Materials Research Proceedings 33 (2023) 134-139 https://doi.org/10.21741/9781644902677-20

High-fidelity simulation of a supersonic parachute for Mars descent

L. Placco[1*], F. Dalla Barba[2] and F. Picano[1,2]

[1] Centro di Ateneo di Studi e Attività Spaziali 'Giuseppe Colombo' (CISAS), Università degli Studi di Padova, via Venezia 15, 35131 Padua, Italy

[2] Department of Industrial Engineering, Università degli Studi di Padova, via Venezia 1, 35131 Padua, Italy

*luca.placco@unipd.it

Keywords: Supersonic Parachute, Supersonic Flows, Large Eddy Simulation, Fluid-Structure Interaction

Abstract. The project aims to characterize the unsteady dynamics of the parachute-capsule in a supersonic flow during the descent phase on planetary entry. Presently, Large-Eddy Simulation in combination with an Immersed-Boundary Method is employed to analyze the time-evolving flow of a rigid supersonic parachute trailing behind a reentry capsule during the descent phase through Mars atmosphere. The flow is simulated at Ma = 2 and Re = 10^6. A massive GPU parallelization is employed to allow a very high fidelity solution of the multiscale turbulent structures present in the flow that characterize its dynamics. We show how the interaction of wake turbulent structures with the bow shock produced by the supersonic decelerator induces strong unsteady dynamics. This unsteady phenomenon called 'breathing instability' is strictly related to the ingestion of turbulence by the parachute's canopy and is responsible of drag variations and structure oscillations observed during previous missions and experimental campaigns. The next steps will take into account the flexibility of the parachute.

Introduction

The recent unsuccessful European missions (i.e. ExoMars 2016) proved how the prediction and the understanding of the dynamics of the descent capsule under the effect of a supersonic decelerator is still an open question in the active research scene that revolves around space exploration. The failure of Schiaparelli EDM landing indeed was ultimately caused by an improper evaluation of the coupled oscillatory motions existing between the descent module and the deployed parachute. The models and the experimental evaluations that were employed to predict the general behaviour of the capsule under the effect of a supersonic decelerator proved to be insufficient, triggering the premature end of the mission [1]. In this context, the main aim proposed by this research activity is to develop a novel technique to study effectively how compressible and turbulent flows interact with non-rigid structures, to properly evaluate and predict their non-steady behaviour. The interaction of a flexible body surrounded by a flow is considered as a constant presence in many different disciplines, despite being a very specific condition. It is acknowledged as fluid-structure interaction and plays also a major role in the study of flows that involve both space and suborbital applications. This is most true when considering planetary atmospheric reentry or the use of decelerators and active flying devices to extend the observation time of a probe. The non-linearity that characterises both the fluid and the solid behaviour proves to be still a challenge when approaching the subject and this becomes further complicated when considering unsteady compressible flows.

Materials Research Proceedings 33 (2023) 134-139 https://doi.org/10.21741/9781644902677-20

Case study and intended approach

The research community that involves fluid-structure interaction matters have always been present in the computational scene; the rising interest also on its implementation on advanced conditions is attracting further attention, especially for being applied on state-of-art cases and problems that have an immediate impact on the active missions and studies.

This is especially true for space programs, given the recent technological advancements concerning both exploration and robotics (i.e. the latest NASA's Mars Perseverance and the Chinese probe Tianwen-1). The typical case study of a supersonic decelerator (parachutes and inflated aeroshells) is representative for both the complexity of the study and for its scientific importance; ExoMars 2016 mission, which contained the Schiaparelli descent module, failed to demonstrate the entry, descent and landing procedure as, according to the Schiaparelli Anomaly Inquiry Report ([1]), the major event that brought to the saturation of the Inertial Measurement Unit was the highly dynamic oscillatory motion caused by an improper evaluation of the disk-gap-band type parachute that deployed at Mach 2.

The description of this phenomenon is very elaborate, being affected by several uncertainties such as atmosphere fluctuations, unsteady flow dynamics and structure oscillations [2],[3]. At the current time experimental procedures involving the design of these devices struggle to observe the properties of these problems since scale effects and the artificial environment recreated by wind tunnels are not able to replicate the effective operative conditions [4]. Flight tests instead are considered impractical, so they can only be employed at the end of the design process to obtain the validation that cannot be obtained in a simulated environment [5]. Even standard computational approaches appear to be limited in the provided results: Reynolds Averaged Navier Stokes technique (RANS), which is the engineering standard for design and validation campaigns, is unable to correctly reproduce the non-stationary nature of such turbulent flow conditions [4].

