# Fluid-Structure Interaction simulation of parachute dynamic behaviour

Christine Espinosa[*] Yves de Lassat de Pressigny[†] and Pascal Bordenave[‡] Luke Henke[§]
*DGA, Toulouse, 31000, France*

During the late 90', CAP (now part of CEV, the French DoD's flight test centre) started developing fluid-structure interaction (FSI) methods for the numeric simulation of parachutes, with a complex coupling of two independent, implicit codes. ENSICA had a strong experience of explicit numeric simulation and proposed in 2002 to evaluate LS-DYNA as an emerging solution for parachute simulation[1]. ENSICA first focused its work on the numeric conditions for successful simulations : element hypotheses and material models available, meshing limitations and constraints, stable FSI formulation, results available. CEV provided then only analysis cases. In 2003, CEV acquired the same software an started running simulations with a new purpose : its computing power was not enough to model parachutes and airflow with the required precision and current techniques (ENSICA had had to restrict to a quarter model with symmetry conditions) ; the work was then oriented to prototype or specify new simulation features that would eventually lead to the full scale simulation of a parachute or event a cluster of parachutes. The main result of its work was a new formulation of the fluid properties and boundary conditions that made possible the use of non-parallelepiped fluid element, allowing the minimisation of the fluid elements count with a butterfly-type mesh. The simulations of full model parachutes were then fast enough to allow several simulation cases a month, with acceptable if not maximum accuracy. One of these simulations was then used to demonstrate the necessity to develop a new FSI feature in the finite element analysis software : initiates of the FSI initially thought the existing porosity simulation feature was suitable for parachute simulation. In fact it had been designed for airbag simulation, closed and surrounded by ambient atmosphere, which allows to delete the leaking gas from the simulation. In parachute simulation it is necessary to restore the leaking gas on the outer face of the fabric. In 2004, CEV demonstrated the interest of such a feature based on Ergun's law and got help from the former LS-DYNA developer M'Hamed Souli to demonstrate its feasibility with minimal changes to the software, and wrote a common proposal with Irvin Aerospace for LSTC (the developer firm) to implement it in a new version. The official release of version 971 makes the feature available for all users, and allows new developments in ENSICA for more accurate simulations, based on the cases defined in CEV.

## Nomenclature

*ALE* = Arbitrary Euler-Lagrange formulation
*FSI* = Fluid-Structure interaction
*DGA* = Délégation Générale pour l'Armement, common direction for CEV and ENSICA

---

[*] Numeric simulation specialist, ENSICA, Mechanical Engineering Department, 1 place Emile Blouin, 31056 TOULOUSE Cedex 5
[†] Aerospace Engineer, Centre d'Essais en Vol, base de Toulouse, BP 3061 - 18 rue Roquemaurel, 31024 Toulouse CEDEX 3, France, AIAA Member.
[‡] Numeric simulation specialist, Centre d'Essais en Vol, base de Toulouse, BP 3061 - 18 rue Roquemaurel, 31024 Toulouse CEDEX 3, France.
[§] Now at United Space Alliance – Systems Engineering & Integration – Aerodynamics – MSFC 4202/312

# I. Introduction

As the French ministry of defence procurement agency, DGA (Délégation Générale pour l'Armement) has an evaluation capability based on expertise and test centres, such as CEV (Centre d'Essais en Vol). It also controls national reference engineering schools such as ENSICA, that teaches civilian engineers dedicated aeronautics industry as well as military engineers dedicated to work in DGA. Mutual support exists between the engineering schools and evaluation centres, the first procuring backup for the application of emerging technologies in armament development and evaluation programs, the second procuring real life technical cases to be included in the students' education.

CEV has a base in Toulouse, formerly known as CAP (Centre Aérotransport et Parachutage) dedicated to the technical evaluation of aeromibility. CAP, the once national designer of French airdrop systems, started developing the numerical simulation of parachutes (SINPA) in the 90' as an aid to development[2-4]. CEV now continues developing a modelling capability as a support to evaluation and expertise, and as a sponsor for the technique to become usable by industry.

ENSICA, also based in Toulouse, has a long expertise of FEM explicit analysis (Finite Element Model formulations adapted to non-linear, dynamic cases) that developed a FSI capability in the early 2000' (Fluid Structure Interaction). In 2001, making a review of recent user applications as described in conferences proceedings, ENSICA found for the first time a publication[1] about parachute modelling with the LS-DYNA software, by a team that since established itself as a leader and reference in the effective use of the technique[5]. A co-operation was initiated with CEV to develop and promote the capability in France.

