ASSESMENT OF EXPLICIT F.E. CAPABILITIES FOR FULL SCALE COUPLED FLUID/STRUCTURE AIRCRAFT DITCHING SIMULATIONS

R. Ortiz, G. Portemont, J.L. Charles, J.F. Sobry
ONERA/DMSE
Structure and Damage Mechanics Department
5, Bd Paul Painlevé – F59045 LILLE Cédex

Keywords : *Crash, aeronautical structures, ditching modeling, ALE, SPH*

Abstract

The presented work is being done within EU CRAHVI G4RD-CT-2000-00395 programme, with the support from the European Community[1]. Differently to the previous EU "crash" programmes dealing with aircraft safety (restricted to vertical 10 m/s crash speed), the current one is more dedicated to high velocity problems, meaning in our case that we now are taking into account the horizontal 50 m/s speed. Another specificity of the programme is that it is concerned with problems such as bird, debris or ground obstacles impacts, and crash on rigid or soft soils.

The proposed paper presents hard landing simulations of large aeronautical structures using explicit F.E. codes, such as RADIOSS. The main technical difficulties arise from the dimensions and the complexity of the structures to be modelled on the one hand, and from the complexity of the very local ruin phenomena (rupture of material or failure of riveted joints) on the other hand. When we study aircraft behaviour, it becomes necessary to consider different configurations (nature and stiffness of the impacted surface). An aircraft could crash on the ground or on the water.

The present work deals first with some experimental work which has been performed at ONERA (in the frame of the EU BRPR-CT97-0464 SEAWORTH Programme), and their modeling with various F.E. methods. These laboratory tests aimed at the starting/completion of a database on basic water impacts on simple geometry or configurations [2]. The simplicity of these tests is of great interest to confront the numerical approaches which are under development (lagrangian, ALE and SPH) in the field of fluid/structure interactions[3]. The other aim is to improve confidence and assess representativness of models for hard ditching simulations.

In the second part, the document describes crash simulations of a complete commercial aircraft model which is being built up by ONERA and AIRBUS France. The objective is to valid the numerical capabilities and computer need to solve coupled fluid/structure problems like ditching, which were still considered to be out of reach some years ago.

The objective of such models is also to run parametric cases (speed and orientation of the aircraft relatively to the impacted surface, for instance) to evaluate the influence of these parameters on the structural behaviour and loads transferred to the cabin environment and passengers, in case of hard landing. The specificity of this study is to determine structural loading of a complete aircraft under realistic crash conditions, and more particularly relatively to the impacted surface stiffness (rigid or water) and generate a load database (accelerations, velocities, displacement, forces) for the cabin environment which can be used for the design of innovative cabin safety features with the aim to improve passenger safety. Another interest is to study the failure of secondary structure and components inside the cabin with the aim to improve passenger survivability and reduce fatality rates[4].

1 Background data for basic water impact tests

1.2 Experimental simplified tests

In the SEAWORTH programme the ONERA work consisted in testing impacts on simple geometries or configurations. The simplicity of those tests was mainly required in order to enable 2D analyses with the different partners numerical tools. The general issue was to evaluate the available methods which could give access to some relevant knowledge about fluid-structure interactions. ONERA/DMSE had to perform some 2D simulations and is now trying to complete its study with 3D simulations. Thus, these tests permit to confront different numerical approaches (ALE and SPH) which are of great interest in the case of ditching simulations. ONERA chooses to perform simulations with the dihedral specimen used in the SEAWORTH programme.

1.3 Selection and definition of simplified experimental tests

1.3.1 Specimens geometry

Owing to the fact that the purpose of those tests is to validate the numerical tools, a 130° dihedral shape has been chosen with respect to the difficulties which can appear in our future studies (hard ditching, contact problems ...).

Concerning boundary effects, the overall dimensions of the rigid impactor are much larger that those of the instrumented flexible area. For instance, on ONERA dihedral past experiments, the size of the rigid impactor was :

- Length = 450 mm,
- Beam = 300 mm.

The size of the instrumental flexible specimen was :

- Length = 150 mm
- Width = 75 mm

1.3.2. Tests instrumentation

A single measuring plate element was set on the rigid frame. Each specimen being instrumented with one accelerometer, natural frequencies have been checked (in air) to verify that the accelerometer has been correctly placed.

1.4 Comparison of numerical and experimental methods

2D models were derived in Lagrangian, Eulerian and in ALE formulations, with fluid, structure and boundary element codes. The discrepancies observed throughout the results made them difficult to analyse. Furthermore, these works enabled to show a major difficulty in comparing calculations, linked to the pertinence of the experimental procedures and the interpretation of the measurements.

