Investigation of the Aeroelastic Coupling between a Nozzle and a Supersonic Jet

Eric SCHALL*, Bruno KOOBUS+ and Charbel FARHAT#
* INRIA, 2003 Route des Lucioles
06902 Sophia-Antipolis Cedex, France
+ Université Montpellier II, Place Eugène Bataillon
Departement de Mathématiques, CC 051
34095 Montpellier Cedex 5, France
Department of Aerospace Engineering Sciences
and Center for Aerospace Structures
University of Colorado, Campus Box 429
Boulder, CO 80309-0429, USA

Abstract

We consider the solution of a three-dimensional aeroelastic dynamic response problem involving a nozzle under the action of a supersonic flow. More specifically, we present the transient aeroelastic response obtained with a simplified model of nozzle when the flow condition induces shock waves either behind or inside the nozzle. We comment these results. In this problem, the aerodynamic forces acting on the structure are predicted by a non-linear flow theory. We use a three-field arbitrary Lagrangian-Eulerian method to formulate the coupled fluid/structure/moving grid problem, and employ a partitioned procedure for time-integrating the corresponding coupled semi-discrete equations. We consider a second order time- and space-accurate implicit scheme for solving the flow problem.

1 Introduction

Loads induced by supersonic flows on aerodynamical surfaces can lead to transient effects with sometimes flutter phenomena. A well known example is the flutter of a flat panel when a fast enough supersonic flow is applied along it [1]. The panel instability problem can be already simulated by 2D calculations relying on a classical Euler model for the fluid flow. Some papers also show this kind of behavior for axisymmetric nozzles under the action of an axisymmetric flow [2]. In that case, the excited mode is not one of the main modes of the nozzle structure. Indeed, for generic divergents the first main modes are not axisymmetric. The purpose of the present work is to consider non axisymmetrical coupling between an inviscid flow and a simplified model of nozzle. Different flow configurations involving shock waves behind or inside the nozzle are considered, and the possibly flutter phenomenon is investigated in each case. For investigating the instability nozzle problem, the software developped at the university of Boulder (Colorado) is used. The remainder of this paper is organized as follows.

In Section 2, we present the formulation of transient nonlinear aeroelastic problems. In Section 3, we summarize the numerical methodology for solving the nonlinear aeroelastic problem of the nozzle, and especially the space- and time-discretization, and the partitioned solution procedure. In Section 4, we present the instability nozzle problem. We describe the structure and flow problems, the structural eigen modes, and the transient aeroelastic simulations. Then we comment on the results obtained. Finally, in Section 5 we conclude this paper.

2 Formulation of transient nonlinear aeroelastic problems

The problem of the motion of the fluid/structure interface that occurs in coupled aeroelastic problems is addressed by solving the fluid equations on deformable dynamic meshes. An Arbitrary Lagrangian Eulerian (ALE) [3, 4] formulation is used in order to perform the integration of the fluid equations on a moving mesh. The coupled aeroelastic problem to be solved can then be viewed as a three-field problem [5, 4]: the fluid, the structure and the dynamic mesh which is represented by a pseudo-structural system. The semi-discrete equations governing the three-way coupled problem can be written as follows

$$\begin{aligned} \frac{\partial}{\partial t}(V(x,t)w(t)) + F^c(w(t),x,\dot{x}) &= R(w(t),x) \\ M\frac{\partial^2 q}{\partial t^2} + f^{int}(q) &= f^{ext}(w(t),x) \\ \tilde{M}\frac{\partial^2 x}{\partial t^2} + \tilde{D}\frac{\partial x}{\partial t} + \tilde{K}x &= K_c q \end{aligned} \tag{1}$$

where t designates time, x the position of a moving fluid grid point, w is the fluid state vector, V results from the finite element/volume discretization of the fluid equations, F^c is the vector of convective ALE fluxes, R is the vector of diffusive fluxes, q is the structural displacement vector, f^{int} denotes the vector of internal forces in the structure, f^{ext} the vector of external forces, M is the finite element mass matrix of the structure, $\tilde{M}$, $\tilde{D}$ and $\tilde{K}$ are fictitious mass, damping and stiffness matrices associated with the moving fluid grid and K_c is a transfer matrix that describes the action of the motion of the structural side of the fluid/structure interface on the fluid dynamic mesh.

