Multiphysics and Environmental Modelling

FLUIDYN'S CONTRIBUTION IN CHANDRAYAAN-3

Chandrayaan-3's journey to success was a collaborative effort. From the skill-full hands working on-site meticulously crafting and assembling components, to the engineers who fine-tuned every detail, scientists who conceptualized and refined the mission, and the visionary national leaders who recognized the importance of pushing the boundaries of exploration.

We are immensely proud that Fluidyn software tool was trusted by the Launch Pad Safety and cryogenics teams to ensure a smooth mission execution. The Cryogenics team used our tool fluidyn-MP to address various kinds of cryogenic flow problems.

It's a testament to the dedication of Fluidyn's team that our technology played a pivotal role in such a significant mission.

Risk Assessment of accidental spillage of Launch Vehicle Fuel for SDSC-SHAR

Abstract:

Satish Dhawan Space Centre (SDSC) SHAR, one of the R&D wings of ISRO at Sriharikota, is the spaceport of India and is responsible for providing Launch Base Infrastructure for the Indian Space Program. The centre has two launch pads from where the rocket launching operations of PSLV and GSLV are carried out.

The safety analysis of various facilities handling highly flammable rocket propellants was carried out by 3D CFD based dispersion tool, *fluidyn*-PANEPR as sought by the high profile technical review committee. Emergency preparedness during any eventual accidental spillage of the propellants at different storage & handling scenario were considered. The geometrical model (Fig. 1) setup included all those microscale elements that are bound to influence the air flow and vapour dispersion, such as land cover (land-sea interaction), centre facilities (buildings, tanks, LPs etc..).

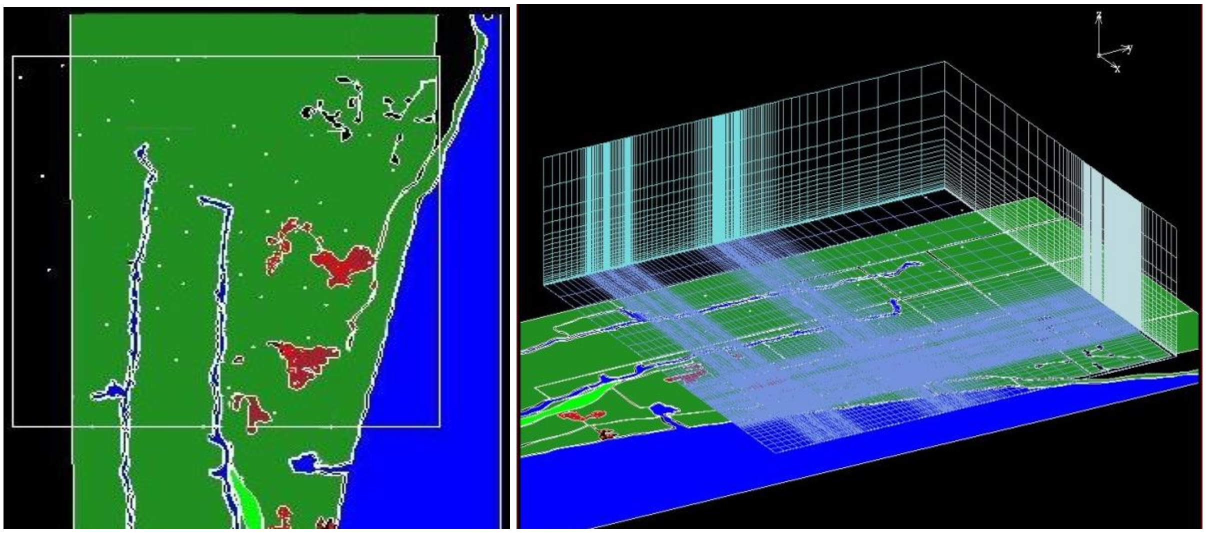


Fig. 1: Site Topography considered for the study along with 3D mesh

The wind flow analysis was carried out using built-in atmospheric boundary layer models incorporating site specific atmospheric turbulence and recirculation. The CFD outputs in terms of flammable clouds and transient vapour footprints were intended to be used for emergency preparedness. Virtual measurement stations were placed all across the domain to record the variation of propellant concentration v/s time.

