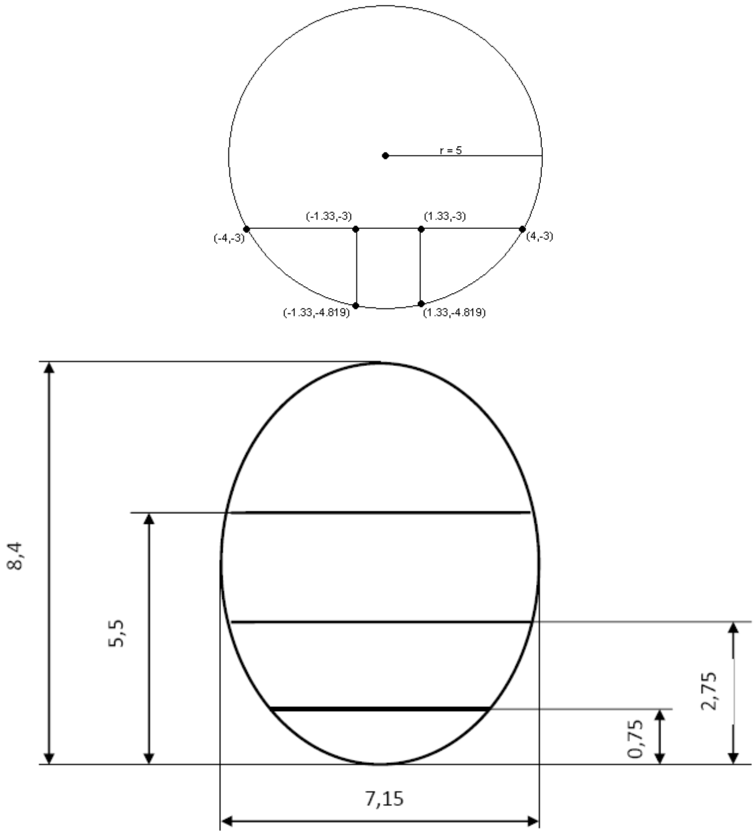


Figure 1. Typical aircraft fuselage sections geometry

Concerning the mechanical design of the fuselage, the floor, and the panels supporting the floor we used shell features. Then to reinforce the structure we added wire features. Hence shell and beam elements have to be created. The used material is aluminum with following properties (Young modulus: 69 GPa, Poisson ratio: 0.3, density: 2700g/m^3). The ground was defined as a surface which was set to be as undeformable as possible by giving it a very high Young modulus. Concerning the beam elements they are set up using the “Create section” tool in Abaqus/Explicit. The chosen shape is the I-shape to which we attributed specific dimensions. Typical beam section dimensions are shown in figure 2.

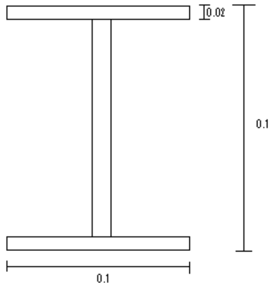


Figure 2. I-shape beam dimensions

Finally we obtain the whole model which contains all the geometrical properties needed and the material definitions for the aluminum structure and the ground. Concerning the relative positioning of the fuselage with respect to the ground, two crash scenarios were considered in order to observe the deformation modes of the fuselage. In the first case the fuselage is positioned parallel to the ground as shown in figure 3 and a vertical crash landing is monitored. In the second case we wanted to observe a 30 degrees steep glide slope fuselage crashing on the ground as it is illustrated in figure 4.

