A Loosely Coupled Analysis of the Fluid-Structure Interactions for Inflatable Aerodynamic Decelerators

Matthew S. Bopp* and Stephen M. Ruffin†

Georgia Institute of Technology, Atlanta, GA, 30332-0150

The interaction between fluid and structural dynamics has become an important topic with regards to understanding the overall dynamics of inflatable aerodynamic decelerators. The present work aims to establish the capability to perform loosely coupled, fluid-structure interactions (FSI) through the use of the Cartesian Navier-Stokes solver, NASCART-GT and the finite element analysis (FEA) tool, LS-DYNA. Verification and validations are presented for the computational fluid dynamics (CFD) in order to demonstrate sufficient accuracy and applicability. These include CFD simulations of moving geometries, as well as a stationary analysis of a rigid tension cone. The FSI capability is demonstrated by examining the flow over a wedge with a deformable membrane, as well as flow over a semi-rigid tension cone configuration.

Nomenclature

\( c_p \) pressure coefficient
\( E \) modulus of elasticity, GPa
\( h \) material thickness, m
\( M \) Mach number
\( p \) pressure, Pa
\( t \) time, s
\( w \) wall velocity, m/s
\( x \) Cartesian coordinate, m
\( y \) Cartesian coordinate, m
\( z \) Cartesian coordinate, m
\( \alpha \) angle of attack, deg
\( \nu \) Poisson ratio
\( \rho \) density, kg/m\(^3\)
\( ()_\infty \) freestream property

I. Introduction

During the entry, descent, and landing (EDL) phase of a planetary exploration mission, the process of deceleration and selection of an aerodynamic decelerator is critical. Deployable systems such as parachutes and inflatable aerodynamic decelerators (IADs) can provide deceleration during the low supersonic/subsonic phase and potentially supersonic to hypersonic regimes respectively. Various deceleration approaches have been studied, tested, and flown over the past 50 years.1 Parachutes have been used for all U.S. planetary entry missions since the 1960s. However, there have been minimal improvements in deceleration technology over the last 40 years, since the development programs for the Viking missions. This technology relies on the 70\(^o\) sphere-cone, Viking aeroshell shape with the purpose of minimizing the ballistic coefficient. A deployable device, such as the Disk-Gap-Band (DGB) parachute has been effectively used over the years as a

---

*Graduate Research Assistant, School of Aerospace Engineering, 270 Ferst Drive, AIAA Student Member.
†Associate Professor, School of Aerospace Engineering, 270 Ferst Drive, AIAA Associate Fellow.
means of deceleration during planetary entry. However, due to parachute capability limitations, its range of applicability is becoming no longer valid for missions of interest. The successful landing of the Mars Science Laboratory (MSL) is an example of a mission for which the limits of the current parachute technology were pushed to the limits.\textsuperscript{2} This EDL sequence required the placement of the largest payload (900 kg) to date on the surface of Mars.

Future robotic missions, including rover explorations and sample returns, will continue to require increases in landed mass. With the intent to send humans to the surface of Mars, much larger (40-80 metric tons) payloads are projected.\textsuperscript{2} Such an endeavor will require drastic improvements to the current parachute technology or a significant study into alternative methods. One option that has been under development is the use of inflatable aerodynamic decelerators. As part of the development process for IADs, there are still significant advances in ground based tests, flight tests, and computational analyses that must be completed. The focus of these investigations requires detailed fluid dynamic and structural dynamic analyses, along with the coupling of these phenomena. The effects of aeroelastic coupling may prove to be a design-driving characteristic that must be accurately simulated and tested.

Over the past ten years, renewed efforts have been focused on the advancement of computational analyses, experimental testing, and flight-testing. Computational analysis has increasingly become more capable in simulating both the complex flow environments, as well as the ability to couple multiple disciplines to provide more accurate simulations. A review of the recent computational FSI analyses now follows, as it is relevant in understanding the current state of the art. Rohrschneider\textsuperscript{3} developed a multi-fidelity analysis tool to investigate the aeroelastic behavior in the continuum, transitional, and rarefied flow regimes. In the continuum regime, aerodynamics were calculated using the Cartesian solver, NASCAR-GT.\textsuperscript{15–19} Several structural analysis codes were investigated, with LS-DYNA\textsuperscript{4} producing the most consistent solutions for membrane-like materials. Static FSI analyses were performed on two geometries in the continuum regime. Lee\textsuperscript{5} augmented this work by including the effects of thermochemical nonequilibrium into the inviscid flow solution.