To overcome such limitations, the proposed approach involves more accurate methodologies which are Large-Eddy Simulations (LES) as they can properly represent the time-evolving multi-scale dynamics of the flow. They were always considered unfeasible due to the prohibitive computational cost required to run the analysis; however, the advent of GPU technology for parallel computing is enabling the usage of Large-Eddy Simulations [6], making them the most groundbreaking approach now available and increasing dramatically both the spatial and temporal resolution of the obtained results thus improving the quality and the quantity of the captured details. Furthermore, Immersed Boundary Methods, which are fundamental to deal effectively with moving solid boundaries, have received more attention recently [7], enabling the application of computational strategies for compressible flows that are able to solve the fluid-solid interface of porous thin structures and shells. Still, all of the analysis that have been performed on the matter up to now were not able to draw together all of these novel aspects, limiting the findings. For this matter, the ultimate aim of this ongoing research activity is to perform high-fidelity simulations of the full deployment and inflation sequence of a parachute for planetary descent in a supersonic regime as it interacts with the turbulent wake generated by the forebody.

Special attention will be given to the modeling of the typical 'breathing' behaviour of the supersonic decelerator during the descent phase of a reentry spacecraft. To achieve these results, a novel technique to deal with the fluid-structure interaction of compressible flows and thin membranes is in the process of development, starting from the existing Immersed Boundary Methods' strategies.

As a starting point for the implementation of the final configuration, a large-eddy simulation of a rigid mock-up parachute trailing behind a reentry capsule has been performed, showing both the potential of the LES approach and the primary dependence of the breathing phenomenon to the interaction of the turbulent wake of the descent module with the front bow shock produced by the inflated decelerator. We present the preliminary results obtained from the simulation.

Computational approach and simulation setup

Compressible Navier-Stokes equations are solved with the high-order finite difference solver STREAmS [6]. Turbulent structures are ultimately identified using the implicit large eddy simulation (ILES) approach; in this way, conventional LES turbulence modeling has been omitted, using instead the numerical dissipation given by the numerical discretization as artificial viscosity acting at small scales. Thus, the three-dimensional compressible Navier-Stokes equations solved are the following:

$$\begin{aligned} \frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_j)}{\partial x_j} &= 0, \\ \frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} + \frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j}\Big(\mu\Big(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\Big)\Big) &= 0, \quad (1) \\ \frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho E u_j + p u_j)}{\partial x_j} + \frac{\partial}{\partial x_j}\Big(\lambda\frac{\partial T}{\partial x_j}\Big) + \frac{\partial}{\partial x_j}\Big(\mu\Big(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\Big)u_i\Big) &= 0, \end{aligned}$$

where ρ is the density, u_i denotes the velocity component in the *i* Cartesian direction (*i =1,2,3*) and p is the thermodynamic pressure. With the intent of reproducing the effect of Mars' atmosphere, the fluid is considered as an ideal gas of CO_2; the ratio between the specific heat at constant pressure C_p and the specific heat at constant volume C_v is set to 1.3 while Prandtl number is 0.72. $E = C_v T + u_i^2/2$ represents the total energy per unit mass and the dynamic viscosity μ is assumed to follow the generalized fluid power-law. The thermal conductivity λ is related to μ via the Prandtl number with the following expression: $\lambda = C_p\mu/Pr$. Convective and viscous terms are discretized using a sixth-order finite difference central scheme while flow discontinuities are accounted through a fifth-order WENO scheme. Time advancement of the ODE system is given by a third-order explicit Runge-Kutta/Wray algorithm. No-slip and no-penetration wall boundary conditions on the body are enforced through an Immersed-Boundary Method (IBM) algorithm. The simulation was performed at $Ma = 2$ and $Re = 10^6$ to simulate the condition at which the parachute deploys. The reference fluid properties associated to the free-stream condition correspond to an altitude of about 9 km from the planet surface and have been obtained using a simulated entry and descent trajectory through the Mars atmosphere of a generic reentry probe [2].

The flow domain (fig. 1) selected to perform this first simulation has a size of $L_x = 20D,\ L_y = 5D,\ L_z = 5D$, where *D* is the maximum diameter of the descent module; parachute diameter is set to 2.57*D.* the mesh is a rectilinear structured grid that consists of $N_x \cdot N_y \cdot N_z = 2560 \cdot 840 \cdot 840$ nodes. The grid density changes in both axial and transverse directions, gaining resolution in the central portion of the domain; the position of the capsule nose is set at [1*D*,0,0] while the parachute center lies at [10*D*,0,0]. Computations have been carried out on CINECA Marconi100 cluster, allowing the domain parallel computing on a total of 64 GPUs.