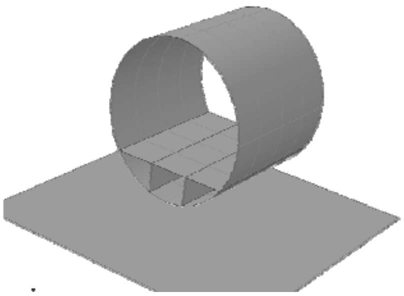


Figure 3. Vertical crash of the fuselage relative to the ground

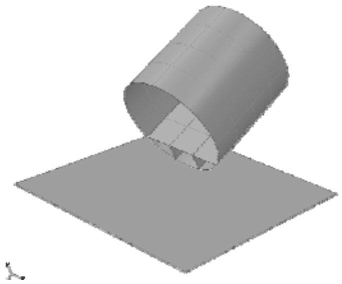


Figure 4. 30 degrees fuselage crash relative to the ground

2.1.2 Boundary conditions and contact definitions

Once the complete geometrical model is generated and the material and section properties have been assigned, we proceed to the definition of boundary conditions to be associated with the fuselage and the ground. A predefined velocity field was created and assigned to the whole fuselage. The used distribution is uniform and purely translational towards the ground surface. The value of the translational velocity was modified to test multiple crash speeds and observe the behavior of the fuselage. The deformation modes at a three different translational speeds of 10, 50 and 70 meters per second were considered.

Concerning the contact definition the “General contact” option was used since we will be working using Abaqus/Explicit for the crash simulation. This option greatly simplifies the definition of the contact model and of its parameters. It allows setting an interaction between every surface of the model without having to enter all the possible surface contact pairs. The interaction property we used is contact.

2.2 Results obtained

As an analysis algorithm, explicit dynamics option is chosen. Several crash simulation scenarios have been run. Figure 5 and figure 6 shows typical results obtained for a landing speed at 70 meters/second velocity for the two different angles.

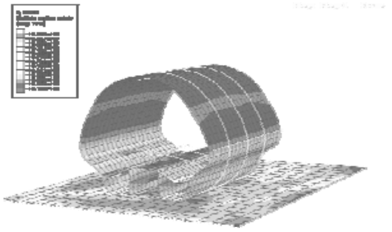


Figure 5. Vertical impact at 70m/s

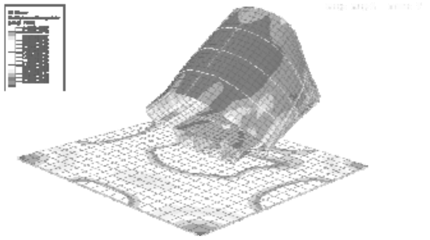


Figure 6. 30 degrees impact at 70m/s

3. Bird strike on composite wing leading edge

The second example is concerned with the design of a composite wing leading edge subjected to bird strike. The main goal here is thus to model a bird impact scenario on a wing's leading edge (LE) made of composite materials. The knowledge of the behavior of the wing LE under a high velocity impact load is a fundamental issue in designing an airplane. This analytical test is critical and enables engineers to optimize the layup of composite plies in order to withstand a given high energy bird impact. But the modeling of a composite layup using Abaqus/Explicit and the generation of accurate associated composite's damage behavior are not yet easy tasks. To obtain required and reliable information of the behavior of advanced multilayered composite material it is often necessary to create a user material subroutine. Thus, we introduced a VUMAT to define the various damage and failure modes and to perform the required progressive failure analysis.

Soft body impact is still a hot topic in aircraft safety requirement. But in order to reduce the number and costs of physical certification testings, numerical simulations are increasingly being used. The areas covered by recent research have been the assessment of the performance of different numerical methods versus experimental results and an investigation of the effect of the shape of the projectile. Both ALE and SPH formulation have been used in our previous work using different softwares [Lavoie et al 2008] but the goal here is to assess Abaqus/Explicit capability and user's friendliness in handling the same kind of problems.

3.1 Design methodology in Abaqus/Explicit

The definition of the model involves several steps. First it is necessary to create the wing leading edge as well as the bird model geometries. Then we have to define the material models for bird substitute and for the leading edge. It is at this step that the definition of the composite structure is important. Since no multilayered constitutive composite material model is readily available in

Abaqus/Explicit, the definition of the composite thus involves the creation of a VUMAT subroutine required to capture its damage behaviour under impact loading. The final step is to define the boundary conditions of the bird and the leading edge.

3.1.1 Geometrical definition of interacting bodies

The model has been created using shell features. We restricted ourselves to the definition of the leading edge portion of the wing where the bird strike effects are localized. In addition to modeling the outer surface of the leading edge we added six spars to reinforce the wing structure as it can be seen in figures 7 and 8.