3 Numerical methodology for solving coupled nonlinear aeroelastic problems

In this section, we give the main features of the numerical methods employed in this work for solving the coupled nonlinear aeroelastic problem given by Eqs. (1).
For more details about the numerical aspect, the reader is invited to read the works given in references.

3.1 Discretization of transient nonlinear aeroelastic problems

- **Spatial discretization**

  The spatial discretization of the fluid equations is based on a finite element/finite volume formulation on unstructured meshes. It combines a Roe upwind scheme for computing the convective fluxes, and a Galerkin centered method for evaluating the viscous terms. Second-order space accuracy is achieved through a piecewise linear interpolation method based on the MUSCL (Monotonic Upwind Scheme for Conservation Laws) procedure [6, 7], and a slope limitation algorithm [8] can be employed in order to damp or eliminate the spurious oscillations that may occur in the vicinity of discontinuities. For addressing the problem of flow simulations on moving grids, an ALE formulation is incorporated in the flow solver. The numerical algorithms used with this ALE formulation satisfy the Geometric Conservation Laws (GCL) [9, 10] that govern flow computations on moving grids.

  The structure is represented by a finite element model, and its dynamics behavior is predicted using the true displacement, velocity and acceleration degrees of freedom.

  At selective time-steps of an aeroelastic simulation, the dynamic fluid mesh is updated to conform the most recently computed configuration of the structure. The points lying on the fluid/structure boundary are first adjusted to conform to the new position of the surface of the structure, then the remainder of the fluid grid points are repositioned accordingly. In the methodology used in this work, the new position of the interior grid points is determined from the displacement solution of a discrete pseudo-structural problem representing the unstructured dynamic fluid mesh. The pseudo-structural system is constructed by lumping a fictitious mass at each vertex of the fluid mesh and attaching fictitious lineal springs to each edge connecting two vertices [12, 4, 5]. This discrete system is represented by the third of Eqs. (1) where $\tilde{M} = \tilde{D} = 0$.

  Finally, in fluid/structure interaction problems the fluid and structure meshes have often non-matching discrete interfaces. In that case, we use the load and motion transfer algorithms described in [13] for evaluating properly the pressure forces on the surface of the structure, and transferring correctly the structural motion to the fluid mesh. In particular, the loads induced by the fluid on the structure are computed in a conservative way.

- **Time discretization**

  For solving accurately and efficiently the flow equations given by the first of Eqs. (1) on dynamic meshes, a second-order time-accurate implicit algorithm preserving the GCL [11] is employed. The time discretization is based on a second-order backward difference scheme. The nonlinear flow equations derived from the time-discretization are solved by a defect-correction (Newton-like) method [14, 15].

  The structural equations of dynamic equilibrium given by the second of Eqs. (1) are solved with a second-order time-accurate implicit scheme where the trapezoidal method is used. The nonlinear algebraic system of equations is solved by a Newton-like method.

3.2 Staggered solution procedure

The solution of the coupled fluid/structure problem (1) is computed by a staggered solution procedure in the time domain [4, 16]. More precisely, we use the staggered algorithm given in [17] which satisfies the GCL as well as the continuity of both the displacement and velocity fields at the fluid/structure interface. This algorithm can be written as follows

1. Using the mesh position $x^{n-\frac{1}{2}}$, and the mesh velocity $\dot{x}^n$ that matches the structural velocity $\dot{q}^n$ on the fluid/structure interface , update the mesh coordinates as follows

$$x^{n+\frac{1}{2}} = x^{n-\frac{1}{2}} + \Delta t \dot{x}^n$$

2. Using $x^{n-\frac{1}{2}}$, $x^{n+\frac{1}{2}}$ and $\dot{x}^n$, update the fluid state vector $w^{n+\frac{1}{2}}$ in a manner that satisfies the GCL

3. Using the pressure computed from $w^{n+\frac{1}{2}}$, compute q^{n+1} and $\dot{q}^{n+1}$ using the midpoint rule.

4 Aeroelastic coupling between a nozzle and a supersonic jet

4.1 Description of the structure and flow problems

In this paper, we consider a simple structural model of a nozzle made with a cone-shaped part, a divergent, and a stiffener at the exit. The nozzle discretized in 9300 nodes is represented by an equivalent finite element model with 18400 triangular shell elements for the cone-shaped part and the divergent, and 100 beam elements for the stiffener (Fig. 1). The nozzle is clamped at its base, and immersed in the flow domain which is discretized into 87648 vertices and 483912 unstructured tetrahedra. A partial view of the fluid mesh is shown in Fig. 2.