Clark\textsuperscript{6} established the capability of two CFD tools to compute steady-state flows over a tension cone. A series of wind tunnel tests was completed, providing a useful data set to validate computational analyses. The work also assessed the use of viscous versus inviscid solutions. Viscous analyses were required for configurations at high angles of attack, where turbulent boundary layers were likely to exist. Tamer\textsuperscript{7} presented the follow-on FSI work for these tension cone tests. The structural dynamics was computed using LS-DYNA, based on the previous studies by Rohrschneider\textsuperscript{3} and the CFD was computed using FUN3D, a Navier Stokes solver written at NASA Langley. This work provided a detailed set of LS-DYNA validation cases, which further proved its capability. Static FSI analyses were completed for a set of the previously conducted wind tunnel tests. Emphasis was made that the material modeling was an area for which continued research would be important. Gidzak\textsuperscript{8} demonstrated an in-step time coupling method between the unstructured, finite volume solver, US3D,\textsuperscript{9,10} and an explicit finite difference membrane solver. A DGB parachute configuration was analyzed for an MSL entry using a static parachute that was allowed to pitch and yaw in a two-dimensional plane (2D). A tension cone IAD was also investigated for a notional Mars entry, simulating the interaction between the fluid flow, structural dynamics, and heat transfer to the surface.

The analysis of supersonic DGB parachutes is presented by Karagiozis, et al.\textsuperscript{11} as a demonstration of a loose coupling method between a Large-Eddy Simulation and a structural membrane solver. Simulations were computed for capsule/DGB conditions corresponding to wind tunnel experiments performed by NASA.\textsuperscript{12} The configuration of interest was a DGB parachute attached to an aeroshell capsule. The flow was modeled using the compressible LES equations on a Cartesian mesh with adaptive mesh refinement. The structural dynamics was modeled using Kirchhoff-Love thin-shell theory, treating the structure as a membrane with non-zero bending stiffness. Additionally, research being conducted at Louisiana State University by Gilmanov, et al.\textsuperscript{13} presents a coupled FSI analysis tool using a particle-based meshless method for the structural analysis, along with a Cartesian based Navier-Stokes solver. This approach provides a framework to simulate flows over complex geometries that can undergo large deformations.

This paper presents a method which couples a Cartesian Navier-Stokes solver to a well-established and robust finite element tool. The results presented are for static analyses, though the extension to dynamic analyses does not require significant changes to the method. The paper begins with a discussion of the approach and methodology, followed by verification and validation of the CFD tool. Finally, FSI simulations are presented for the flow over a wedge and a tension cone configuration.
II. Approach and Methodology

The coupling of multiple disciplines within one solution scheme can take a variety of approaches, ranging from weak to strong coupling. Weak coupling generally refers to strategies where the solution to each physical phenomena is not advanced in time simultaneously. Instead, solutions proceed by alternating in time between two distinct solvers. On the other hand, strong coupling solves the set of governing equations for both phenomena as a complete system. In any dynamic analysis, the requirements for where the scheme falls on this spectrum depend on the nature of the problem and how strong the interactions are between phenomena. While the strong forms of coupling can provide very accurate results, the code development and solution time required can often be prohibitively expensive. On the other hand, weaker forms of coupling are far easier tools to develop and run times can be significantly faster. The current approach utilizes a loosely coupled framework, where the fluid dynamics and structural dynamics are each solved using independent codes. This provides a methodology for analyzing both static simulations as well as time accurate simulations. Time resolved solutions require small enough time steps to capture the dynamics of the system and a means of transferring data between solvers.

Figure 1: Flow chart demonstrating the FSI process of a loose coupling between the CFD and FEA tools.