Materials Research Proceedings 33 (2023) 134-139
https://doi.org/10.21741/9781644902677-20

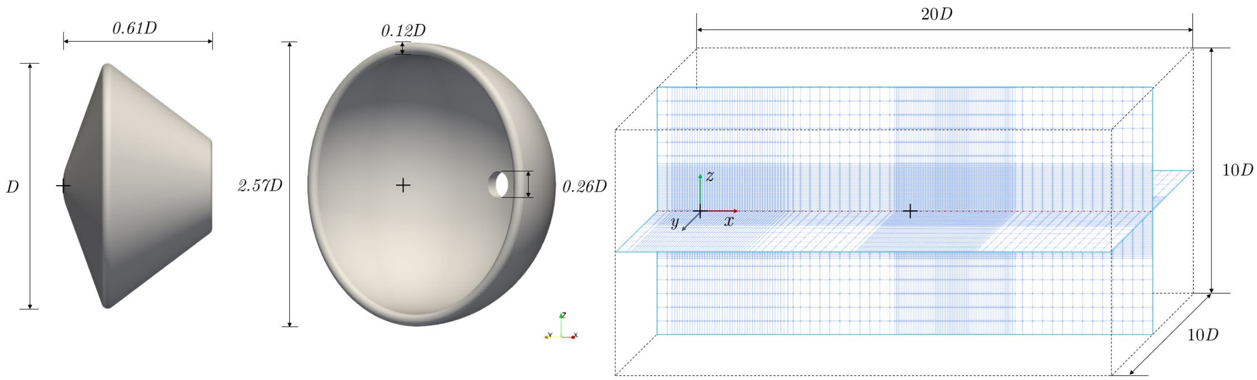

***Figure 1**: Setup of the simulation domain and dimensions of the descent module and parachute.*

Results

In figure 2 we observe the two-dimensional instantaneous flow field obtained by isolating the $y = 0$ slice from the full 3D domain; Mach number contours are shown. Subsonic flow regions (in red), sonic regions (in white) and supersonic areas (in blue) can be identified. We observe the generation of two bow shocks ahead of the capsule and the canopy. Subsonic regions are concentrated around the axis of symmetry, behind the two bow shocks. The highest velocity is reached by the flow exiting the vent, with a Mach number of $Ma \cong 3.5$. The flow at the vent section is sonic. Pushed by the high pressure within the canopy and finding a larger passage section, it rapidly accelerates to the highest Mach number of the flow field. The high value associated to the Reynolds' number results in the generation of an extended wake behind the capsule. Near wake regions are characterized by subsonic recirculations while supersonic regime supersedes as the flow progresses towards the outflow. After interacting with the canopy bow shock, the flow develops into vortical structures all around the canopy and at its back, producing large subsonic regions.

The capsule bow shock is steady, as well as the subsonic region between the shock and the capsule. On the contrary, the flow appears more unstable in the canopy region: the source of this instability is the turbulence shock interaction occurring due to the passage of the turbulent wake of the descent module through the bow shock of the parachute. The intensity of turbulence carried by the wake is amplified as the flow travel across the canopy bow shock (turbulence ingestion by the shock), leading to large fluctuations of momentum and pressure. In addition, being disrupted by the irregularities of the wake, the canopy bow shock does not reach a steady state and shows an oscillatory motion. This motion is related to the parachute breathing cycle, which is present regardless the parachute rigidity. The breathing motion involves inhomogeneous pressure/density fluctuations, leading to large drag variations, despite the canopy area remaining constant. Main cause of the breathing cycle seems to be the aerodynamic interaction between capsule wake and canopy bow shock. In real applications of flexible parachutes, canopy deformations even couple with this interaction, amplifying drag variability. The deformability effect of thin porous membranes will be taken into account in the following steps of the research project. Figure 3 shows the different phases of the cycle that surrounds the periodic motion of the front bow shock along the flow direction: an increasing density inside the canopy pushes the shockwave away, allowing a larger flux to escape from the canopy (from [1] to [2]). Thus, this creates a decrease in the density that in turn draws back in the shockwave ([2] to [3]) and restarts the cycle ([3] to [4]). Further details on the flow dynamics will be given during the conference.