# II. Simulation architectures and models

## A. SINPA : the former implicit method

The first FEM FSI software used by CEV was based on the external coupling of two codes, N3S as a fluid solver and SAMCEF as a structural solver. The use of external coupling and implicit solvers made it hard to optimize the computing time and the result was not robust to large or rapid displacements or deformations. One strong constraint of this simulation method was the necessity for coincident nodes for the fluid and the structure mashes : whan the structure's deformation led to strong local evolutions of the fluid mesh size and topology, re-meshing became necessary. As a consequence, simulation cases like the a parachute opening or an arbitrary interaction of two parachutes would have required a lot of software development with no guaranty that the softwares used as a base would not reveal non-solvable limitations.

## B. The advantages and characteristics of the new explicit method

A FSI feature exists in LS-DYNA that allows to run models with non-coincident structure and fluid meshes. The fluid formulation is Arbitrary Euler – Lagrange (ALE), which means simulation time steps are Lagrangian (fluid nodes move) but between simulation steps, the fluid characteristics are re-computed at the initial node locations by a procedure called "advection". The result is thus equivalent to an Euler formulation.

To couple the fluid and the structure's behavior, the software identifies the host fluid element for each structure node and applies a coupling rule between the structure node and the fluid element's nodes : the Lagrangian formulation for the simulation time steps allow to manage it with generic contact features (either constraint methods, based on Lagrange multipliers, or penalty method, that allow a negligible relative displacement but apply a high proportional coupling force).
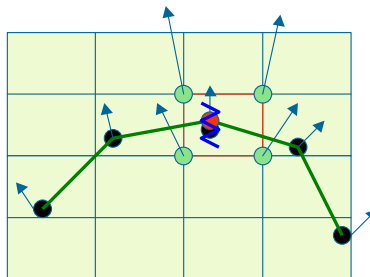


**Figure 1 : FSI coupling of a menbrane in ALE fluid.** The red membrane node's location is described in local fluid element coordinates, which allow to track its relative position and apply coupling forces.

The structure model is based on shell elements with a specific formulation for fabric and a MAT_FABRIC material formulation for orthotropic, elastic behavior that can be set to zero or low flexion stiffness and zero or low compression stiffness that allow the surface to wrinkle. Bands, lines and other 1D elements are based on BEAM, CABLE or SEATBELT formulations. The first is the most polivalent, allowing to control the pre-tension, flexion stiffness and orthotropicity of the elements but is not designed for such applications ; it can have a messy behavior when bands don't stay in-plane with the fabric or cause odd angles with the suspension lines. CABLE has no flexion stiffness, which makes it fine for lines, and a pre-tension or initial slack is easy to model. Yet it is highly unstable when slacked, which spoils its interest. SEATBELT is a very stable element formulation with no flexion stiffness, and probably the best formulation to use, yet its major drawback is the impossibility to model pre-tensions or initial slackness. Therefore, slack lines have to be modeled with an actually slacked geometry that requires an extensive use of Pythagore's formula.
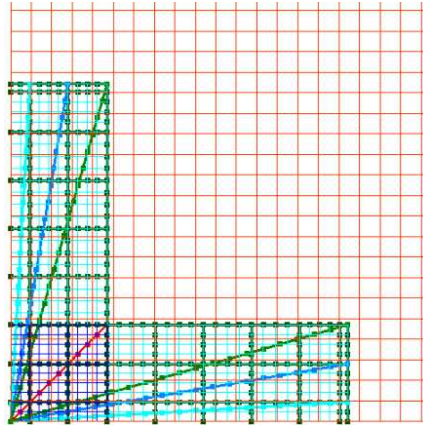