At the end of these works, the DMSE Department decided to continue, in the EU CRAHVI project, and complete the evaluation already begun, considering the recent developments of the simulation codes (dealing with the 3D problem and including the Smooth Hydrodynamic Particle approach).

1.4.1 Dihedral impact tests

All tests were carried out at ONERA Lille by the DAAP/MMH unit. This part presents the results of the dihedral impacts. The geometry was chosen with an opening angle of 130°. Different specimens, on which the whole of the instrumentation is set, are fixed to the dihedral geometry. The specimens are made of different materials (aluminum and steel) and different thicknesses (0.5, 1, 2 mm). These specimens (Figure 1) are equipped with six pressure transducers (KULITE XTC-76A-190M), and nine strain gauges and one accelerometer (ENDEVCO).

The rigid frame is fixed on a hydraulic jack (figure 1). The impacts of the model were studied on a calm lake for different speeds (1 m/s, 2 m/s).

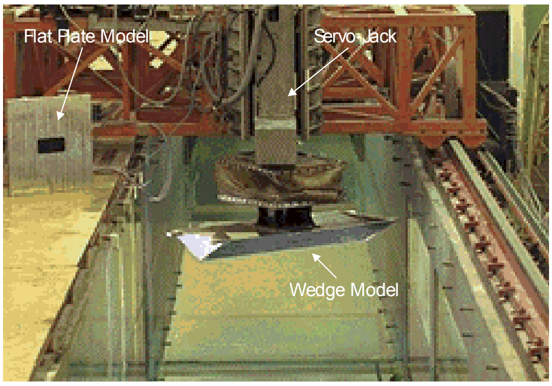

Figure 1 : Hydrodynamic tunnel and servo jack

1.4.2 Assessment of numerical methods

There are several ways to deal numerically with the problem of fluid/structure interaction :

(i). The use of a FE fluid code purely eulerian. This type of codes simulates the flow around rigid shapes or boundaries. They solve the Navier - Stockes equations and offer different options, for example, taking into account biphasic media, viscosity, laminar or turbulent flows [5].

(ii). The use of solid FE codes. They do not deal with the simulation of complex fluid flows, but they are particularly well suited to deal with the solid mechanics. They solve the propagation wave equations in the continuum media.

(iii). The coupling of fluid and solid codes. Nevertheless, this type of coupling requires two FE codes.

(iv). Some of the solid codes are able to deal with mixed formulations like lagrangian–eulerian (ALE) or SPH/Lagrangian, which give an approximate simulation of certain types of non turbulent flow, around structures slightly deformable, but fixed [6,7,8].

The dihedral shape is equipped with rigid specimens so that a comparison of the fluid and solid codes can be done on a common basis. The impacts are performed at a 2 m/s speed. To simplify the problem and reduce the computing costs all the simulations are bidimensional.

Figure 2 shows the differences observed between the numerical and experimental methods. The mesh allows to compare the measured pressure with the calculated one at the same specimen locations.

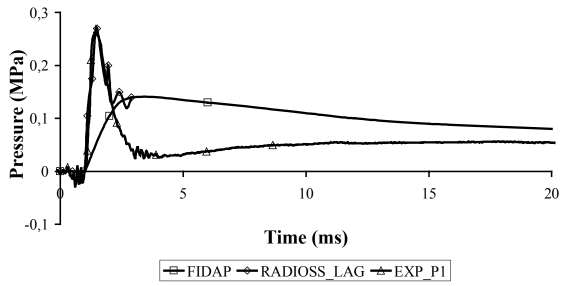

Figure 2 : Numerical/experiment comparisons

A qualitative comparison of the tests can be undertaken with the numerical simulations. The pressure peak at impact is more easily correlated by the solid lagrangian approach whereas the stagnation pressure is better represented by the fluid eulerian method.

2 3D Dihedral impact simulations

2.1 Introduction

The hydrodynamic tunnel is modelled to observe the influence of boundary conditions (reflected waves) on the F.E. simulations. This case is considered to be more representative of the difficulty that should bear the complete A321 ditching modelling (coarse mesh, contact interface, mesh deformation).

A comparison between ALE and SPH models has been carried out. The SPH meshless method is of great interest for simulations because :

1. There is no stability problems
2. The mesh deformation observed in ALE is avoided.

The air has not been modelled. ALE and SPH are compared with the same behavior law for water (Hydrodynamic viscous fluid).