Figure 1 depicts the flow process of a loosely coupled FSI analysis such as the one presented here. Communication between the CFD and FEA solvers is accomplished through the use of the software, Discrete Data Transfer Between Dissimilar Meshes (DDTBDM),\textsuperscript{14} which was developed at the NASA Langley Research Center. DDTBDM provides a means for transferring CFD pressure loads to nodal forces on an FEA mesh such that the total force and moments of the system are conserved. The computational meshes that are used for each of the analyses will in general, be different. Differences often exist as a result of mesh resolution requirements, especially since regions where large gradients in flow properties exist don’t necessarily correspond to regions of large stresses or strains. Additionally, the topology of the two meshes is often different. In the traditional finite volume and finite difference CFD methods, meshes are constructed around airtight geometries that have an identifiable interior and exterior. However, in the structural analysis of plates and membranes, there is no requirement for a geometry to form a closed body. The analysis of plates and membranes in a CFD tool requires a modification that adds an artificial thickness to these regions of the structure. This is exemplified in Fig. 2, where a comparison is made between the symmetry planes of a CFD surface mesh and an FEA surface mesh. The FEA centerline mesh consists of only the portions of the model that undergo deformation or rigid body motion. Thus, the aeroshell and sting are non-existent. The

Figure 2: Comparison between an FEA surface mesh and a CFD surface mesh, taken as a slice along the symmetry plane in order to highlight the topological differences. This is an example of a tension cone configuration.
CFD surface centerline mesh demonstrates the use of all components of the model (aeroshell, tension shell, torus, and sting). Note that a finite thickness has been added to the tension shell. It is this subtle difference in the meshes that requires special treatment with respect to the data transfer process.

The development of this FSI framework required a method for handling code execution and file management for data transfer. This has been accomplished by developing a Python wrapper to execute each of the tasks identified in Fig. 1. The user specifies several input parameters, one of which includes the number of FSI iterations to be performed. This corresponds to the number of complete cycles around the coupling flow chart. The static analyses performed in the current work proceed by first converging a CFD solution and then converging an FEA solution. FSI convergence can be determined when changes in both the flowfield and structural dynamics become negligible.

The structural analysis is computed using LS-DYNA, a nonlinear finite element solver developed by the Livermore Corporation. The primary advantage of using this software is the fact that it is extremely robust. As computational models increase in complexity, this method of analysis will be desired over the use of a simple plate/membrane solver. It will be important to have the ability to create models with varying element types as the structural analysis may require the interaction between fabrics, beams, and rigid bodies. The primary cost associated with using such a code will come down to the limitation that the solver cannot be accessed at the level of computer memory, potentially adding significant computational costs associated with it.

The fluid dynamics is computed using the solution adaptive, Cartesian, Navier-Stokes solver, NASCART-GT. The primary benefit of using this tool includes the ability to automatically generate the computational mesh over complex, three-dimensional (3D) geometries. This feature can save significant manual grid generation time, as well as provide the framework for which the intersections between the grid and geometry are updated as body motion occurs. In order to accurately compute flows near moving surfaces, the boundary conditions must take into account this motion. NASCART-GT uses the immersed boundary method to satisfy wall boundary conditions by directly imposing the state vector in a cell interior to the body. A detailed description of the implementation of the immersed boundary method can be found in the Ph.D thesis by Sekhar. For inviscid wall conditions, stationary bodies require the normal component of velocity and the tangential velocity gradient to both vanish at the wall. The extension to moving bodies simply requires the normal velocity at the wall to be equal to the wall normal velocity of the moving body. For the slip wall condition, the wall tangential component of motion has no effect on the boundary condition.

For moving body cases, the Cartesian mesh remains stationary while the geometry passes through it. At each time step, the new location of every surface element is shifted to its new location and the cell intersections are re-computed. The current methodology uses explicit time integration, which puts a limit on how large these body movements can be in one time step. This restriction is quantified by the CFL number based on wall velocity, \( \text{CFL}_w = \frac{\Delta t}{\Delta x} \), which must remain smaller than unity. An extension to implicit, Cartesian, moving body simulations has been presented by Murman, et al. An example Cartesian mesh is depicted Fig. 3, illustrating the initial grid around a cylinder. The entire surface cells have been shown in order to emphasize the intersections between the cylinder and mesh. The inset draws attention to a location near the surface of the body where Fig. 4 further demonstrates body movement through the fixed Cartesian domain. Moving from left to right across Fig. 4, there are three instants in time showing how the grid intersects with the cylinder at each step. The shaded cells are surface cells, defined as cells which contain body intersections. Note that there are no physical cells inside the body, and as surface cells become completely enclosed by the body, they are removed. Cells also emerge from inside the body, which can present difficulties since no flow properties were associated with cells previously inside the body. The
current implementation uses neighboring cell data to set the state vector in the newly emerged cell.