**Figure 2 : One-quarter model of a cross section parachute in a coarse fluid mesh.** The model being built in the flat constructed shape, the lines drawn with a straight geometry don't all have their actual unstrained length. A feature to setup pre-tension or slackness facilitates the modeling.
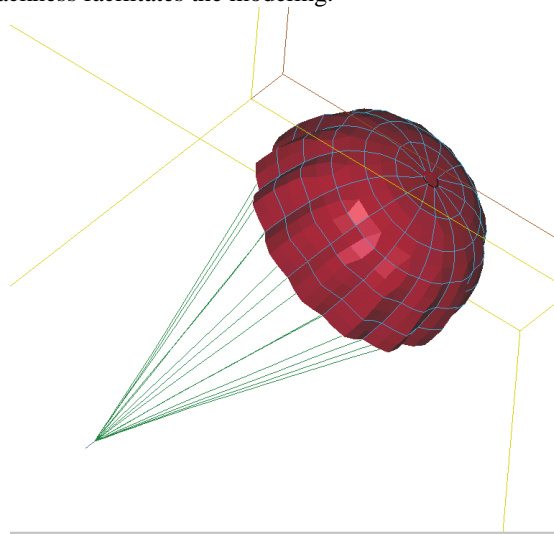


**Figure 3 : Full model of a main parachute extractor.**

## III.   Early simulation results

The initial purpose of the development team was to identify modelling options, evaluate their strength and weakness, and evaluate the necessary level of detail for the meshes and associated computation time. Due to the available computing power and mesh detail level chosen, the first model represented only a quarter of a hemispherical parachute with a limited fluid domain. This case was the base for ENSICA to check the feasibility and optimise parameters, and for CEV to learn how to build, run simulations and process the results.
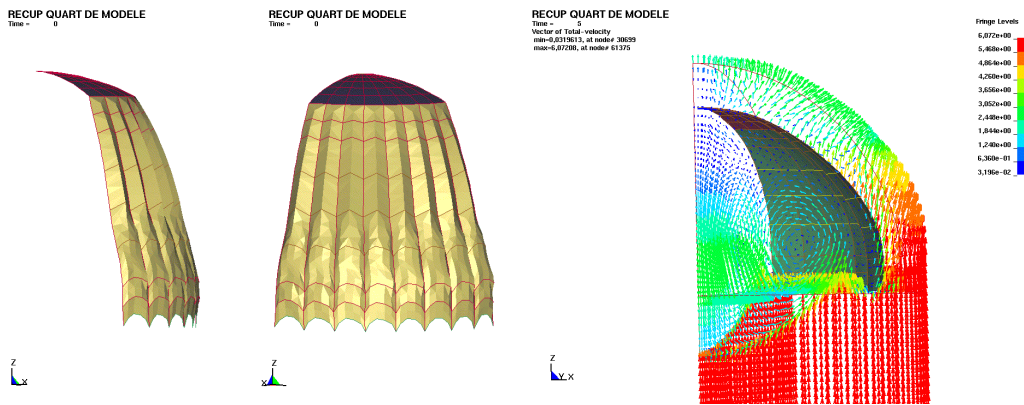
**Figure 4 : Early ¼ model for an hemispherical parachute and 3D map of fluid velocity.** Due to the fine mesh size chosen initially to ensure local precision and stability, the fluid domain modelled with 500 000 elements was yet not large enough to get correct boundary conditions. It allowed to estimate the allowable fluid element size and count for further simulations.

The first attempt at modelling a fluid domain large enough to get correct, unperturbed boundary conditions was then made on a quarter model for a cross parachute (Fig. 2). Based on the prototype simulation's result, a fluid element size was determined that would allow enough local precision for the fluid velocity field to be representative if not 100% accurate, and the fluid domain at least 3 times larger than the parachute's inflated diameter. The model counted then 80 000 elements, making it possible to run simulations in a few hours. By then, the boundary conditions and formulation for the fluid were based on the actual atmospheric pressure, a prescribed velocity for the flaw in applied on the fluid domain's boundary, and a prescribed pressure for the sides and outflow surface. This formulation soon appeared not to be adapted for the sides of the domain, where the flow diverged partly because of the boundary conditions used, and partly because of the insufficient size of the fluid domain. The side conditions were then replaced by a null horizontal velocity constraint. A new perturbation appeared then, because the new boundary conditions were reflecting pressure waves and causing resonance.
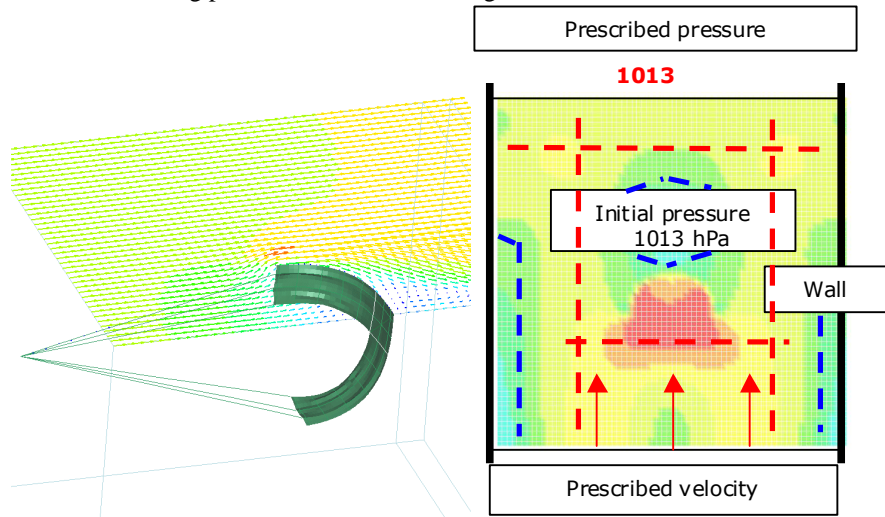