The fluid is viscous and compressible.

$$S_{ij} = 2\rho\nu\dot{e}_{ij} \qquad (1)$$

S_{ij} is the deviatoric stress tensor. e_{ij} is the deviatoric strain tensor.

$$p = C_0 + C_1\mu + C_2\mu^2 + C_3\mu^3 + (C_4 + C_5\mu)E \quad (2)$$

p is the pressure, μ the viscosity coefficient, E the specific energy.

2.2 ALE Simulation

2.2.1 Hydrodynamic pool modeling

Figure 3 shows the half ALE model used for the complete dihedral impact test case with the hydrodynamic tunnel modelled.
The ALE approach requires the definition of boundary elements called "outlets", used classically in the modeling of fluid/structure interaction. These elements are set in order to absorb the reflected waves at the limits of the model. Thus, they permit to reduce the size of the model.

The law used for these specific elements is the following :

$$\frac{\partial P}{\partial t} = \rho c\left(\frac{\partial}{\partial t}(V_n) - V_n.div(\nabla - V_n.n)\right) + c\frac{(P_\infty - P)}{2l_c}$$

where c is the sound speed in the material and l_c a characteristic length.

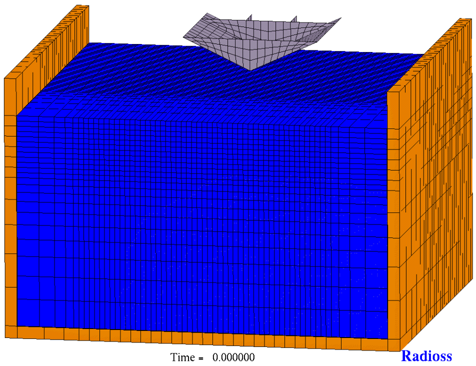

Figure 3 : ALE tunnel model

The model contains :

- 78479 nodes
- 10 materials
- 70586 brick elements
- 392 shell elements.

The Lagrangian-ALE tied interface has been validated in the first 2D simulations. So, the interest is now focussed on the observation of the ALE mesh distortion. It is interesting to know what level of distortion this kind of model can bear without trouble.

2.2.2 Simulation results

The following figures present the results of the simulation. The pressures within the fluid (water) are shown (figure 4 and 5).

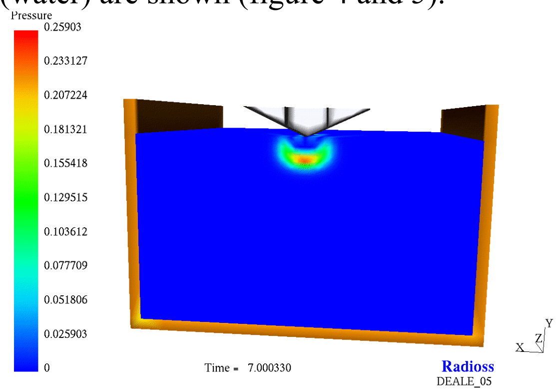

Figure 4 : ALE Pressure (t=7 ms)

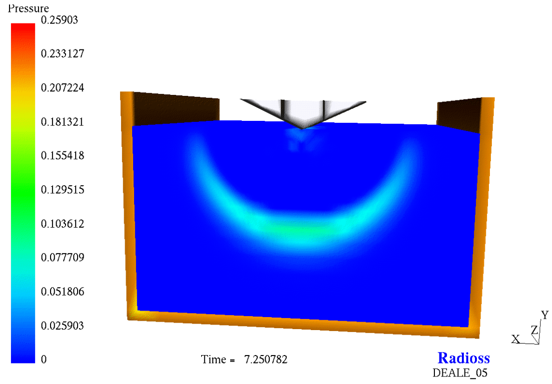

Figure 5 : ALE Pressure (t=7.3 ms)

It is possible to notice that the waves propagation is correctly represented. The level of pressure and distortion are in good agreement with the expectations.

The following figure 6 presents the deformation of the ALE mesh at the maximum penetration of the impactor.

We can also note that this type of modeling permits to simulate the wave and bears an important mesh deformation. This simulation could possibly suit to the case of the complete aircraft ditching. We can point out that the introduction of air will not be a major complication (bi-phase law) but will probably increase the computing costs.

