COMPRESSIBLE FLUID-STRUCTURE INTERACTION SIMULATIONS OF AN INFLATABLE TENSION CONE DECELERATOR FOR PLANETARY RE-ENTRY

Carlos Pantano[*1], Richard Kramer[*] & Fehmi Cirak[**]
[*]*Department of Mechanics Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, USA*
[**]*Department of Engineering, University of Cambridge, Cambridge, UK*

Summary Inflatable aerodynamic decelerators present potential advantages for planetary re-entry in missions of robotic and human exploration. The design of these flexible structures faces many engineering challenges, including the complex dynamics of ultra-thin flexible structures, anisotropic material response, and coupled shock-turbulence interactions. We describe a comprehensive computational fluid-structure interaction study of a tension cone decelerator in supersonic flow at Mach 2.5 and compare the simulations with earlier published experiments conducted at NASA Langley. The simulations utilize adaptive mesh refinement, large-eddy simulation, and large deformation thin-shell mechanics with self-contact to represent the flow and structure interaction. This study focuses on the compressible fluid mechanics, the dynamics of the structure as the free-stream conditions vary gradually, the two-way interaction between the fluid and the structure, and the incipient formation of aeroelastic and structural instability.

INTRODUCTION

Theoretical point-designs suggest that an inflatable aerodynamic decelerator (IAD) is a promising means for deceleration of a heavy payload into a planetary atmosphere. It offers a low mass to drag-producing area ratio for a reduced ballistic coefficient and large deceleration rates at high planetary altitudes, where atmospheric density is low. This enables higher elevation landing sites along with reduced vibrations and large-scale structural oscillation during the entry sequence. Preliminary experimental results from the 1960's along with recent experiments [1] indicate that IADs may offer advantages over supersonic parachutes for high-payload deceleration, avoiding well-known instabilities encountered in these parachutes within the Mach number range of interest. To date, only reduced-scale IADs have been tested in the laboratory or in test campaigns in Earth's atmosphere. Given the large number of material, structural, and flow parameters involved, there is substantial uncertainty regarding the process that should be followed to rescale the experimental drag and moments to full-scale geometries. In this paper, we expand on our recent results presented earlier [2] using a structured computational fluid dynamics (CFD) solver with adaptive mesh refinement (AMR) that is coupled with a large-deformation thin-shell finite-element (FEM) solver to investigate the stability and aerodynamic behavior of a tension cone inflatable aerodynamic decelerator in supersonic flow. The main progress discussed here is the use of an accurate structural model that closely approximates the flexible experimental model used in [1], which includes seams between the segments of the 16-sided polygonal tension cone and torus, and a detailed investigation of the origin of a new type of aeroelastic instability that is observed in the simulations.

COMPUTATIONAL MODEL AND APPROACH

A three-dimensional large-eddy simulation (LES) computational fluid model employing a conservative finite-difference method with low numerical dissipation is used to simulate the flow. The membrane structure is modeled as a thin shell and is discretized with subdivision finite elements [3]. A Kirchhoff-Love type finite-deformation kinematics is assumed, which allows for arbitrarily large structural displacements and rotations. The corresponding internal energy functional is discretized with smooth subdivision shape functions, thus resulting in a conforming finite element discretization of the shell. The discretized Hamilton's principle of stationary action is used to determine the equilibrium configurations of the shell with inertia and possible self-contact. The spatially discrete action integral is integrated in time with a Newmark method in predictor-corrector form. Moreover, the time integration loop is augmented with a contact algorithm to prevent the interpenetration of the structure with itself, which occurs during the large-scale buckling of the entire IAD. The fluid and solid models are loosely coupled in a thin region near the interface boundary using a temporal splitting method. This interaction is modeled using the ghost-fluid method (GFM), which imposes the fluid boundary condition on ghost cells using level sets in a Cartesian scheme. Slip boundary conditions are employed at the structural surface, which is expected to be a fair approximation in these massively separated flows where the transmural pressure is the main contributor to the structural deformation. The computational environment is based on an object-oriented AMR framework. The solver suite is part of the Virtual Test Facility (VTF), originally developed at the California Institute of Technology for the Advanced

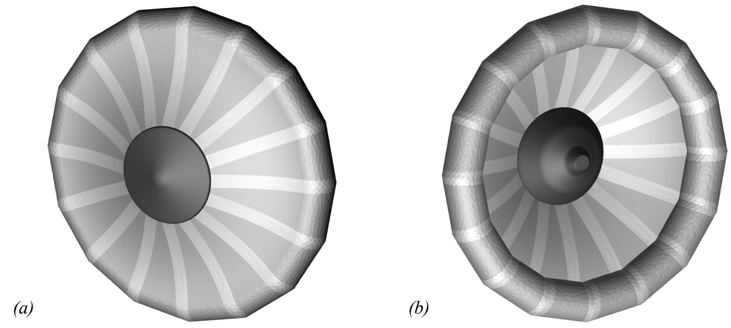
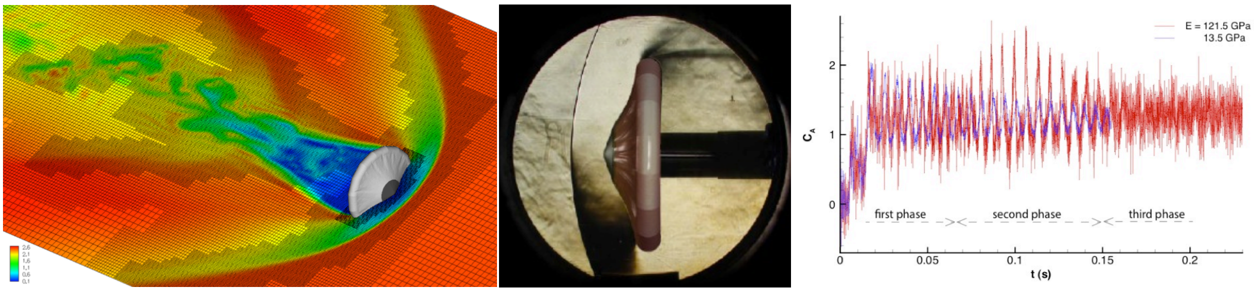


Figure 1: Front and back views of the IAD structural model based on the experimental geometry (16-sided polygon). Light regions highlight the seams (modeled as twice the thickness of the fabric).