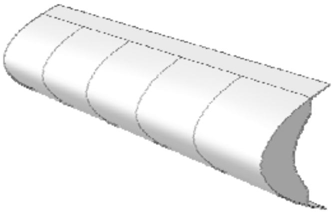


Figure 7. Front view of the leading edge

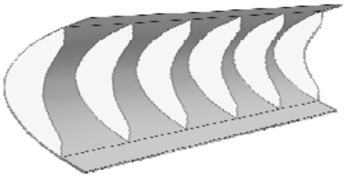


Figure 8. A view of the spars

The spars are made out of aluminum and the outer surface of the wing leading edge is a composite layup. The aluminum material is defined in the guest user interface of Abaqus/CAE in a classical manner by selecting the “Material” option and defining an isotropic material having the properties we already presented for the fuselage model. The definition of the composite layup is a new feature that is explained in the following paragraph.

In this paper, the bird is treated as a soft gelatine projectile whose geometry is modeled as a cylinder with two hemispherical ends, as shown in Figure 9. The ratio of the length to the diameter of the bird is selected to be 2:1. The weight of the bird is prescribed depending on the test conditions (4.0 lb for airplane mode and 2.2 lb for VTOL mode). The length of the bird, *L*, can be determined readily based upon the assumed density. Namely, *L* can be calculated as 4.438 inch (11.27 cm) and 2.43 inch (6.17 cm) for the airplane mode and VTOL mode, respectively. The bird models constitutive parameters have been validated through benchmark problem that simulates the soft gelatin bird impacting on a rigid steel plate is constructed in LSDyna [Lavoie M-A et al, 2008]. To define the bird geometry one can proceed as follows. First create a new part, select « 3D, Deformable, Solid, Revolution » and enter as « Approximate size ». In the Sketcher, define the profile as shown in figure 9.

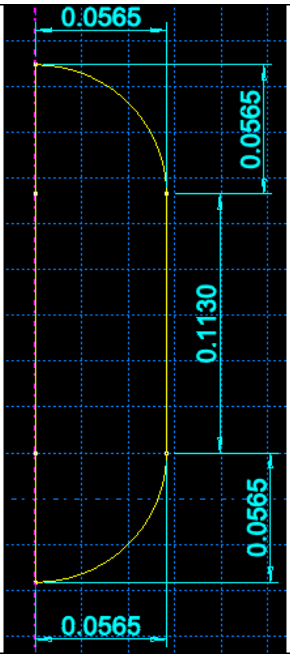


Figure 9. Profile of the bird geometric model

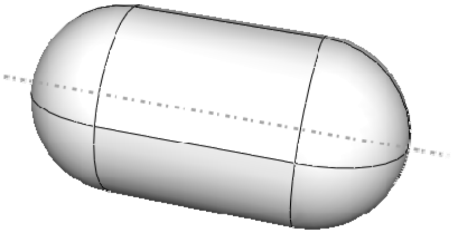


Figure 10. Bird geometry

The resulting solid geometry of the bird model is then presented in figure 10. For bird specific material properties, we use bird substitute made of gelatin with plastic hydrodynamic material model used by several researchers [M-A Lavoie, 2007]: density of 950, “Elasticity” with Young modulus, of 100e6 and Poisson ratio of 0.3. Under “Plasticity”, enter 1e6 as Yield Stress with 0 Plastic Strain.

3.1.2 Wing leading edge composite material layup definition

The next step is concerned with the design of a multilayered composite wing leading edge taking account of its damage behavior through a user subroutine. In Abaqus/Explicit, the definition of the composite material layup starts by defining the properties of the material using the “Material Manager” in the Property module. Then in the “Edit Material” window, we expand the “General” tab and select “User Material”. This user subroutine is used to define specific laws regarding the material elasticity and damage. Next we enter the values of the material properties specified in the VUMATt subroutine. In our case 19 material parameter values have to be entered in Abaqus/Explicit data file following a specific sequence as required by the user’s subroutine. The parameters to be entered are as follows:

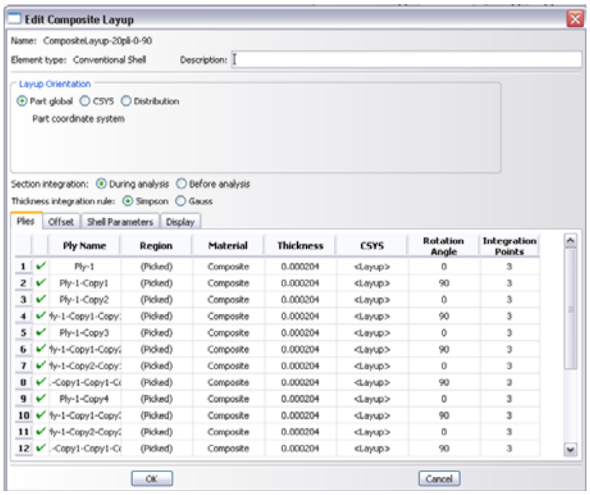
- Young's modulus in direction1, direction 2 and direction3 respectively: E1, E2, E3;
- Poisson's ratio, nu12, nu13 and nu23;
- Shear modulus, G12; G13 and G23;
- beta damping parameter;

- Ultimate tens stress in 1-direction, sigu1t and ultimate comp stress in 1-direction, sigu1c;
- Ultimate tens stress in 2-direction, sigu2t and ultimate comp stress in 2-direction, sigu2c;
- Ultimate tens stress in 3-direction, sigu3t and ultimate comp stress in 3-direction, sigu3c;
- Ultimate shear stresses, sigu12; sigu13 and sigu23.

An extract of the corresponding section in the input file concerning the definition of the basic ply material properties is given below:

```
*Material, name=Composite
*Density
 1570.,
*Depvar, delete=5
 17,
*User Material, constants=32
2.35e+11,  1.7e+10,  1.7e+10,     0.32,     0.32,     0.45,  4.5e+09,
4.5e+09, 2.5e+09,    1e-09,  3.9e+09, 1.11e+08,    5e+07,  2.4e+09,
2.9e+08,  2.9e+08, 1.2e+08, 1.37e+08,    9e+07
```

The second step is to define the composite layup; this can be done by selecting “Composite Layup Manager” also in the Property module. With this tool we can define the number of plies, their thickness, the orientation of the fibers for each ply and the region where to apply each one. One then has to click on “Create”, give a name to the layup and select the number of plies to be created and the element type one wants to use. Then in the next “Edit Composite Layup” window, one defines each ply of the layup as shown in figure 11.



Name: CompositeLayup-20pli-0-90
Element type: Conventional Shell
Description:

Layup Orientation
● Part global ○ CSYS ○ Distribution
Part coordinate system

Section integration: ● During analysis ○ Before analysis
Thickness integration rule: ● Simpson ○ Gauss

		Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	Integration Points
1	✓	Ply-1	(Picked)	Composite	0.000204	<Layup>	0	3
2	✓	Ply-1-Copy1	(Picked)	Composite	0.000204	<Layup>	90	3
3	✓	Ply-1-Copy2	(Picked)	Composite	0.000204	<Layup>	0	3
4	✓	'ly-1-Copy1-Copy:	(Picked)	Composite	0.000204	<Layup>	90	3
5	✓	Ply-1-Copy3	(Picked)	Composite	0.000204	<Layup>	0	3
6	✓	'ly-1-Copy1-Copy:	(Picked)	Composite	0.000204	<Layup>	90	3
7	✓	'ly-1-Copy2-Copy:	(Picked)	Composite	0.000204	<Layup>	0	3
8	✓	.-Copy1-Copy1-C	(Picked)	Composite	0.000204	<Layup>	90	3
9	✓	Ply-1-Copy4	(Picked)	Composite	0.000204	<Layup>	0	3
10	✓	'ly-1-Copy1-Copy:	(Picked)	Composite	0.000204	<Layup>	90	3
11	✓	'ly-1-Copy2-Copy:	(Picked)	Composite	0.000204	<Layup>	0	3
12	✓	.-Copy1-Copy1-C	(Picked)	Composite	0.000204	<Layup>	90	3

Figure 11. Edit Composite Layup Window

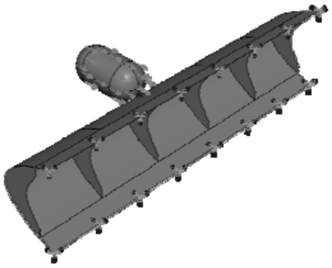


Figure 12. Boundary conditions for the impact simulation

In the introduced user’s subroutine, we assumed the material to have purely elastic orthotropic behavior at the beginning of the analysis, and since the failure modes of the matrix and the fibers are different, two different failure criterions were used in the evaluation. The three-dimensional Hashin criterion is used to predict the failure modes for the fibers; while the Puck criterion is used for the matrix failure [Puck, A. et al, 1998].