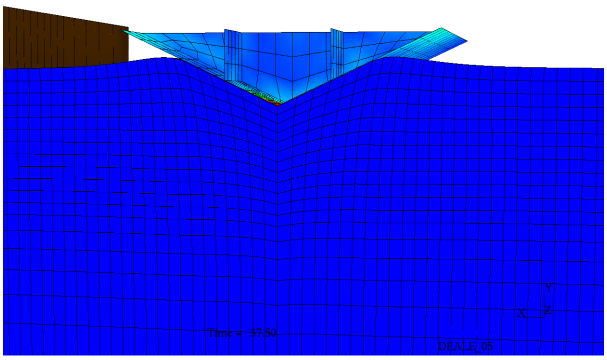

Figure 6 : Formation of the wave (ALE solution)

2.3 SPH simulation

2.3.1 Hydrodynamic tunnel modelling

Because of the size of the model, and the weak influence of air in the case of the dihedral impact problem, the air has not been meshed.

The complete dihedral impact test case has been achieved using the SPH method. This method can become very heavy in terms of computing costs if all the domain is modelled by particles. For this reason, only a part of the area is meshed with SPH, the rest of the domain being modelled by lagrangian elements brick (figure 7). These “two meshes" are tied through an available lagrangian interface in Radioss. This model contains :

- 82595 nodes,
- 29988 particles,
- 44186 brick elements,
- 392 shells.

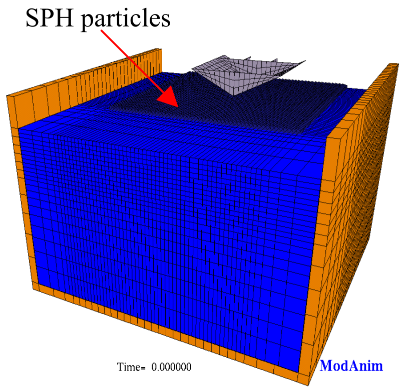

Figure 7 : Hydrodynamic tunnel modeling (SPH)

2.3.2 Simulation results

The following figures show the results of the SPH model. The results are very close to those observed with the ALE method (qualitatively). The analysis of the wave propagation and the values of pressure are in the same order of magnitude as the experimental observations (impact pressure near 0.3 MPa). In comparison with the ALE method, the pressure field looks more diffuse in SPH. In spite of an easy use, the SPH method remains very heavy because of the number of particles (the convergence of the calculations depends on the number of neighbouring particles). If this number is not enough the results will be incorrect. In the case where this number would be too important, the calculations will become too long.

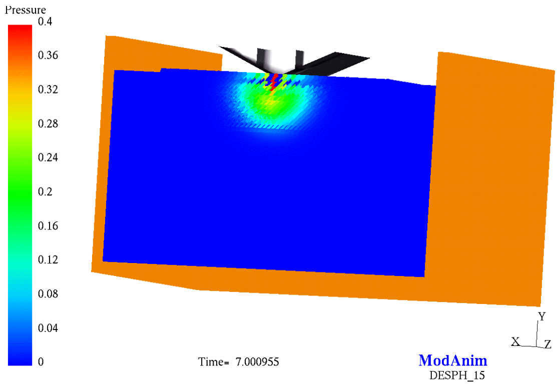

Figure 8 : SPH Pressure (t=7ms)

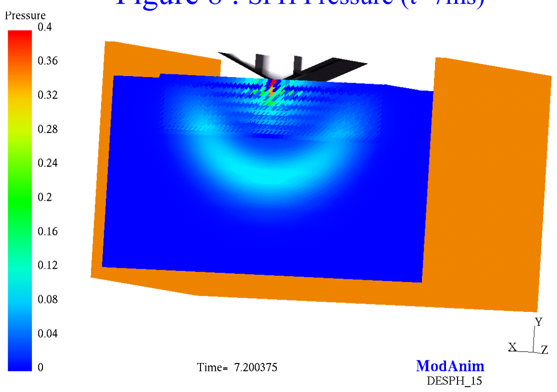

Figure 9 : SPH Pressure (t=7.2 ms)

Nevertheless, this SPH method seems to suit well to the modeling of the ditching scenario. The absence of mesh distortion would permit to reach the time necessary to model the complete ditching phenomenon. However, such an approach requires to mix a F.E mesh with the SPH particles to achieve a complete model.

3 Analogy with Ditching problems

Some general information is given concerning geometrical or structural characteristics of helicopters/aircraft structures and ditching:

- mass: from 1 up to hundreds of tons,
- geometry/dimensions: from 1 up to tens of meters
- horizontal speed: from 0 m/s up to 60 m/s,
- vertical speed: about 10 m/s,
incidence: from 0° to 15°,
- Maximum longitudinal acceleration : 2g,
- Maximum vertical acceleration: 3g,
duration: from 10 ms (first impact) up to 10 s,
- materials: aluminium or composites,
- thickness: from 1 up to 10 mm.