Figure 4: Close-up views of the surface mesh intersecting with the geometry. From left to right, the progression in time yields body movement to the left through the stationary grid. Shaded cells are marked off to indicate surface cells.

III. Results and Analysis

The analysis of fluid-structure interactions requires CFD and FEA solvers capable of accurately simulating each discipline independently. Although NASCART-GT has the capability of computing viscous, chemically reacting flows, the current work is restricted to inviscid analyses as a means for establishing the FSI implementation. Validation cases are presented in order to demonstrate the applicability of the solver to the current class of problems and to demonstrate the moving boundary implementation. Validations of LS-DYNA have been presented by Rohrschnieder\(^3\) and Tanner\(^7\) for the specific types of cases that are presented in this paper. Thus, similar validations of LS-DYNA's accuracy are not explored in this current document. In the present study, 3D flow over a rigid tension cone configuration is first presented, followed by three validation cases involving moving bodies. Finally, two FSI cases are presented that examine the static analyses of flow over a deformable membrane on a wedge and flow over a semi-rigid tension cone configuration.

III.A. Rigid Tension Cone

The first case presents an inviscid analysis of supersonic flow over a rigid tension cone shape. This serves as a validation of the CFD solver’s 3D steady state inviscid capability. Experimental schlieren imagery and pressure coefficient data\(^6\) are used as a means to validate the computational solution. With a freestream Mach number of 2.5 and an angle of attack of 20°, this case serves as a useful benchmark to examine a 3D flowfield using the current CFD tool. The freestream density and pressure are 1.082 x 10\(^{-1}\) kg/m\(^3\) and 4,481.59 Pa, respectively. The simulation was run using a 2\(^{nd}\) order accurate, AUSMPW+ inviscid flux scheme and a 1\(^{st}\) order accurate implicit time integration scheme. The computational mesh consisted of approximately 3.3 million cells with the smallest cell size on the order of 6 x 10\(^{-4}\) m. Grid adaption based on velocity divergence was utilized in order to resolve gradients in the flow, especially along the bow shock.

Figure 5 shows the distribution in pressure coefficient along the symmetry plane. The computational solution compares very well with the experimental data along the windward portion of the surface with an under-prediction occurring beyond the nose of the aeroshell and a slight over-prediction as the leeward shoulder is approached. Figure 6 shows comparisons between the computed density field and schlieren imagery. These comparisons show close agreement with respect to the shock structure in the forebody region. Note that the accuracy of the solution in the wake region is hindered by the inviscid analysis, given that this...
region is dominated by viscous effects.

![Bottom half of CFD solution.](image1)
![Top half of CFD solution.](image2)

Figure 6: Comparison between the computed density field and the experimental schlieren images for the rigid tension cone at 20° angle of attack.

### III.B. Constant Velocity Piston

Given the non-stationary nature of the surface interactions in FSI problems, it is necessary to analyze situations with moving boundaries. The first analysis presents the one dimensional, inviscid solution of a piston moving at a constant supersonic velocity inside a channel. An analytic solution to this problem exists, in which the flow properties are known as functions of space and time. The solution to such a problem is presented in texts such as John or Anderson. This provides a straightforward case for validating the boundary conditions associated with moving bodies. The problem has been setup on a two dimensional grid with a rectangular box (piston) moving in the positive x-direction. The channel is simulated by applying symmetry conditions at constant y locations.

For the current analysis, the piston is provided with a constant velocity of 1000 m/s into stationary air at 4 x 10^5 Pa and with a density of 1 kg/m^3. This corresponds to a Mach number of 1.34. The solution is advanced in time using the explicit 2nd order accurate Hancock predictor-corrector scheme and 1st order accuracy is used in space. A constant time step of 5.0 x 10^{-6} s was used, which resulted in the piston moving 50% of one cell size per iteration. The density field is depicted in Figure 7, where the piston has traveled 10 m. Note that the actual piston used in the simulation had a finite thickness associated with it, while the plots have been presented such that the piston is infinitely thin. The computational results are compared to the analytic solution in Fig. 8 at a flow time of 0.01 s. Figure 8a shows excellent agreement between the two solutions. There is a discontinuity in density that is created across the piston as the gas is expanded behind it and compressed in front of it. Similarly, in Fig. 8b the velocity distribution exhibits both the compression and expansion, as well as its uniformity immediately on either side of the piston.