The fluid mesh is decomposed into 14 subdomains and assigned to 14 processors for parallel computations. The finite element structural model is sequentially computed and therefore is not partitioned.

In this work, two different flow configurations are considered: depending on the inflow pressure, shock waves occur behind or inside the nozzle. For both configurations, the inflow density and Mach number are respectively set to $\rho_{inflow} = 0.423\ kg/m^3$ and $M_{inflow} = 1$, and the outflow condition corresponds to the ambiant air. The value of the inflow pressure is taken either to $P_{inflow} = 1060000\ Pa$ (shock wave behind the nozzle) or $P_{inflow} = 230000\ Pa$ (shock wave inside the nozzle).

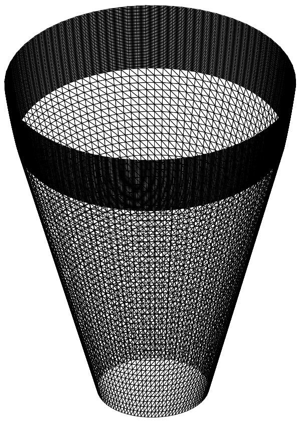


Figure 1: *Finite element model of the nozzle*

4.2 Transient aeroelastic simulations

The first fundamental modes of the structure are computed, and as expected, they are not axisymmetric. This tends to prove that a truly three-dimensional aeroelastic computation is necessary for simulating realistic nozzle coupled problems which are dominated by the first fundamental modes. The first structural mode corresponds to an ovalization mode whose

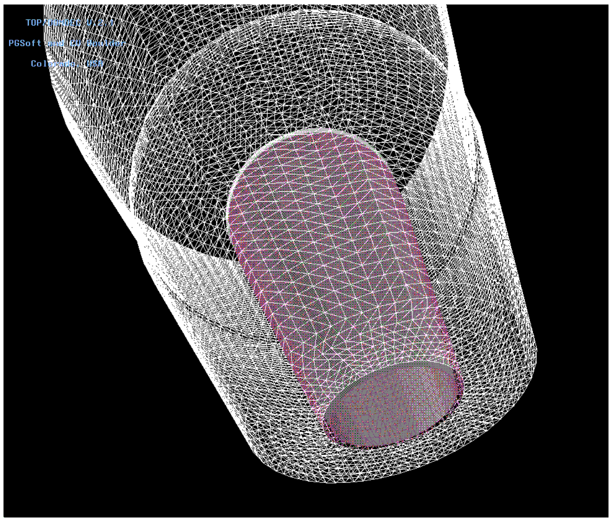


Figure 2: *Partial view of the fluid mesh of the nozzle and the immersed structure mesh.*

frequency is $freq = 25\ Hz$ (Fig. 3).

In a first step, the finite element structural model of the nozzle is perturbed along its first fundamental mode, and a steady state solution is computed around the deformed shape of the nozzle. A longitudinal plan cut of the obtained density, Mach and pressure fields is represented in Fig. 4 for an inflow pressure value $P_{inflow} = 1060000\ Pa$. A zoom of the mach isovalues around the nozzle is depicted in Fig. 5. One can notice a large fluid acceleration inside the nozzle and oblique shock waves attached to the exit. In the jet downstream the nozzle, there is a barrell-like structure, as well as a slip discontinuity at the boundary of this jet.

A longitudinal plan cut of the density, Mach and pressure fields obtained around the perturbed shape of the nozzle is represented in Fig. 6 for an inflow pressure value $P_{inflow} = 230000\ Pa$. A zoom of the mach isovalues inside the nozzle is depicted in Fig. 7. One can remark the presence of a normal shock wave preceding a subsonic zone inside the nozzle.

Next, this perturbation is used as an initial condition, and the aeroelastic response of the nozzle is computed by the partitioned procedure described in the previous section. The objective is to predict numerically the subsequent nozzle motion and flow evolution for both flow conditions.