A complete Airbus A321 model has been built up by ONERA and Airbus France, making the simulation of different crash scenarios possible and realistic enough. However, this model can't permit to represent non local linearity.

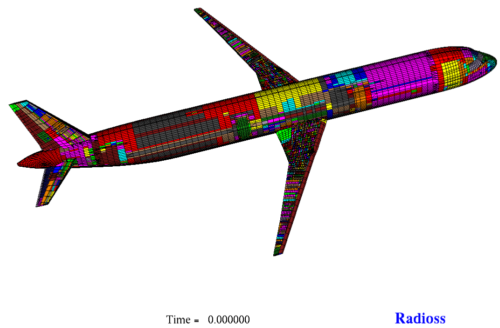

Figure 10 : Airbus A321 mesh model

Once these first simulations achieved, it is possible to think about the feasibility of the complete Airbus A321 ditching simulation. The first model relies on the ALE method. Indeed, at a first step, we want to verify and to confirm that this method is still applicable to a full aircraft structure simulation. The difficulty of this method concerns the definition of the tied interface between the aircraft and the air (deformation and negative volumes on the interface elements). The law used in the ALE modelling (for the air and water) is a biphasic law. The choice of this biphasic law is to take into account in our future works possible air cushion effects.

3.1 Hydrodynamic Biphase Liquid Gas

A biphasic material can contain gas (air) and liquid (water) in various proportions. The following equations show this material law used in the Radioss code.

1. Viscous stress tensor :

$$\sigma^{V}{}_{ij} = 2\nu\left(\dot{\varepsilon}_{ij} - \frac{1}{3}\delta_{ij}\dot{\varepsilon}_{ll}\right) + \zeta\delta_{ij}\dot{\varepsilon}_{ll} \qquad (3)$$

with $\dot{\varepsilon}$: strain rate tensor
ν : cinematic viscosity of the fluid

ζ : volume viscosity
δ : Kronecker symbol

2. Fluid equation of state

$$\Delta p^l = \rho^l{}_0 . c^2 \left(\frac{\rho^l}{\rho^l{}_0} - 1 \right) \quad (4)$$

with p^l : pressure in the fluid
c: sound speed in the fluid
$\rho^l{}_0$: initial density of the fluid

3. Gas equation of state

$$\Delta p^g = p^g{}_0 \left(\left(\frac{\rho^g}{\rho^g{}_0} \right)^\gamma - 1 \right) \quad (5)$$

with γ : perfect gas coefficient

4. Equilibrium:

$$p^l = p^g \quad (6)$$

3.2 Ditching configuration

A usual configuration of approach taken by the pilot during ditching is the following:

- Pitch of plane: 9°
- Impact velocity: 50 m/s

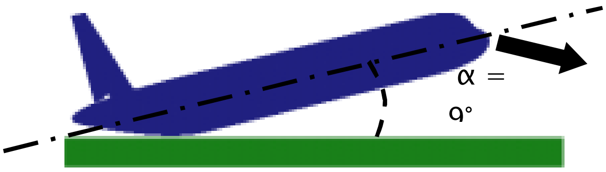

Figure 11 : Ditching configuration

3.3 ALE model

The following model represents the modeling of the Airbus A321 ditching. The domain "has been lengthened" therefore using the “outlets elements” presented before.

The ALE Airbus A321ditching model contains:

- 171431 nodes,
- 134090 brick elements,
- 11840 shells elements (quadrangle),
- 1488 triangles,
- 3728 beams,
- 213 springs,
- 187 materials,
- 7725 geometric properties.

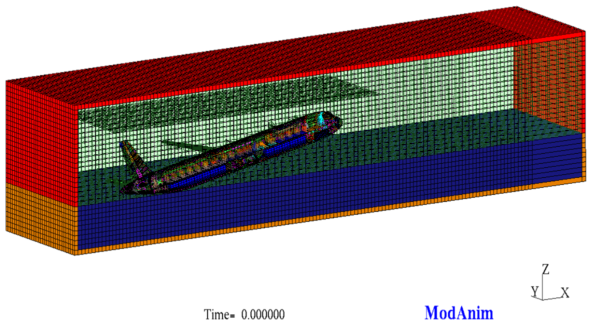

Figure 12 : ALE Model

3.4 SPH model

As for the coupled Lagrangian/ALE modelling, a Lagrangian formulation is kept for the aircraft. Only fluids (air and water) will be modelled by SPH particles. The SPH modeling is actually a Lagrangian formulation where the observer follows a number of particles in movement (part of a fluid or a solid). Indeed in a SPH modelling, a continuous medium is represented by a finite number of particles.