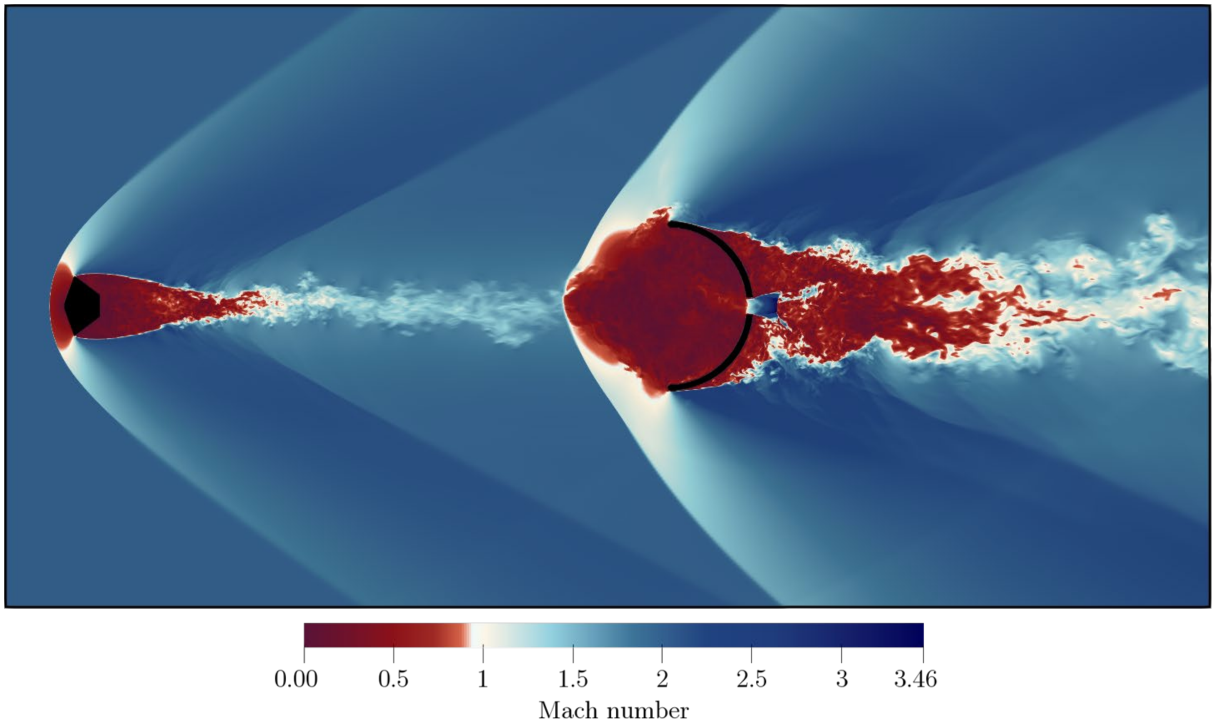

Figure 2*: Instantaneous Mach contours (y = 0 cross section) of the simulated flow domain.*

Conclusions

The present work proposes an high-fidelity time-evolving simulation of the interaction between the turbulent wake of a supersonic descent module and a generic rigid artificially thick decelerator. We show how the critical 'breathing' instability associated to supersonic parachutes is intrinsically connected to the interaction of the turbulent wake flow of the descent module and the front bow shock produced by the decelerator. To overcome the limitation of the current setup and further extended the representation of its dynamics, the implementation of a novel immersed boundary method technique is in progress. This will require the solution of fluid-structure interaction of compressible supersonic flows and flexible thin membranes. The new framework will involve an extension of the current IBM module and a finite element method model to deal with flexible moving boundaries (zero-thickness), representing the very thin structure of the simulated decelerator. In this way, the approach in development will allow to represent properly both the entire deployment sequence and the system unsteadiness in all its components, thus providing the full representation of the 'breathing' phenomenon.