**Figure 4 : Early simulation results for the ¼ cross parachute.** Pressure waves are outlined red and blue in the pressure field representation.

Various attempts were made to get a larger fluid domain with a limited fluid element count, which requires a variable mesh size, or suppress the perturbation related to the boundary conditions. The use of variable size meshes with non parallelepiped elements caused additional perturbations that were probably caused by limitations in the advection method (which was confirmed eventually). A group of solutions were found in 2003, leading to a brand new formulation for the fluid.

American Institute of Aeronautics and Astronautics

## IV. Current simulation results

Advice was sought in 2003 during the LS-DYNA user's seminar about ALE and FSI. The pressure waves, boundary condition divergence and advection issues are in fact three classics in ALE simulations. A set of techniques now help getting better results :

- the fluid formulation is no more based on the physical atmospheric pressure value but an equation of state linearized around a null reference pressure. An additional bulk stiffness parameter allows to model the energy change in the fluid related to volume change .

$dE = -P.dV$ where $P \sim 1013$ mBar

is replaced by

$dE = -K.dV -PdV$ where $P \sim 0$, and $K = 1013$

$dE$ is a change in internal energy and $-P.dV$ is the work of pressure forces, $K$ is called "bulk stiffness"

- the boundary conditions at sides and outflow are now free (which is coherent with the new equation of state formulation with $P \sim 0$)
- an additional element layer below the inflow surface with constant pressure and density conditions avoid the feedback phenomenon between the pressure field disturbed by the parachute and the inflow pressure condition
- a new advection method is now available, that reduces the perturbations caused by non-parallelepiped elements. The advection trouble is also greatly reduced by the use of the new equation of state.
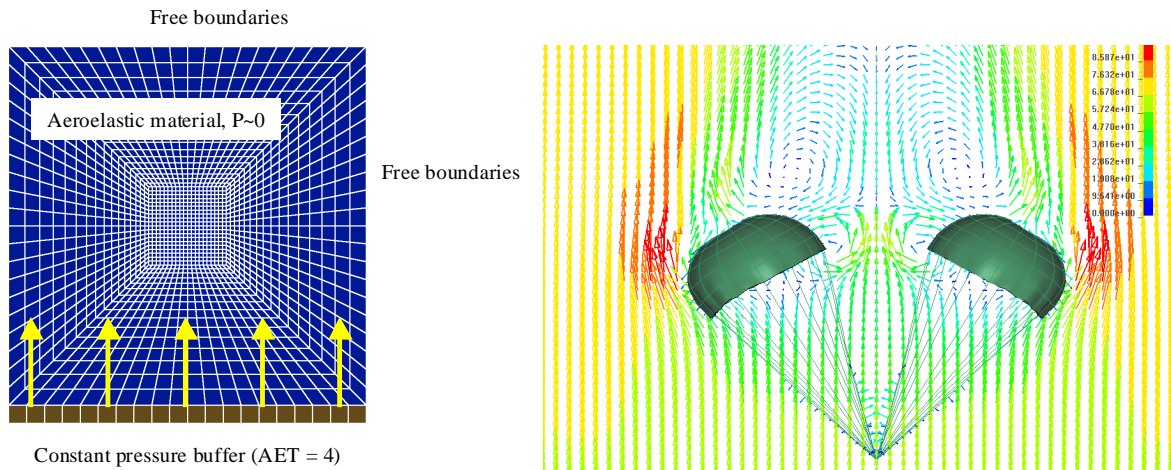