The advantage of the modeling of a continuous environment with SPH particles stands in the fact that it is not necessary to mesh the domain with finite elements. However it is important that the distribution of the particles is uniform. For it one can refer for example to an atomic distribution. In our case, a cubic distribution was chosen.
The SPH model contains :

- 171431 nodes
- 49005 particles
- 1980 brick elements

- 11840 shells
- 1488 triangles
- 11732 beams
- 213 springs
- 184 materials
- 7724 geometric properties

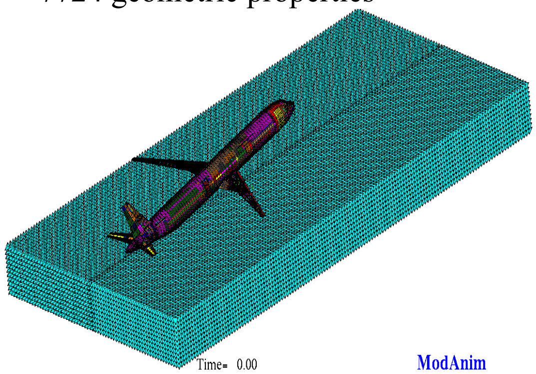

Figure 13 : SPH Model

3.5 ALE-SPH comparison

3.5.1 Deformation- Strength

In order to analyse the influence of the impacted surface on the structural deformation, specially in the impact zone, a comparison with a simulation on hard soil has been done (Figure 14).

One could notice in a first time that in the 3 cases (hard soil, water impact -ALE and SPH-), the deformation of the aircraft is localised in the impact zone.

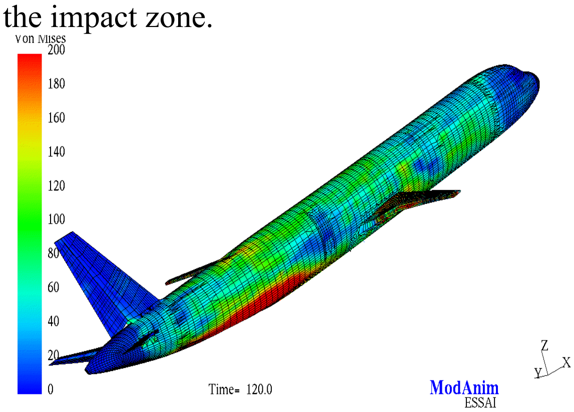

Figure 14 : Von Mises Strength (Crash)

The deformations are distinctly less pronounced for the ditching simulations (Figure 15 to 16). We can see between the hard soil crash and the ALE ditching simulation that the amplitudes of the stresses as well as their localisation are very similar.

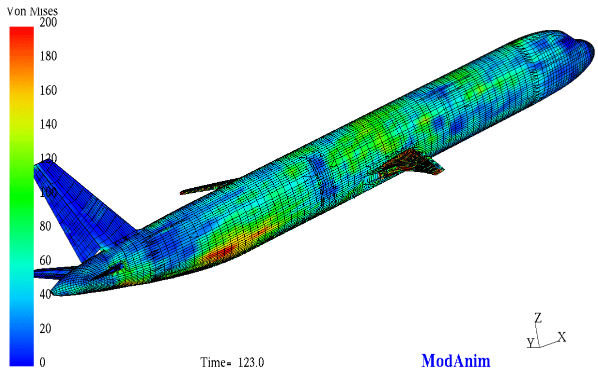

Figure 15 : Von Mises Strength (ALE)

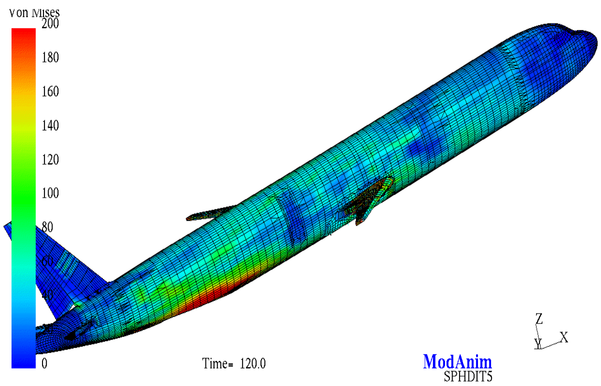

Figure 16 : Von Mises Strenth (SPH)

3.5.2 Pressures distribution

We can see that the air is voluntarily hidden in our representation to better observe the aircraft. The pressures observed in the SPH simulation (Figure 18) are raised higher than those observed in ALE (Figure 17).
On figures 17 and 18 (SPH and ALE modeling), at 20 ms, a decreasing gradient of pressure is clearly observed for the ALE modelling. This gradient is not observed in the SPH model. This difference comes from the use of two different laws (biphase for ALE and hydrodynamic for SPH).