3.1.3 Boundary conditions and contact definitions

To define the contact between the bird and the wing leading edge is quite simple; as before we chose to use a "General Contact" definition for its easy set up. Using the pre-defined field options, the bird is given a velocity of 140 meters per second toward the leading edge. The wing is constrained in every direction and rotation of space and therefore cannot move. Figure 12 shows the assembled model and boundary conditions for a pure lagrangian analysis.

3.1.4 Element deletion

Since fully damaged elements do not provide any further strength resistance, they are thus deleted during the simulation. This is done through specification of "Element deletion" parameters to be defined in element control section of the Abaqus/Explicit input file as presented below.

```
ELEMENT CONTROLS
**
*Section Controls, name=EC-1, ELEMENT DELETION=YES, MAX DEGRADATION=0.5
1., 1., 1.
*Section Controls, name=EC-2, ELEMENT DELETION=YES, MAX DEGRADATION=0.5
1., 1., 1.
```

The value of maximum degradation is the maximum stiffness degradation and the element deletion occurs when degradation reaches this level.

3.2 Results obtained

3.2.1 Parametric study

A parametric study was conducted over several parameters defining the composite layup which are ply thickness, number of plies and fiber orientation. Thus we tested several possible layup by changing the fibers orientations, the thickness and number of the plies and observed the behavior of the wing under impact. Table 1 summarizes some of the results obtained.

Number of plies	**Fiber angle**	**Ply thickness**	**Maximum Von Mises Constraints**	**Logarithmic Strain Components at Integration Point (LE)**
20	0°	0,000204 m	1.501e09 Mpa	8.201e-02
2	0°	0,00204 m	1.369e09 Mpa	2.689e-02
20	0° / 90°	0,000204 m	1.988e09 Mpa	1.115e-02
2	0° / 90°	0,00204 m	1.816e09 Mpa	1.584e-02
40	0° / 90°	0,000204 m	8.059e08 Mpa	4.628e-03
60	0° / 90°	0,000204 m	4.621e08 Mpa	3.360e-03

Table 1. Results for various composite layups

3.2.2 Basic observations

We can observe that the 0°/90° is stronger than a simple layup made only of 0°. In addition the thickness of the plies is also a very important parameter, the thicker the ply, the stronger is the resulting layup. Also a layup with a large number of plies is capable of withstanding a more important impact without being damaged drastically as we can see in figure 13. In figure 13 and 14 we present the graphical results obtained for 40 plies and 60 plies.