**Figure 5 : The current fluid formulation and variable size mesh. A velocity field with the new fluid formulation.**
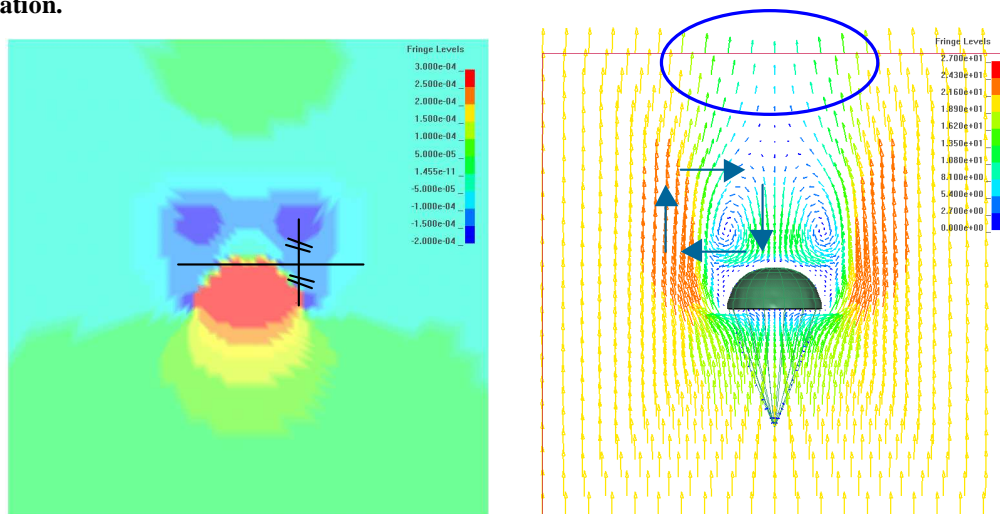


**Figure 6 : Pressure and velocity field with the new formulation.**

American Institute of Aeronautics and Astronautics

The new mesh for the fluid domain (Fig. 5, left) allows to make simulations with a 14 m wide fluid domain with 50,000 elements, with the same element size in the core area as the earlier 6 x 6 x 7 m fluid domain with 350,000 elements. Yet the hemispherical, non-porous parachute reveals that the fluid domain size is not yet big enough (the outflow is still perturbed on fig. 6, right) and the fluid element size is not small enough. As a result, the drag computed is 20% higher than the theoretical value based on drag coefficient Cd = 1,42 for half a sphere (based on the inflated diameter). This is the price to pay so far to get reasonable computing times, and be able to experiment new simulation parameters and features. Such simulations allow nevertheless to get results of good relative precision, such as the 2 parachutes grape simulation (Fig. 5, right), demanded by experts so as to evaluate the drag area of the pair. They expected it to be lower than the sum. The simulation indicated that for two parachutes with a drag area of 2 m² each, the pair had a drag area between 4.5 and 5 m². Subsequent flight tests demonstrated SCd = 5 m².
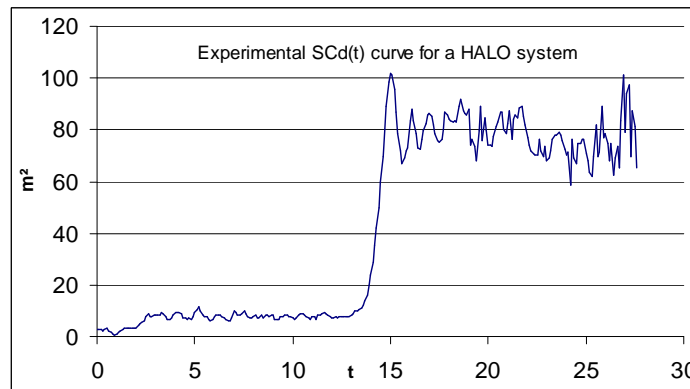
Experimental SCd(t) curve for a HALO system

**Figure 7 : Experimental Drag area vs. time curve for a High Altitude Low Opening system.** From t = 0 to t = 2 s, the drogue is not deployed yet. SCd ~ 2 m/s is the drag area of the load itself. From t = 2 s to t = 13 s, SCd ~ 7 m/s is the drag area of the load plus the twin hemispherical drogue. The main parachute opens from t = 13 s.