For this dynamics problem, the time-step of the structure problem is set to $\Delta t_S = T_6/80 = 0.00025\ s.$, where T_6 denotes the period of the sixth mode of the nozzle. For the flow solver, the CFL number is set to 100. It turns out that for the given fluid mesh, this value of the

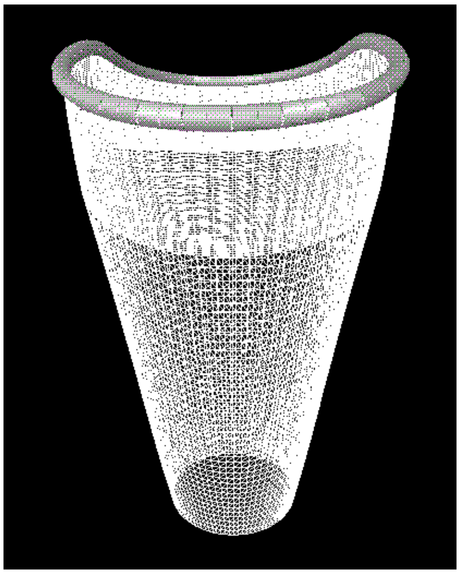
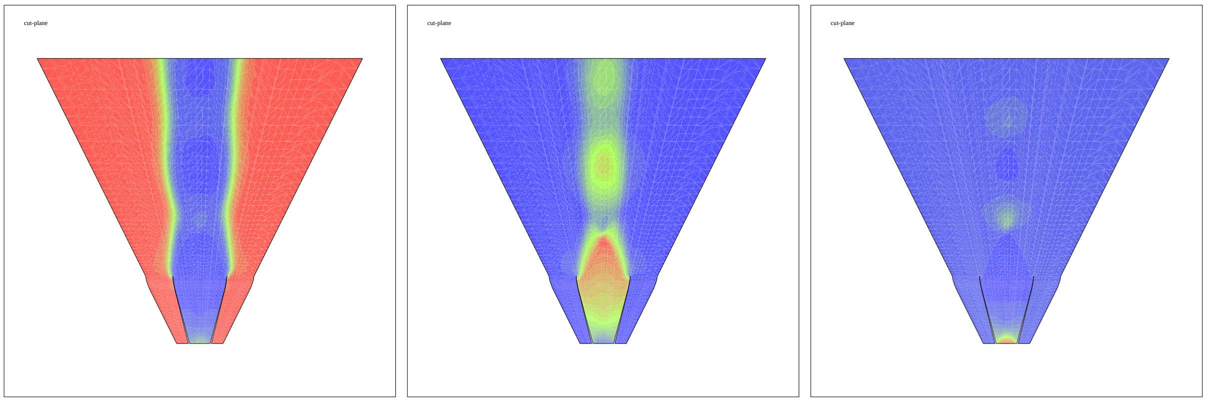


Figure 3: *First fundamental mode. The enlarged shape at the exit of the nozzle corresponds with the stiffener*



Density isovalues   Mach isovalues   Pressure isovalues

Figure 4: *Longitudinal plan cut of the solution for a perturbation along the first ovalization mode with an inflow pressure set to* 1060000 Pa

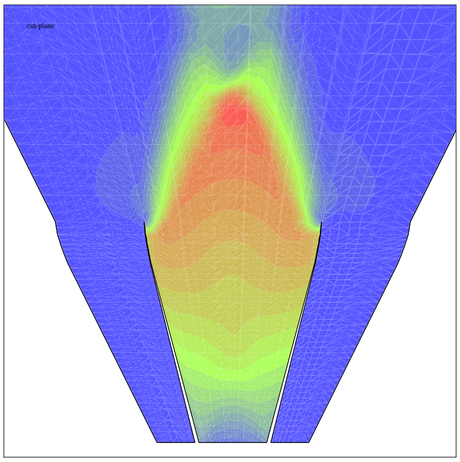
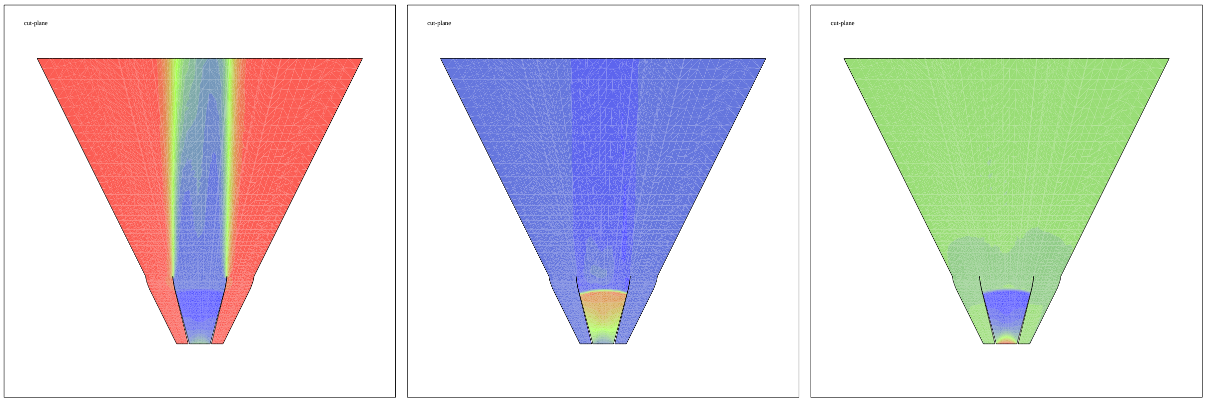


