6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

COMPUTATIONAL MECHANICS FOR LOW-DENSITY SUPERSONIC DECELERATORS

Charbel Farhat[1], Daniel Huang[2], Raunak Borker[3], Phil Avery[4] and Sebastian Grimberg[5]

[1] Department of Aeronautics and Astronautics, Department of Mechanical Engineering, and Institute for Computational and Mathematical Engineering, cfarhat@stanford.edu
[2] Institute for Computational and Mathematical Engineering, zhengyuh@stanford.edu
[3] Department of Aeronautics and Astronautics, rborker@stanford.edu
[4] Department of Aeronautics and Astronautics, pavery@stanford.edu
[5] Department of Aeronautics and Astronautics, sjg@stanford.edu
Stanford University
William F. Durand Building, 496 Lomita Mall
Stanford, CA 94305-4035, USA

Key Words: *Embedded Boundary Methods, Fluid-Structure Interaction, Parachutes, Turbulent Flows.*

In order to enable future exploration missions featuring sophisticated robots and safely land heavier spacecraft on Mars, parachute-based technology for decelerating a spacecraft from the high speed of atmospheric entry to the final stages of landing on Mars must be advanced to a new level of performance at supersonic speeds. For this purpose, larger than before high-speed parachutes and inflatable drag devices are needed. For a number of reasons ranging from cost to technical considerations, the design, development, and maturing of such devices for future use at Mars need assistance from predictive simulations based on a high-fidelity, multi-disciplinary, computational model for parachute inflation dynamics and drag prediction. The development of such a computational model is a formidable challenge. It must be able to predict shocks and wakes, as well as their effects on various instabilities of a parachute such as flutter and pulsation which may be encountered in the supersonic regime. It must be capable of assessing the influence on a parachute performance of several factors such as material and geometric porosities, the relative size of the parachute forebody with respect to its diameter, its distance from the forebody, the shape of the forebody, the line length, canopy design, and the Mach number. The computational model must also be able to predict the influence of temperature and strain rate on the stress field a parachute can experience in the supersonic regime. In short, the development of such a computational model requires a number of innovations in computational mechanics that are being pursued at Stanford University in collaboration with the Jet Propulsion Laboratory and the NASA Ames Research Center [1]. This lecture will discuss these innovations, with emphasis on: recent advanced in embedded boundary methods for turbulent viscous CFD and fluid-structure interaction problems [2]; sub-grid scale modeling of fluid-structure interaction effects [1]; adaptive mesh refinement and near-wall modeling in the context of RANS computations and LES on non body-fitted meshes [3]; and flux computations across porous material interfaces that may experience massive self-contact [1]. The lecture will also report on the results of some unprecedented simulations of parachute inflation dynamics at supersonic speeds that give some insight about the performance and structural integrity of low-density supersonic decelerators.

REFERENCES

[1] Z. Huang, P. Avery, C. Farhat, J. Rabinovitch, D. Derkevorkian and L. D. Peterson, Simulation of parachute inflation dynamics using an Eulerian computational framework for fluid-structure interfaces evolving in high-speed turbulent flows. *AIAA-2018-1540, AIAA SciTech 2018*, Kissimmee, Florida, January 8-12, 2018.

[2] D. Huang, D. De Santis and C. Farhat, A family of position- and orientation-independent embedded boundary methods for viscous flow and fluid–structure interaction problems. *Journal of Computational Physics*, Vol. **365**, pp. 74–104, 2018.

[3] R. Borker, S. Grimberg, P. Avery, C. Farhat and J. Rabinovitch, An adaptive mesh refinement concept for viscous fluid-structure computations using Eulerian vertex-based finite volume methods. *AIAA-2018-1072*, *AIAA SciTech 2018*, Kissimmee, Florida, January 8-12, 2018.