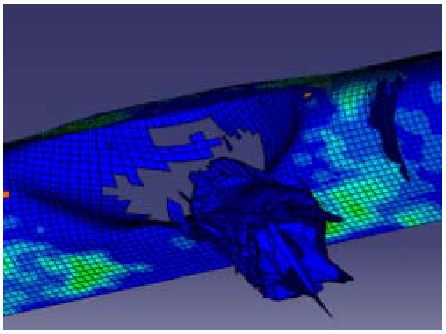


Figure 13. Results for 40 plies

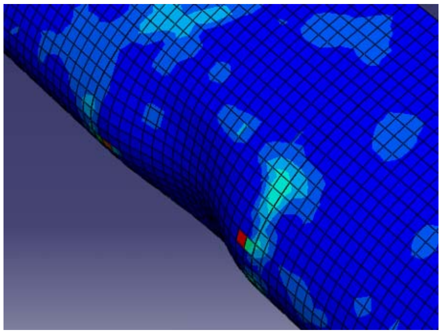


Figure 14. Results for 60 plies

In the graphical result shown in figure 13, we can see the bird going through the wing. The element deletion option is suppressing the wing's damaged elements. One must be careful in defining the element deletion option. If the "max degradation" value is too low then we obtain the result presented in figure 15 where shell elements are widely suppressed on the entire surface of the wing which is obviously not realistic indicating that the multi-scale nature of failure has not been well captured. The element erosion algorithm, with element deletion controlled by a certain local failure criterion, is an attractive technology that stems from its simplicity and its reduced associated CPU time since fracture often appears in highly deformed elements. Hence, provided that sufficiently small elements and adequate erosion criteria are used, the method is accurate. However there is one drawbacks stemming from the mass loss, whose effect is very severe for large elements, but could be partially circumvented by associating masses with nodes. Element erosion based on local critical stress or strain criterion suffers from mesh size dependency, but the energy-based failure criterion has shown to yield mesh size-independent results, but the associated path dependent integrals to be evaluated still represent a difficult task in 3D situations.

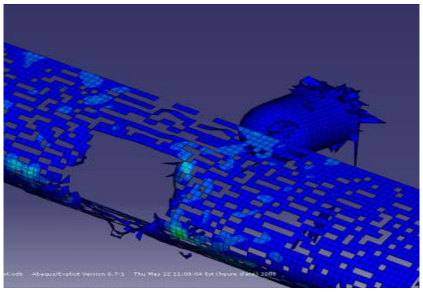


Figure 15. Element deletion parameter too low

3.2.3 Coupled eulerian-lagrangian analysis

Finally the methodology for coupled eulerian lagrangian analysis is developed and validated for a bird impact on metallic wing leading edge. Pure Eulerian analysis is a finite element technique in which materials are allowed to flow across element boundaries in a rigid mesh while in the Lagrangian technique, materials are closely associated with an element, and the materials move only with the deformation of the mesh. Because the element quality issues associated with a deformable mesh are not present in Eulerian analyses, the Eulerian technique can be very effective when treating problems involving very large deformations, material damage, or fluid materials. By combining the advantages of these two approaches, the Abaqus simulation functionalities have recently been extended and the Eulerian analysis technique can be coupled with traditional Lagrangian techniques in two ways. First, there is the Arbitrary Lagrangian Eulerian (ALE) adaptive meshing which is a technique that combines features of Lagrangian and Eulerian analysis within the same part mesh. Typically ALE adaptive meshing is used to control element distortion in Lagrangian parts undergoing large deformations, such as in a forming analysis [Gakwaya et al, 2007]. Then there is the new coupled Eulerian-Lagrangian (CEL) capability that allows Eulerian and Lagrangian bodies within the same model to interact. Typically a coupled Eulerian-Lagrangian analysis is used to model the interactions between a solid body and a yielding or fluid material, such as an Eulerian gas inflating a Lagrangian airbag or an eulerian bird impacting a lagrangian wing LE. In our bird impact analysis we created the wing leading edge using a lagrangian formulation because it is the stiffer body and it is expect that its elements will not be deformed too much. Concerning the bird which is highly deformable, it is modeled as a hydrodynamic eulerian material embedded into a eulerian domain surrounding the wing leading edge.

Compared to purely Lagrangian impact analysis of the previous section, the generation of CEL model in Abaqus is quite different. Briefly, to set up the analysis one has to add the following steps :

- Creation of the wing geometry using a « 3D deformable » part.
- Creation of the eulerian domain large enough so the bird material can flow through it.
- Assigning the bird material to a defined set of elements in the eulerian geometry using the « volume fraction tool ».
- Assigning the bird velocity boundary condition to a defined set of nodes.
- Launching the analysis and viewing the results using the « view cut » tool.

Then considering the same parameters and metallic wing LE for the two models, i.e. taking for the leading edge, a material's density set to 2780 and an elastic behavior defined a Young modulus set to 7.31E9 and a Poisson's ratio set to 0.33 with the wing surface shell thickness taken as 0.00203 and 0.00091 for the spars, a CEL analysis was successfully performed. Energy maps for pure lagrangian and coupled eulerian-lagrangian simulation are shown in figure 17 and 18, one can see there is a loss of energy in this preliminary results with the CEL model while energy is conserved in pure lagrangian model. An eulerian mesh refinement did not show any improvement. Here the computed energies (kinetic, internal, total) for the whole model were plotted. Hence as a result of dynamic equilibrium during the first instants of impact, material may flow out of the Eulerian domain upon contact with the lagrangian boundary. This material is lost from the simulation, and corresponding decreases in total mass and energy thus occur. Figure 19 and 20 show the deformed wing leading configuration obtained with the two analysis scheme. Similar trends are observed.