### III.C. Accelerating Piston

A second study involving the 1-D motion of a piston is that in which the piston accelerates from rest. This simulation is different from the previous in that isentropic compression is studied since the exact solution is only valid up to shock formation. The constant acceleration of the piston is 100 m/s^2 and the fluid into which the piston accelerates is air at standard sea level conditions. The same numerical schemes are utilized as in the previous case at constant velocity. The time step was 2 x 10^{-4} s, resulting in a maximum piston movement of 2% of the cell size. Figure 9 shows a velocity comparison between the computational results and the analytic solution as a function of the distance away from the piston and the flow time. The results compare very well for all spatial and temporal coordinates.

### III.D. Diamond Airfoil

The previous 1D analyses are now extended to verify 2D flow. A diamond airfoil at 0° angle of attack is investigated since the pressure distribution along the surface can be computed analytically. The flow over
Figure 7: A piston travels in the positive x-direction at a velocity of 1000 m/s. The density field is shown at a flow time of 0.01 s.

(a) Distribution of density along the channel length.  
(b) Distribution of velocity along the channel length.

Figure 8: Comparison of the computational flow properties with the analytic solution for a piston moving at constant velocity into stationary air.

Figure 9: Velocity distribution ahead of and behind accelerating piston.
a stationary airfoil is presented in addition to the translating case as a means of directly comparing the stationary boundary conditions to the non-stationary conditions. The relative velocity between the airfoil and the surrounding freestream are the same for both cases and the smallest cell sizes are also the same. This essentially serves as a change in reference frame. An airfoil with a chord length of 1 m and a half angle of 21.8° has been chosen. The fluid is air at standard sea level conditions and a relative Mach number of 2. In order to continuously capture the moving gradients in the flow, solution-based grid adaption is employed at every time step.

Figure 10 contains a comparison between a stationary CFD analysis and a moving CFD analysis. The pressure distribution along the moving airfoil in Fig. 10a matches fairly well, given that the largest deviation is under 4% occurring behind the oblique shock on the forward face. The pressure flowfield is compared between the stationary and moving cases in Fig. 10b. The stationary airfoil is shown on the upper half and the moving airfoil is shown on the lower half. The contour lines emphasize some of the differences between the two. Primarily, there are slight differences in the shock front shape and thickness. The shock front in the moving body case is smeared over a larger distance. The shock is also weaker in that the pressure rise is smaller in the moving body case and the effects of the body do not propagate as far as evidenced by the contour lines. These differences are likely due to the spatial resolution and order of accuracy. Additional analyses are required to characterize this further.

III.E. FSI: W edge Flow

The following case is posed as a demonstration of the FSI capability. A 3D, inviscid case is generated which simulates flow over a 20° wedge, as depicted in Fig. 11. The darkened square region on the upper surface represents a deformable membrane, whose deflection is driven by the effects of the fluid. Flow is computed over the upper half of the wedge, treating the lower half as a symmetry plane. Additionally, symmetry planes in the z-direction have been enforced, in order to simulate a 2D flowfield. A list of the input parameters is presented in Table 1. The flow solution over the wedge, prior to membrane deflection, provides a uniform pressure loading to the entire upper surface. The surface pressure contained over the membrane portion of the wedge is converted to a nodal force distribution that is applied to the structural analysis. Within LS-DYNA, the membrane is treated as a set of fully integrated shell elements that are linearly elastic and isotropic with all four sides set to fixed boundary conditions. The mesh consists of 40 x 40 elements.

Four FSI iterations were completed to achieve a steady state convergence of the membrane’s deflection. Figure 12 contains a flow solution over the deformed geometry as well as a plot of the pressure coefficient along the surface. The velocity flowfield shown in Fig. 12a shows the primary attached shock wave caused...
by the wedge. Within the shock layer, additional flow features arise due to the deflection of the membrane, which can also be tracked by following the distribution in pressure coefficient in Fig. 12b. The flow initially expands as it moves across the membrane and begins to compress as it flows up the aft portion. This is then followed by another expansion from the edge of the membrane back onto the wedge. The flow then adjusts to a constant pressure downstream.