Figure 5: *Zoom of the Mach isovalues in the nozzle region for an inflow pressure set to* 1060000 Pa



Density isovalue Mach isovalues Pressure isovalues

Figure 6: *Longitudinal plan cut of the solution for a perturbation along the first ovalization mode with an inflow pressure set to* 230000 Pa

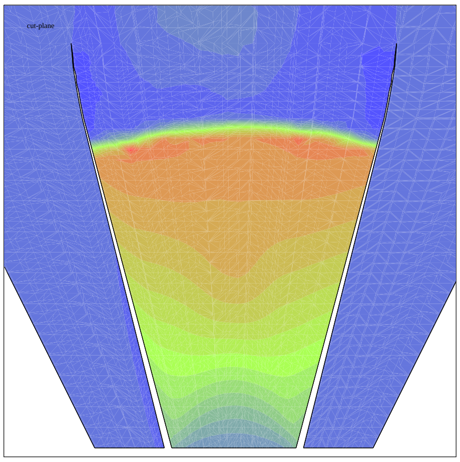


Figure 7: *Zoom of the Mach isovalues in the nozzle region for an inflow pressure set to* 230000 Pa

CFL number corresponds to an average value of the fluid time-step over the duration of the aeroelastic simulation $\Delta t_F = \Delta t_S$. The aeroelastic response obtained for both case of flow configurations is described hereafter.

- $P_{inflow} = 1060000\ Pa$
  With this flow condition, we have a large fluid acceleration inside the nozzle and oblique shock waves attached to the exit. The obtained radial displacement history of points located on the stiffener (point $p1$) and on the nozzle wall at 25% of the total length after the throat (point $p2$) is depicted in Fig. 8 for the first 0.2 physical seconds. We observe that the motion of the nozzle around its initial configuration is damped, and no flutter phenomenon accurs with this flow condition. The presence of small high frequency perturbations on the displacement history corresponding with the point located at 25% of the nozzle length is probably due to the fact that the nozzle is clamped at its base and its stiffness. Fig. 9 shows the evolution of the pressure at point $p1$, it consists in a damped oscillating average signal with small perturbations. The history of the longitudinal resulting force due to the pressure acting on the nozzle surface is depicted in Fig. 10. We remark that the averaged signal of this force is constant with time, and that the amplitude of the high frequency component of this signal is less than one percent of the averaged force.

- $P_{inflow} = 230000\ Pa$
  With this flow condition, we have a normal shock wave inside the nozzle. Because of the nozzle geometry and the simulation of an Euler flow, the shock wave induces no flow separation. The obtained displacement history of points located on the stiffener (point $p1$) and on the nozzle at 25% of the total length after the throat (point $p2$) is depicted in

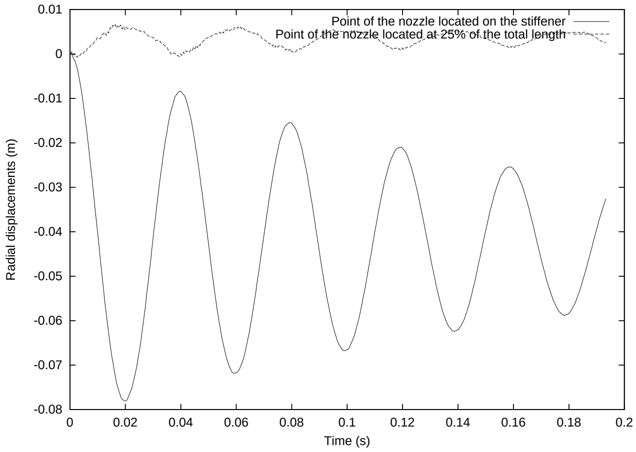


Figure 8: *Radial displacement history for an inflow pressure set to* 1060000 *Pa*

