Towards Hybrid Grid Simulations of the Launch Environment

S. Moini-Yekta[†], M. Barad[*], E. Sozer[‡], J. Housman[*], C. Brehm[‡] and C. Kiris[*]
Corresponding author: shayan.moini-yekta@nasa.gov

[*] NASA Ames Research Center, USA.
[†] University of California Davis, USA.
[‡] Science and Technology Corporation, USA.

Abstract: A hybrid grid approach has been developed for the simulation of next generation heavy-lift space vehicles in the launch environment. The motivation for the hybrid method is to reduce the turn-around time of computational fluid dynamic (CFD) simulations and improve the ability to handle complex geometry and flow physics. The hybrid scheme, LAVA-Cartesian/Unstructured (Launch Analysis and Vehicle Aerodynamics), consists of two solvers: an off-body immersed-boundary Cartesian solver LAVA-Cartesian with block-structured adaptive mesh refinement and an off-body unstructured solver LAVA-Unstructured with conjugate heat transfer. Two-way coupling is achieved through overset connectivity between the off-body and near-body grids. This work seeks to determine the best practices of the individual flow solvers, perform verification with code-to-code comparisons and validation using experimental/flight data. Analysis is performed on a representative two-dimensional and three-dimensional unsteady Space Shuttle (STS-135) test case.

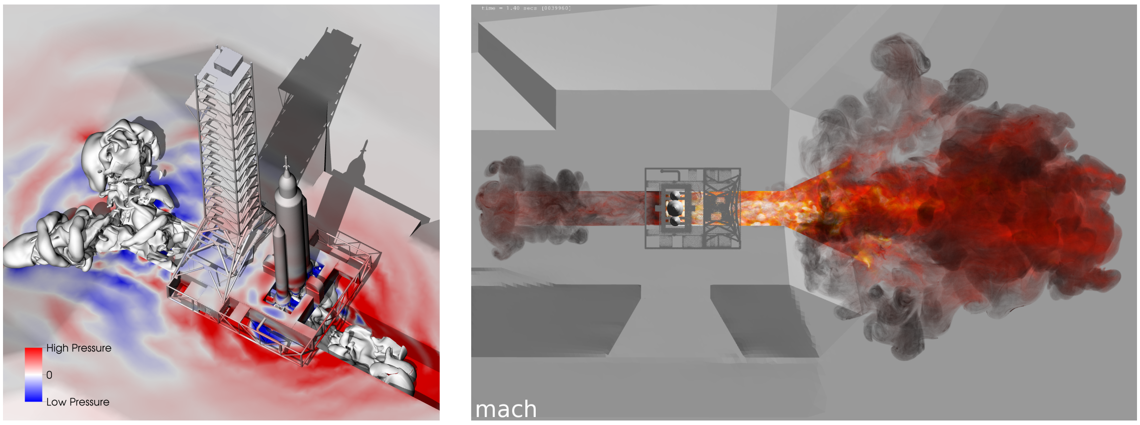
Keywords: Numerical Algorithms, Computational Fluid Dynamics, Cartesian Immersed Boundary, Unstructured, Overset, Hybrid Grid Methodologies, Adaptive Mesh Refinement.

1 Introduction

NASA is currently developing a heavy-lift launch vehicle and next generation launch site to carry large payloads for future human exploration missions beyond low Earth orbit. The greater thrust of heavy-lift vehicles requires accurate analysis to ensure vehicle stability, payload safety, and durability of the jet plume impingement region of the launch pad. CFD support is essential in the analysis and design of vehicles and the induced launch environment. High fidelity CFD simulations allow for the rapid and accurate analysis of vehicle and launch site configurations and designs [1]. The simulations provide time-dependent pressure and thermal loading for large-scale trade and comparison studies. With these capabilities, both vehicle and launch site conceptual designs and configurations can be quickly iterated on during design cycles.

2 Approach and Results

The launch environment contains a variety of highly complex geometric details (Figure 1) and flow physics that are challenging to model with current CFD methods. Examples of this include: multi-phase reacting flow and unburned particles from engines, interaction of jet plumes with the water sound suppression system and unsteady shock structure development. The focus of this work is to assess the performance of a simplified physics models for the pressure and thermal environments. During ignition of the solid rocket boosters (SRB) an ignition overpressure (IOP) is generated and travels between the mobile launch platform (MLP), the main flame deflector (MFD) and the vehicle. The IOP wave occurs during the first second of launch and may affect the stability of the vehicle. The



(a) (b)

Figure 1: Examples of CFD simulation of the launch environment: (a) pressure signature and plume iso-surface and (b) top view of particles visualization colored by Mach number (b).

immersed-boundary Cartesian module of the LAVA framework is used, LAVA-Cartesian, is utilized for the pressure environment. With adaptive mesh refinement (AMR), LAVA-Cartesian has the capability to track and resolve flow features such as pressure waves. This methodology is capable of automatically generating, refining, and coarsening nested Cartesian volumes. Immersed-boundaries are treated with a sharp interface ghost-cell technique, similar to [2].

Thermal analysis of the launch environment focuses on the main flame deflector, which must withstand the harsh conditions of vehicle launches. Refractory material coating is applied to the MFD to absorb the high temperatures, heat rates and protect against erosion and debris. Boundary layer resolution is critical for the heating and shear prediction for such environments. An unstructured approach is used in order to model complex geometry with viscous wall spacing. An arbitrary polygon unstructured solver, LAVA-Unstructured, is used with a conjugate heat transfer method for surface heat transfer between the fluid and solid interface. With the conjugate heat transfer method the properties of the refractory material can be modeled which is not possible with adiabatic simulations.

A hybrid LAVA-Cartesian/Unstructured grid approach is motivated by CFD prediction requirements of both pressure and thermal environments. More specifically, the hybrid approach seeks to reduce CFD simulation turn-around times and improve the ability to handle complex geometries and flow physics. LAVA-Cartesian/Unstructured seeks to combine the computational efficiency and AMR capabilities of LAVA-Cartesian and the flexibility of unstructured grids with LAVA-Unstructured. Unstructured near-body meshes can be used for complex geometries and regions in which viscous resolution is required, while an immersed-boundary AMR off-body Cartesian mesh can be used to track flow features. To achieve the proposed hybrid approach code-to-code comparisons are done on a two-dimensional trench case and validation is done on a Space Shuttle (STS-135) test case with flight data.

References

[1] Kiris, C., Housman, J., Gusman, M., Chan, W., & Kwak, D. Time-Accurate Computational Analysis of the Flame Trench Applications. *21st Intl. Conf. on Parallel Computational Fluid Dynamics*, pp. 37-41, 2009.

[2] Mittal, R., H. Dong, M. Bozkurttas, F.M. Najjar, A. Vargas, A. von Loebbecke, A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries. Journal of Computational Physics, 227(10), pp 4825-4852, 2008.