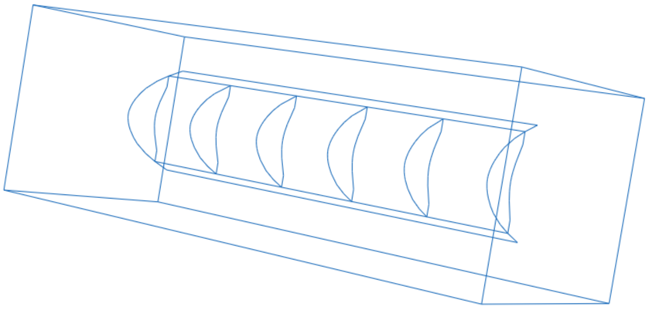


Figure 16: CEL model assembly

By the time being the CEL model is much larger then the pure lagrangian model, because in our case the eulerian medium was chosen so as to completely surround the bird and the wing LE. It is possible to reduce that size and thus reduce the time required for computations. Hence work is under way in order to reduce the model size and hence the computational costs.

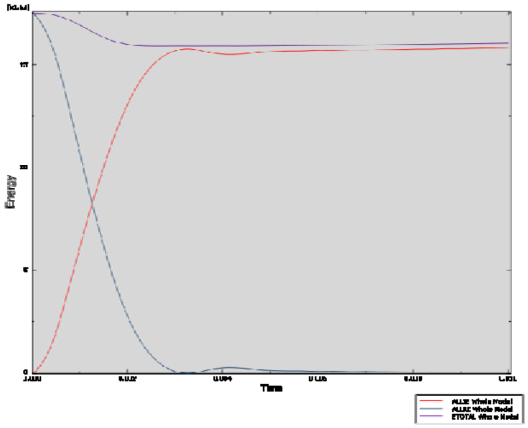


Figure 17 : CEL energy curves

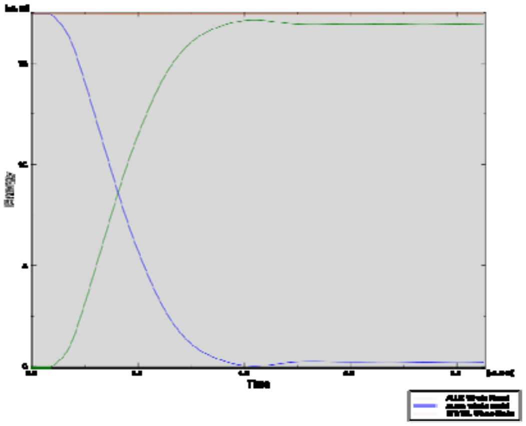


Figure 18 : Purely lagrangian energy curves

Concerning the stress distribution shown in figure 19 and 20 the stress level is higher in pure lagrangian analysis in conformity with observed strain energy level. The deformation is more concentrated in the Lagrangian analysis while it is more spread in CEL due to better bird flow around the wing LE. Due to approximation used in the CEL analysis, the CEL model is less accurate in this case because it is best suited for the analysis of a eulerian highly deformable bird impacting a rigid wing. It was also observed that the CEL model results regarding the bird deformation and the energy distribution do not depend on the mesh size.

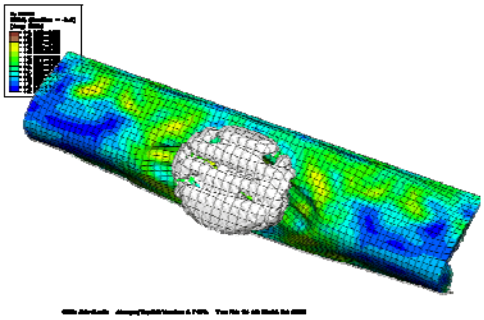


Figure 19 : CEL von Mises stress results

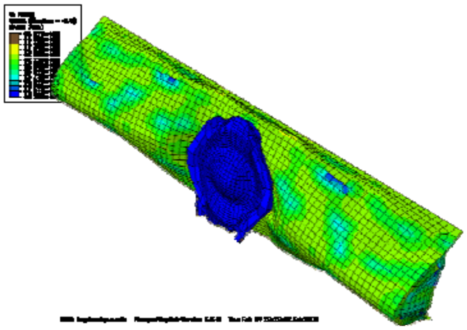


Figure 20 : Purely lagrangian von Mises stress results

Further work should be done in order to ensure a better global energy balance than actually as compared to pure lagrangian analysis.