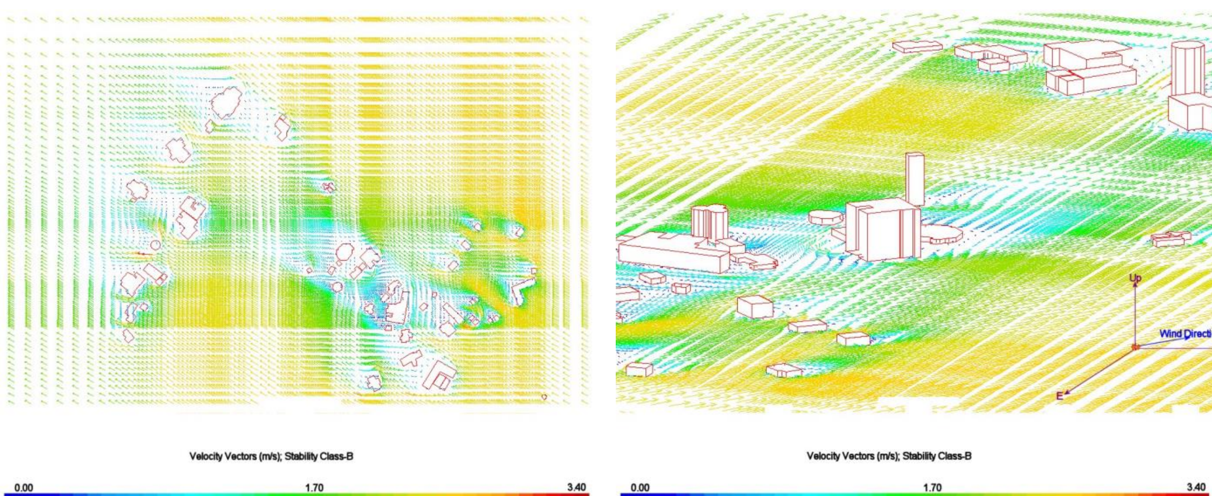


Fig. 1: Wind flow analysis in and around the study region

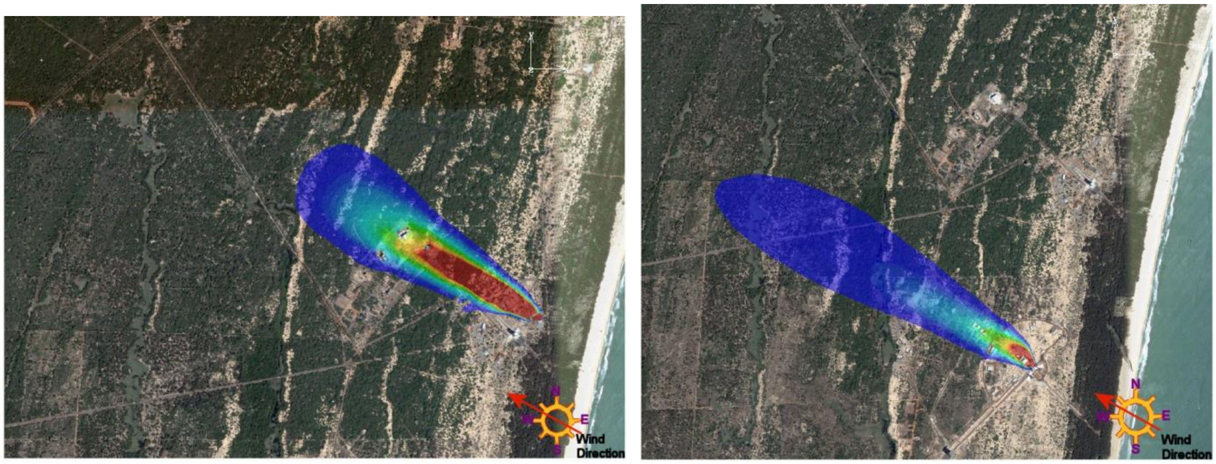
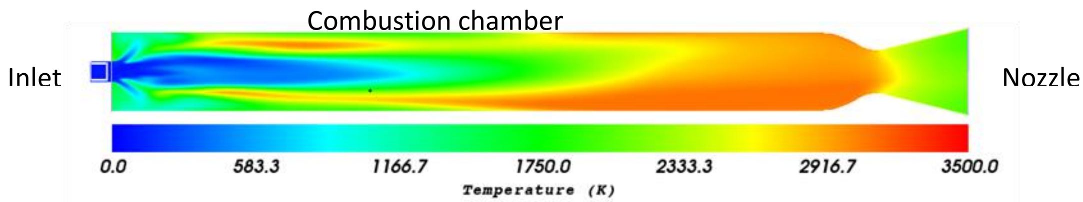


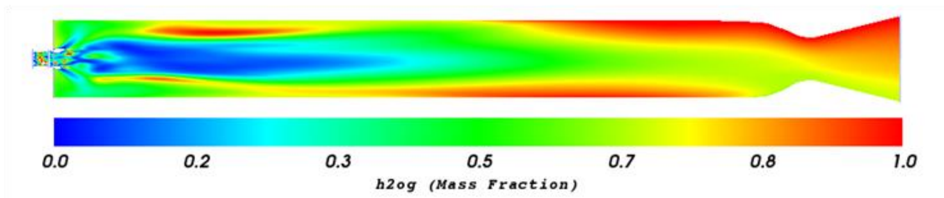
Fig. 2: Dispersion of spilled propellants for one of the weather scenario considered

Cryogenic Combustor Simulation using Fluidyn

Cryogenic propellants (liquid hydrogen and liquid oxygen) are preferred for satellite launch vehicles, because they produce the highest exhaust velocities of all the propellant combinations, despite their extreme complexity in terms of storage and handling. Continuous increase in the payload weight demands development of cryogenic rocket engines with higher thrusts and hence higher pressures and introduce additional flow features such as trans-critical flows. The design, development and testing of these engines is a highly time and resource consuming processes and computer simulations based on 3D CFD are being extensively used to reduce the development efforts. However, the processes inside the injectors, combustors and nozzles of these engines are very complex involving high speed trans-critical flows, swirling, mixing, atomization, evaporation, chemical reactions, conjugate heat transfer and erosion. This makes such simulations highly challenging. Fluidyn has developed and validated multi-physics models and transient CFD methods to simulate these processes in a coupled manner and has been extensively used to simulate various operating conditions and design configurations involving different test combustors. The simulated combustors include the shear co-axial injectors from the Mascotte test cases and the swirl co-axial injectors from ISRO cryogenic engine development cycle.



Temperature distribution in the combustor



Combustion product (water vapour) distribution