![Figure 11: Schematic of 20° wedge with square membrane indicated in black on the upper surface.](image)

Table 1: Flow conditions and material properties for the wedge case and tension cone case.

<table>
<thead>
<tr>
<th>Freestream Conditions</th>
<th>Material Properties</th>
</tr>
</thead>
<tbody>
<tr>
<td>$M_\infty$ 2.441</td>
<td>$E$ 2.6593 GPa</td>
</tr>
<tr>
<td>$\rho_\infty$ 2.6776 x 10^{-2} kg/m^3</td>
<td>$\rho$ 1125.86 kg/m^3</td>
</tr>
<tr>
<td>$p_\infty$ 1034 Pa</td>
<td>$h$ 3.3401 x 10^{-4} m</td>
</tr>
<tr>
<td>$\alpha$ 0°</td>
<td>$\nu$ 0.3</td>
</tr>
</tbody>
</table>

![Figure 12: Supersonic flow over a wedge containing a deformable membrane on the upper surface.](image)

(a) Flow velocity along the centerline of the wedge.  
(b) Pressure coefficient along the centerline of the wedge.

III.F. FSI: Semi-Rigid Tension Cone

The final simulations analyze the static deformation of a semi-rigid tension cone based on a series of wind tunnel tests at the NASA Glenn 10- by 10-ft Supersonic Wind Tunnel. Figure 13a shows the CFD model and the LS-DYNA part model is shown in Fig 13b. Detailed descriptions of the model can be found in the paper by Clark, et al. and the Ph.D thesis by Tanner. The freestream conditions and material properties...
are identical to the wedge case, listed in Table 1. The analysis was performed with symmetry conditions as can be seen by the half models shown in the figures. The tension cone can be identified by its two main components: the tension shell and the torus. The tension shell textile is made of a urethane-coated Kevlar of thickness, \( h \), where the tape seams are modeled by doubling the thickness. The rigid torus is modeled using a rigid material with associated inertia properties. A relatively coarse mesh has been used, consisting of 6,432 elements.

Simulations are presented for flows at angles of attack of \( 0^\circ \) and \( 9^\circ \). The converged solutions are shown in Figs. 14 and 15. Figure 14a shows the density flowfield along the symmetry plane and Fig. 14b shows the Mach number and volume mesh for the \( 0^\circ \) angle of attack case. The same two plots are presented for the \( 9^\circ \) angle of attack simulation in Figs. 15a and 15b respectively. The converged solutions went through 4 FSI iterations. Figure 16 presents a comparison for the \( 0^\circ \) angle of attack case between the deformed tension cone shape along the centerline with the original undeformed state and the results from the wind tunnel test. The deflected shape shows comparable agreement with the wind tunnel data. This case demonstrates the capability of the current methodology to solve loosely coupled, static FSI problems for the tension cone configuration. Further, it lays the groundwork for extending this capability to the analysis of time-accurate simulations.

![Figure 13: Computational models for semi-rigid tension cone.](image-url)
Figure 14: Tension cone converged solution for 0° angle of attack.

Figure 15: Tension cone converged solution for 9° angle of attack.
IV. Conclusion

A loosely coupled approach to analyzing fluid-structure interactions is presented. The fluid dynamics is computed using the Cartesian, Navier-Stokes solver, NASCART-GT, and the structural dynamics is computed using LS-DYNA. Data communication between the dissimilar meshes is accomplished through the use of a discrete data transfer tool called DDTBDM. CFD validation cases are presented for stationary and moving body simulations. A 3D inviscid solution is computed over a rigid tension cone in order to demonstrate applicability of the solver for the current class of problems. Additionally, the boundary conditions associated with non-stationary geometries are verified by investigating the motion of a piston, as well as 2D flow over a diamond airfoil. FSI solutions are presented for flow over a partially deformable wedge, as well as a tension cone configuration. This initial study lays the groundwork for further analyses of static FSI problems and paves the way for extensions to examine time-accurate FSI simulations.

Acknowledgments

This work was supported by a NASA Office of the Chief Technologist’s Space Technology Research Fellowship. The authors would like to acknowledge Christopher Tanner of the Jet Propulsion Laboratory for his assistance in working through some of the details regarding the semi-rigid rigid tension cone model and the data transfer process.

References

of Aerospace Engineering, Georgia Institute of Technology, 2009.