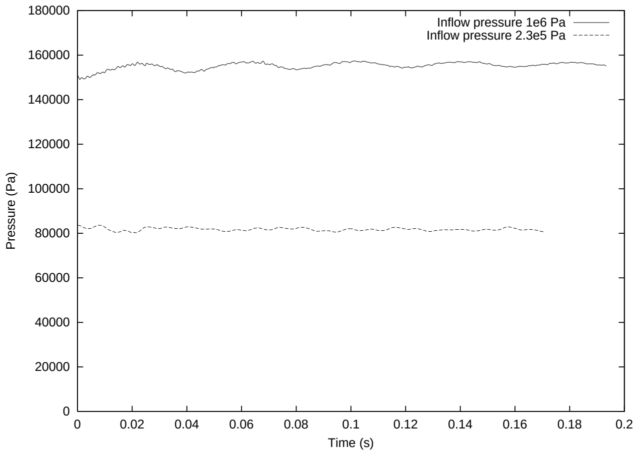


Figure 9: *Pressure history at point* $p1$

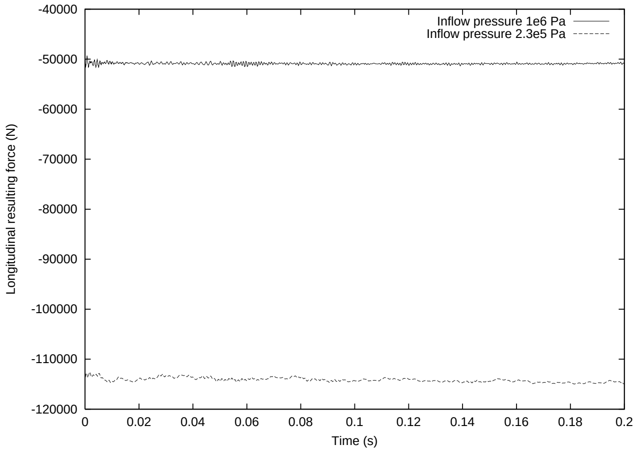


Figure 10: *Longitudinal resulting force history*

Fig. 11. As previously, the initial perturbation is damped, and no flutter phenomenon accurs. The pressure history at point $p1$ and the history of the longitudinal resulting force due to the pressure acting on the nozzle surface are respectively depicted in Fig. 9 and Fig. 10. The remarks made for the previous flow conditions are still valid.

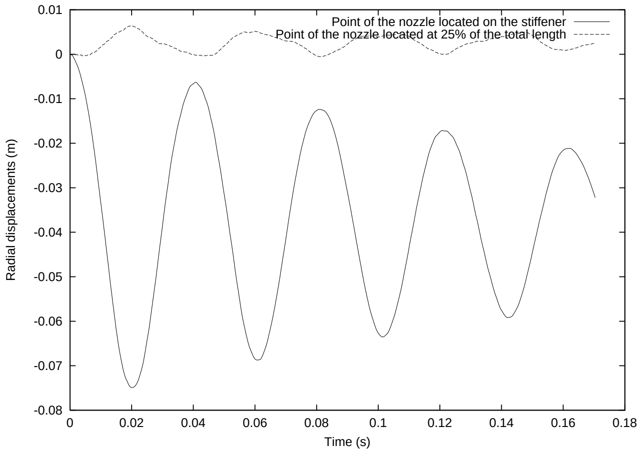


Figure 11: *Radial displacement history for an inflow pressure set to* 230000 Pa

Both aeroelastic problems have required around 800 coupled time-steps for the first 0.2 second of the transient aeroelastic response of the nozzle, and a total simulation time evaluated to 4 hours on a 15-processor SGI Origin 2000 machine.

5 Conclusion

In this paper, we have addressed the problem of simulating aeroelastic coupling between a nozzle and a supersonic jet. More specifically, no axisymmetric assumptions have been done, and we have considered truly three-dimensional jet/nozzle coupled problems with different flow conditions involving shocks either downstream or inside the nozzle. In this work, the flow has been assumed non viscous. We have shown that, for the investigated flow conditions, and with the given geometry and material features of the nozzle, an initial perturbation is damped with time and no flutter phenomenon accurs. The next step will be the investigation of this aeroelastic coupling problem when a shock wave involving a flow separation accurs inside the nozzle. In order to achieve such an aeroelastic simulation, turbulent flows will be considered.