[1] Corresponding author. Email: cpantano@illinois.edu

Simulation and Computing (ASC) program of the US Department of Energy [4]. Figure 1 shows the 16-sided polygonal IAD model investigated here, which has an outer diameter of 0.6 m across the flat sides of the polygon. The decelerator is characterized by a toroidal outer segment, inflated to a controlled pressure, which supports a thin conical membrane that is attached to the shoulder of a Viking-like capsule. The diameters of the torus cross-section and the largest diameter of the tension cone membrane are 7.5 cm and 52.5 cm, respectively. These dimensions match the nominal design of the inflatable experimental scale model investigated by Clark et al. [1]. In the present structural model, the connection between the tension cone and inflatable torus is assumed to be non-clamped, such that the common edges of the finite-element triangles that discretize both surfaces remain connected as the structure deforms while they are free to rotate relative to each other. The cone is fixed to the rim of the rigid capsule, which is prescribed to be fixed in space. The membrane fabric has a thickness of 0.254 mm, density of 1440 kg/m^3, and is modeled as an isotropic linear elastic material with a Young's modulus of 13.5 GPa and zero Poisson's ratio. The seams at the overlaps of the segments forming the polygon have a width of 2.54 cm and twice the thickness of the fabric.



(a) (b) (c)

Figure 2: Slice through the center of the computational domain showing regions of mesh refinement, superimposed on contours of Mach number for an inflation pressure of 300 kPa (a), side view overlay of experimental and computational structural deformation (b), and drag evolution with time under normal and unstable fluid-structure interaction (c).

SUMMARY OF SIMULATION RESULTS

The flow conditions correspond to a freestream Mach number of 2.5, with a stagnation pressure of 28.632 kPa and free stream velocity of 311.358 m/s for a Reynolds number of 1.6×10^6, in accordance with the experiment [1]. The computational flow domain size is 8x2x2 meters and it is discretized dynamically using AMR with three levels of refinement at a refinement factor of 2 between levels. The total fluid mesh size is approximately 4×10^7 cells and varies slightly with time as refinement evolves to accommodate flow features and the deformation of the structure. Refinement is primarily localized near the body and the bow shock; a snapshot of a slice through the center of the domain superimposed onto contours of Mach number with the AMR grid density is shown in Figure 2(a). The simulations are initialized without accounting for any (locked-in) pre-stress in the membranes due to fabrication details of the real device. These simulations use the polygonal cone and torus geometry described earlier that is discretized with a mesh involving 81,280 triangular finite elements. The fully developed flow ahead of the tension cone structure exhibits a bow shock, a turbulent wake behind, and a recompression shock downstream; as shown in Figure 2 (a). These features persist at all torus inflation pressures, though their detailed configuration changes depending on the post-buckled shape of the IAD and, to a lesser extent, on the stiffness of the deployed IAD. Figure 2(b) shows the simulated structure deformation under stable conditions (high torus inflation pressure) overlaid on a photograph from the experiment. Finally, Figure 2(c) shows the behavior of the drag as a function of time for stable (blue) and resonant (red) operating conditions. We will discuss the physical origin of the instability at the oral presentation.

CONCLUSIONS

Detailed results from compressible fluid-structure interaction simulations of an inflatable aerodynamic decelerator consisting of a polygonal tension cone supported by an inflated torus are discussed. We utilize large-eddy simulation, large deformation thin-shell mechanics and adaptive mesh refinement to study the behavior of the vehicle in the supersonic regime. Force coefficients for the capsule–tension-cone system are in broad agreement with experimental results. It is found that it is possible to drive the system to unstable operation by matching an acoustic frequency in the flow with the axial fundamental frequency of the vehicle. A detailed explanation and quantitative results will be presented in the oral version of the paper.

References

[1] Clark, I.G., Cruz, J.R., Hughes, M.F., Ware, J.S., Madlangbayan, A., Braun, R.D. (2009) Aerodynamic and Aeroelastic Characteristics of a Tension Cone Inflatable Aerodynamic Decelerator. In Proceedings: 20th AIAA Aerodynamics Decelerator Systems Technology Conference and Seminar, AIAA Balloon Systems Conference, 4-7th May 2009, Seattle, WA, U.S.A.

[2] Kramer, R.J. F. Cirak, and C. Pantano, "A Computational Fluid-Structure Interaction Study of Inflation Cycles of a Tension Cone Decelerator", 21st AIAA Aerodynamic Decelerator Systems Technology Conference and Seminar, Dublin, Ireland, May 23 - 26, 2011.

[3] Cirak, F., Ortiz, M., Schröder, P. (2000) Subdivision surfaces: A new paradigm for thin-shell finite-element analysis. *International Journal for Numerical Methods in Engineering*. **47**(12), 2039-2072.

[4] The Virtual Test Facility, California Institute of Technology, http://www.cacr.caltech.edu/asc.