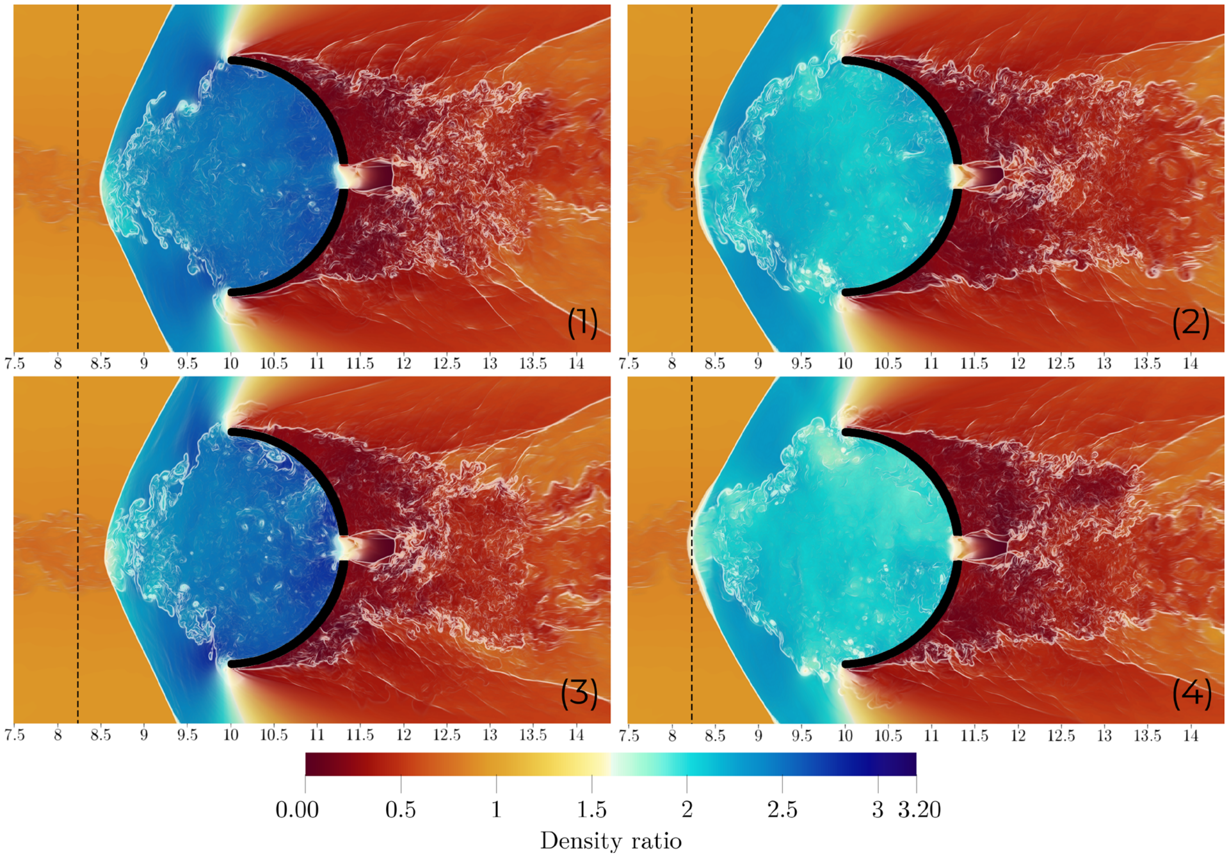

Figure 3*: Instantaneous density ratio contours (y = 0 cross section) at different progressive timestep around the parachute canopy.*

References

[1] T. Tolker-Nielsen. EXOMARS 2016 - Schiaparelli Anomaly Inquiry, 2017.
[2] A. Aboudan, G. Colombatti, C. Bettanini, F. Ferri, S. Lewis, B. Van Hove, O. Karatekin, and Stefano Debei. Exomars 2016 schiaparelli module trajectory and atmospheric profiles reconstruction. Space Science Reviews, 214: 97, 08 2018. https://doi.org/10.1007/s11214-018-0532-3
[3] X. Xue and Chih-Yung Wen. Review of unsteady aerodynamics of supersonic parachutes. Progress in Aerospace Sciences, 125:100728, 2021. ISSN 0376-0421. https://doi.org/10.1016/j.paerosci.2021.100728
[4] Nimesh, Dahal. Study of pressure oscillations in supersonic parachute. International Journal of Aeronautical Space Sciences, (19):24-31, 2018. https://doi.org/10.1007/s42405-018-0025-3
[5] B. S. Sonneveldt, I. G. Clark, and C. O'Farrell. Summary of the Advanced Supersonic Parachute Inflation Research Experiments (ASPIRE) Sounding Rocket Tests with a Disk-Gap-Band Parachute, AIAA 2019-3482. https://doi.org/10.2514/6.2019-3482
[6] M. Bernardini, D. Modesti, F. Salvadore, and S. Pirozzoli. Streams: a high-fidelity accelerated solver for direct numerical simulation of compressible turbulent flows. Computer Physics Communications, 263:107906, 2021. https://doi.org/10.1016/j.cpc.2021.107906
[7] H. Yu and C. Pantano. An immersed boundary method with implicit body force for compressible viscous flow. Journal of Computational Physics, 459:111125, 2022. https://doi.org/10.1016/j.jcp.2022.111125