Acknowledgements

The authors acknowledge the support and the technical informations by CNES (Centre National d'Etudes Spatiales, France) and SEP (Société Européenne de Propulsion). They would also like to thank Alain Dervieux at INRIA Sophia-Antipolis for his helpful discussions on the nozzle problem.

References

[1] R.L. Bisplinghoff and H. Ashley, "Principle of Aeroelasticity", *Dover Publications*, In., 1962.

[2] E. Lefrancois, G. Dhatt and D. Vandromme, "Fluid-Structural Interactions with Apllications to Rocket Engines", *Internat. J. Numer. Meths. Fluids*, in press.

[3] J. Donea, "An Arbitrary Lagrangian-Eulerian Finite Element Method for Transient Fluid-Structure Interactions", *Comput. Meths. Appl. Mech. Engrg.* 1982, 33:689-723.

[4] C. Farhat, M. Lesoinne and N. Maman, "Mixed Explicit/Implicit Time Integration of Coupled Aeroelastic Problems: Three-Field Formulation, Geometric Conservation and Distributed Solution", *Internat. J. Numer. Meths. Fluids* 21 (1995), 807-835.

[5] C. Farhat, "High Performance Simulation of Coupled Nonlinear Transient Aeroelastic Problems", *AGARD Report* R-807, Special Course on Parallel Computing in CFD (l'Aérodynamique numérique et le calcul en parallèle, North Atlantic Treaty Organization (NATO), October 1995.

[6] B. Van Leer, "Towards the Ultimate Conservative Difference Scheme V: a Second-Order Sequel to Godunov's Method", *J. Comp. Phys.* 1979, 32:361-370.

[7] A. Dervieux, “Steady Euler Simulations Using Unstructured Meshes”, *Von Karman Institute Lecture Series*, 1985.

[8] G. D. Van Albada, B. Van Leer and W. W. Roberts, “A Comparative Study of Computational Methods in Cosmic Gas Dynamics”, *Astron. Astrophys.* 1982, 108:76-84.

[9] M. Lesoinne and C. Farhat, “Geometric Conservation Laws for Aeroelastic Computations Using Unstructured Dynamic Meshes”, *AIAA Paper* 95-1709, 12th AIAA Computational Fluid Dynamics Conference, June 19-22, 1995, San Diega, CA.

[10] B. Nkonga and H. Guillard, “Godunov Type Methods on Non-Structured Meshes for Three-Dimensional Moving Boundary Problems”, *Comp. Methods Appl. Mech. Engrg.* 113 (1994) 183-204.

[11] B. Koobus and C. Farhat, “Second-Order Time-Accurate and Geometrically Conservative Implicit Schemes for Flow Computations on Unstructured Dynamic Meshes”, *Computer Methods in Applied Mechanics and Engineering*, Vol. 170, pp. 103-130 (1999).

[12] J.T. Batina, “Unsteady Euler Airfoil Solutions Using Unstructured Dynamic Meshes”,*AIAA paper* 89-0115, AIAA 27th Aerospace Sciences Meeting, Reno, Nevada, January 9-12, 1989.

[13] C. Farhat, M. Lesoinne and P. Le Tallec, “Load and Motion Transfer Algorithms for Fluid/Structure Interaction problems with Non-Matching Discrete Interfaces: Momentum and Energy Conservation, Optimal Discretization and Application to Aeroelasticity”, ...

[14] K. Bohmer, P. Hemker, and H. Stetter, “The Defect Correction Approach”, *Comput. Supp.* 1984, 5:1-32.

[15] R. Martin and H. Guillard, “A Second-Order Defect Correction Scheme for Unsteady Problems”, *Comput. and Fluids* 1996, 25:9-27.

[16] C. Farhat, M. Lesoinne, P. Stern and S. Lanteri, “High performance solution procedure of three-dimensional nonlinear aeroelastic problems via parallel partitioned algorithms: methodology and preliminary results”, *Advances in Engineering Software*,1997, 28:43-61.

[17] C. Farhat and M. Lesoinne, “On the accuracy, stability and performance of the solution of three-dimensional nonlinear transient aeroelastic problems by partitioned procedures”, *AIAA Paper* 96-1388, 37th AIAA/ASME/ASCE/AHS /ASC Structures, Structural Dynamics and Materials Conference, April 18-19, 1996, Salt Lake City, UT.