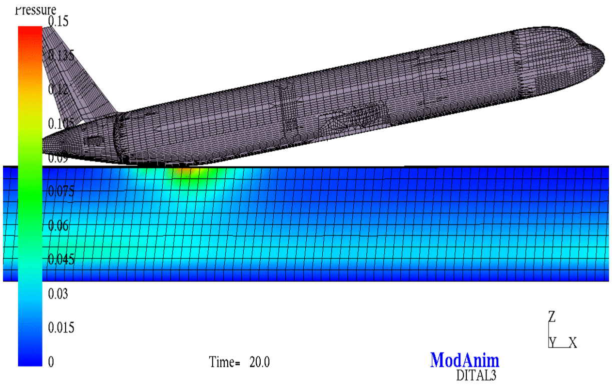

Figure 17 : Pressure t=20 ms (ALE)

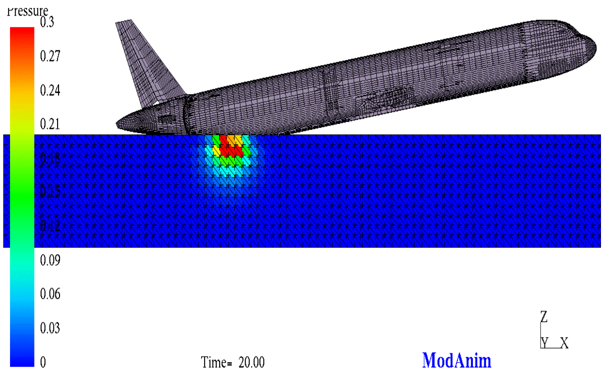

Figure 18 : Pressure t =20 ms (SPH)

At 60 ms (Figures 19 and 20) the pressure wave propagation in water is qualitatively well described (scale pressure is reduced to see clearly the propagation of waves). Some numerical instabilities are observed in the ALE simulation.

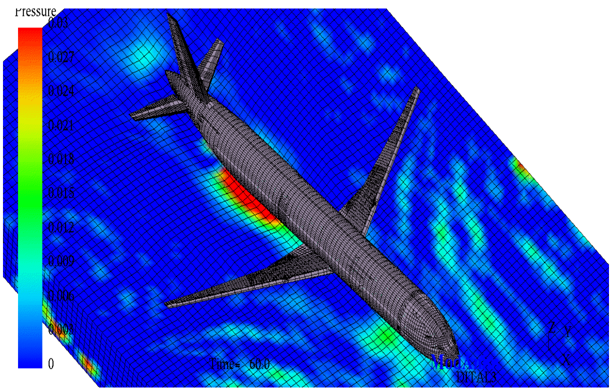

Figure 19 : Wave Propagation (ALE)

The observed pressures seem to be low. In fact, our mesh or particle size are too large (brick of 690 mm, pressure around 0.12 MPa for ALE and 0.25 MPa for SPH). Considering the wet surface, this pressure seems to be realistic. However, a refine mesh was required but would give a CPU time too long.

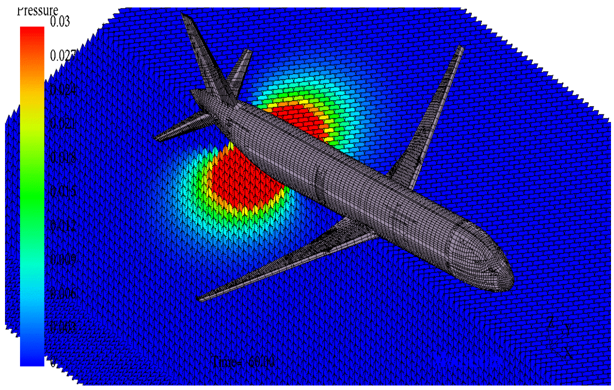

Figure 20 : Wave Propagation (SPH)

3.5.3 Accelerations

Figure 21 shows the accelerations calculated on the passenger floor for the three different simulations (hard soil, ALE and SPH). The accelerations are more important for the hard soil crash than for the ditching scenarios. However, the passengers support longer accelerations in the ditching case. Thus, these accelerations can produce more important damages than those supported in the case of the hard soil crash.