4. Conclusion

Modeling and simulation of impact damage and failure response of various aircraft structures under high velocity loadings have been considered using both pure Lagrangian and new Coupled Eulerian-lagrangian feature of ABAQUS explicit . Two types of problems are considered. The first is related to the crash simulation of a generic airplane fuselage section. A metallic structure is then used as an entry level problem to allow one to set up a valid transient fast dynamics analysis and to lay up a methodology for further investigations involving composite materials in the Abaqus/Explicit environment. The second is related to the bird strike on typical wing leading edge using both metallic and composite structure. An advanced composite design problem relying on the use of a VUMAT subroutine to study a progressive failure of a composite structure under high velocity loading is then successively solved. The new coupled eulerian-lagrangian analysis was successively tested for bird impact on metallic wing leading edge but some loss of energy was observed as compared to pure lagrangian analysis. Further work is still required in order to include our composite VUMAT feature in the analysis and to computationally optimize the model. Results obtained demonstrate the suitability of the used software to handle such fast transient problems.

5. References

1. Fasanella, E. L., Jackson, K. E. *Best practises for crash modelling and simulation.* NASA/TM-2002-211944, October 2002.
2. Hughes K., Vignjevic R., Campbell J. *Experimental observations of an 8 m/s drop test of a metallic helicopter underfloor structure onto a hard surface: part 1.* Proc. IMechE Vol. 221 Part G: J. Aerospace Engineering, pp661-678, 2007.
3. Sareen, A. K., Sparks, C., Mullins, B. R., Fasanella, E. L., and Jackson, K. E., " Comparison of Soft Soil and Hard Surface Impact Performance of a Crashworthy Composite Fuselage Concept", Proceedings of the AHS Forum 58, Montreal, Canada, June 11-13, 2002.

4. Toussaint, G., Gauthier, L., Gakwaya, A., And Nandlall, D., Modeling and simulation of ballistic impact: application to fan blade containment problem and engine debris fuselage protection, REPORT 1: “Report of Literature Survey on Fan-blade containment and Debris protection”, CRIAQ3-1-IMCAS-T411abRP1-04, October 2004, 46pp.

5. Chiara Bisagni, *Crashworthiness of helicopter subfloor structures*, International Journal of Impact Engineering 27 (2002) 1067–1082.

6. Camanho, P.P., Dávila, C.G., and de Moura, M.F.S.F., "Numerical Simulation of Mixed-Mode Progressive Delamination in Composite Materials," Journal of Composite Materials, Vol. 37, No. 16, 2003, pp. 1415-1438

7. Puck, A. and Schürmann, H., “Failure Analysis of FRP Laminates by Means of Physically Based Phenomenological Models,” Composite Science and Technology, 58, 1045-1067, 1998.

8. Pinho, S.T., Dávila, C.G., Camanho, P.P., Iannucci, L., and Robinson, P., “Failure Models and Criteria for FRP Under In-Plane or Three-Dimensional Stress States Including Shear Non-Linearity,” NASA Technical Memorandum 213530, National Aeronautics and Space Administration, U.S.A., 2005.

9. Lavoie M-A., Gakwaya A, Nejad Ensan M. and Zimcik D.G., *Validation of Available Approaches for Numerical Bird Strike Modeling Tools*, International Review of Mechanical Engineering, Vol. 1, no4, pp380-389, 2007

10. Lavoie M-A, Gakwaya A. , Nejad Ensan M., Zimcik D.G., Nandlall D., *Bird’s substitute tests results and evaluation of available numerical methods,* submitted for publication, 2008

11. Gakwaya, A. Guillot, M., Levesque J., “*Application of Finite Element ALE Formulation in Cold Forming with ABAQUS*”, in proceeding of International Conference on Sustainable Manufacturing, NRC-IAR, Montréal, October 2007, pp158-172