If the kind of imprecision related to the fluid mesh size and coarseness can be solved with time and additional computing power, LS-DYNA was so far limited to non-porous fabric models. The consequence is best illustrated with the cross section parachute simulations : the simulation is non porous, while the actual parachute is a highly porous extraction drogue.
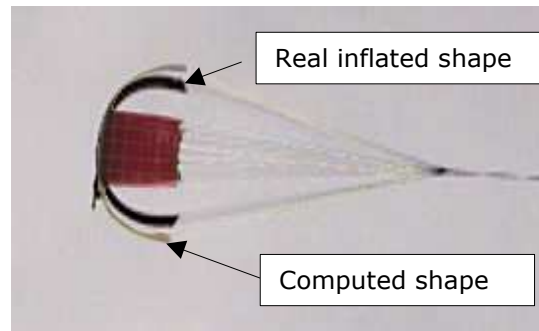
Real inflated shape

Computed shape

**Figure 8 : Inflated shape of an extraction parachute and the porosity law for the arms' fabric**.
The arms are twice as porous as the central part. In a simulation case with an inflow velocity of 67 m/s (130 Kts), the pressure offset between the inside and outside of the arms is up to 60 hPa, with non-porous fabric. The experimental lab data indicate that with 60 hPa, there shall be a leakage through the arms with an normal component of 5 m/s, which is definitely not negligible. The case was reported to LSTC in 2004.

## V.  Latest developments

Having a common interest with Irvin Aerospace for developments of the software's capability, DGA proposed to make a common demand for a new FSI feature in LS-DYNA. CEV was to provide the technical elements justifying the feasibility of the new algorithm base on Ergun's law (Fig. 8, left, red curve) and Irvin Aerospace is a major reference among LS-DYNA users, and has a good power of persuasion as such. The new feature only requires the

input of the viscous and aerodynamic coupling parameters of Ergun's law, and is therefore easy to plug in existing models. It is available since early 2007 in version 971.

## VI. Perspectives

The next steps for the DGA team will be new improvements in the fluid mesh to get accurate results in terms of drag, made possible by more computing power, the evaluation of the porous coupling with the well documented cross section extraction chute, and research for the formulation of free fall simulation cases or parachute opening with deformable and mobile fluid meshes.

## Acknowledgments

## References

[1] A. Taylor Irvin Aerospace Inc, Santa Ana, CA, "Developments in the application of LS-DYNA to fluid structure interaction (FSI) problems in recovery systems design and analysis," *7th International LS-DYNA Users Conferences*, 2001, Fluid/Structure pp. 10-17

[2] L. Chuzet (DGA, Centre Aeroporte, Toulouse, France), "Numerical determination of parachutes performances with SINPA software", AIAA-1997-1509, AIAA Aerodynamic Decelerator Systems Technology Conference, 14th, San Francisco, CA, June 3-5, 1997, Collection of Technical Papers (A97-31273 08-03)

[3] C. Ibos, C. Lacroix (SIMULOG, Labege, France), A. Goy, and P. Bordenave (DGA/CAP, Toulouse, France) , " Fluidstructure simulation of a 3D ram air parachute with SINPA software", AIAA-1999-1713, CEAS/AIAA Aerodynamic Decelerator Systems Technology Conference, 15th, Toulouse, France, June 8-11, 1999, Collection of Technical Papers (A99-30926 07-03)

[4] C. Ibos, C. Lacroix (SIMULOG, Labege, France), L. Chuzet (DGA, Centre Aeroporte, Toulouse, France), and D. Granville (SAMTECH, Liege, Belgium), " SINPA, a full 3D fluid-structure software package for parachute simulation", AIAA-1997-1508, AIAA Aerodynamic Decelerator Systems Technology Conference, 14th, San Francisco, CA, June 3-5, 1997, Collection of Technical Papers (A97-31273 08-03)

[5] B. Tutt and A. Taylor, Irvin Aerospace Inc, Santa Ana, CA; J. Berland, Para-Flite Inc, Pennsauken, NJ; B. Gargano, Quantum Parachutes, Boulder, CO, "The Use of LS-DYNA to Assess the Performance of Airborne Systems North America Candidate ATPS Main Parachutes", AIAA-2005-1609, 18th AIAA Aerodynamic Decelerator Systems Technology Conference and Seminar, Munich, Germany, May 23-26, 2005