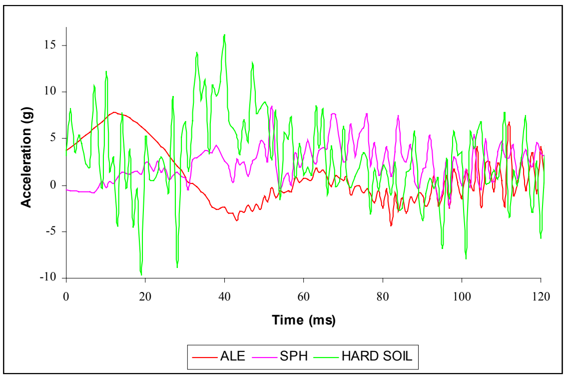

Figure 21 : Passenger accelerations

3.5.4 Analysis of results

Otherwise it has also been noted than not only the deformation but also the strength are always higher in SPH than those obtained in ALE. This difference could come from a bad concordance between the "step mesh" of the SPH particles, the concentrated masses that one assigns to each of it, as well as particle density. If a simply logical formulation of the particle mass m_p is used (with $m_p = \frac{\rho V}{nb}$), a local overestimation of the mass can occur (ρ is the density, V the volume and n_b numbers of particles in V). Using this formula with the SPH "mesh", some heavy particles are able to deform meaningfully the structure. One of the possibility to solve this problem would be to refine the SPH "mesh". Unfortunately the CPU Time is important in SPH simulation. For this reason the modification of the mass or density of particles is chosen. So the mass and density become henceforth parameters.

4 Conclusion and perspectives

As it has already been mentioned before, the final goal of this work was to show that a numeric modelling by finite elements or SPH approach for ditching simulation was foreseeable. Indeed, thanks to the considerable progress accomplished in the means and the computer tools, notably with the apparition of more and more powerful computers, one could consider for example for the ALE model to increase the size of the fluid area without penalising the time of calculation. It would be possible to refine the distribution of particles (SPH approach) without having serious consequences on the CPU time.

The time necessary to obtain all the ditching phenomenon (around 500 ms for 300 hours of calculus) is foreseeable with the SPH approach. The ALE approach gives quickly some numeric instabilities as negative volume (deformation of the mesh too important). The present advance works on the modeling and numeric simulations are very promising, more especially in the domain of the Finite Element and SPH method. Others works are necessary to validate quantitatively the biphasic models and SPH fluid modelling and are in progress at ONERA/DMSE.

References

[1] Ortiz R., Portemont G. *Report on 3D F.E. simulations of the background data with ALE and/or SPH methods.* Rapport ONERA-Lille CRAHVI, RT 3/05368 DMSE, February 2002.

[2] Flodrops J.P., Deletombe E. *Improved ship design for marine safety : Extreme load effects and hydroelastic coupling.* Rapport ONERA-Lille SEAWORTH BRPR-CT97-0464, classification Code: TEC-C032-01, 1999.

[3] Portemont G., Ortiz R., Deletombe E., Markiewicz E., Drazétic P. *Methodology for comparison of the contact pressure between experiment and FE models for coupled fluid/structure problems*. SUSI 2002, May, Montréal.

[4] MalherbeB, Langrand B, Charles J.L, Sobry J.F. Improvement of crash models of large aeronautical structures. 22 nd ICAS Congress, Harrogate (Great-Britain), August 28 September 01.

[5] Le Roy JF, *Simulation of impact of two-dimensional bodies in water,* Rapport Onera-Lille N°RT99/17/DMSE/Y/DAAP, 1999.

[6] P.W Randles, *Smoothed Particle Hydrodynamics: Some recent improvements and applications*, Computer methods in applied mechanics and engineering, 1996. [6] Monoghan J.J. Smoothed particle hydrodynamics. *Annu. Rev. Astron. Astrophys*, vol.30, pp.543-574, 1992.

[7] Swegle J.W., Hicks D.L., Attaway S.W. Smoothed particle hydrodynamics stability analysis. *Journal of computational physics*, vol. 116, pp.123-134, 1995.

[8] Randels P.W., Liberski L.D. Smoothed particle hydrodynamics : some recent improvements and applications. *Comput. methods apll. mech. engrg*. vol. 139, pp.375-408, 1996.