Adaptive Immersed Boundary Simulations for the Launch Environment

M. Barad, J. Housman, and C. Kiris
Corresponding author: michael.f.barad@nasa.gov

NASA Ames Research Center, USA.

Abstract: A high-fidelity computational fluid dynamics simulation of a next generation heavy lift space vehicle during launch is presented. The purpose of the simulation is to evaluate the acoustic overpressures during ignition to permit re-design of the launch site to safely handle heavy lift vehicles. The simulation is performed using the Launch, Ascent, and Vehicle Aerodynamics (LAVA) code, an immersed boundary block-structured Cartesian adaptive mesh refinement based solver. A verification and validation study of LAVA in the launch environment context is also performed, comparing to flight data and previous simulations of a Space Shuttle launch.

Keywords: Ignition Overpressure Waves, Computational Fluid Dynamics, Adaptive Mesh Refinement, Immersed Boundary, and Time Dependent.

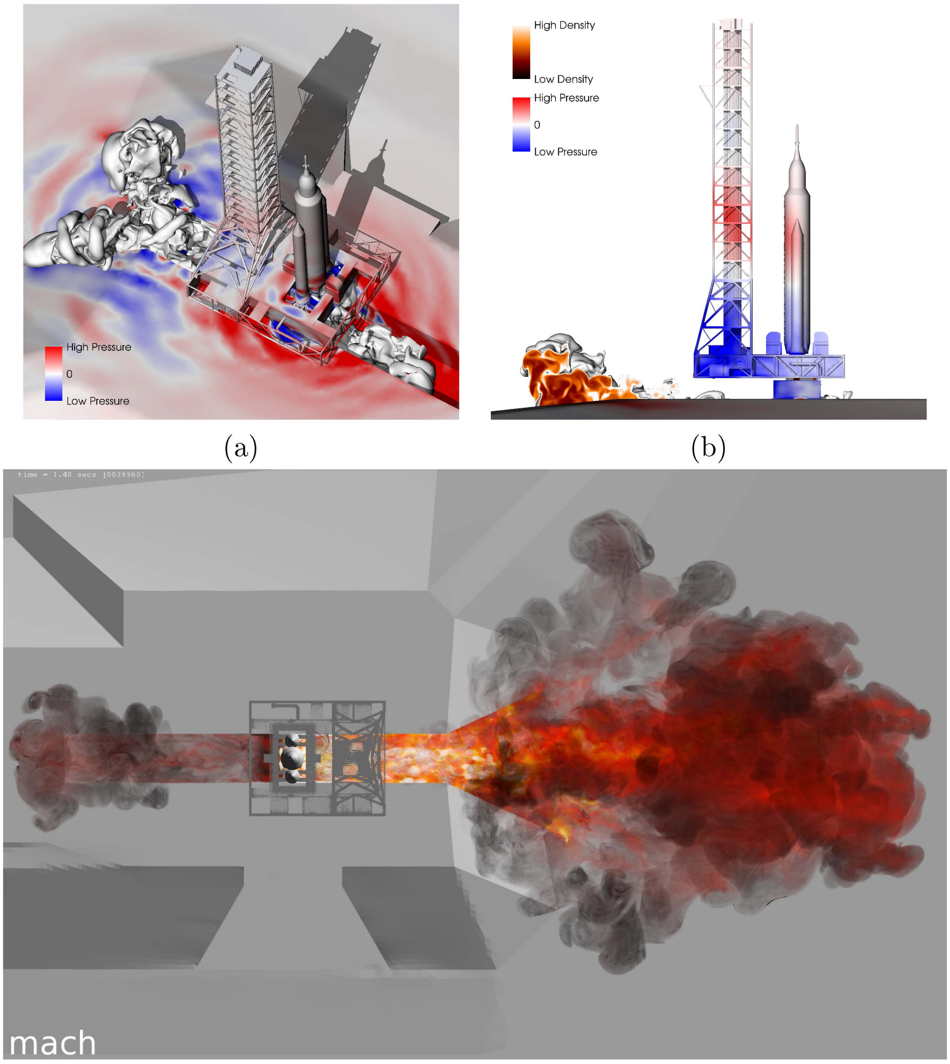
1 Introduction

NASA is currently developing a heavy-lift launch vehicle 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. High-fidelity CFD simulations of ignition overpressure (IOP) and acoustic phenomena during ignition and liftoff have been performed using the existing launch pad configuration and a preliminary heavy-lift vehicle design (see Figure 1). Results from the analysis will help reconfigure the launch pad to reduce the overpressure and sound pressure levels (SPL) of the heavy lift vehicle. During ignition of the solid rocket boosters (SRB), an IOP wave is generated and travels between the mobile launch platform, the main deflector, and the vehicle. The IOP wave occurs during the first second of launch and may affect the stability of the vehicle. After that, the vehicle lifts off and its exhaust plumes impinge on the pad, generating potentially damaging SPL near the payload. The first 1.4 seconds of SRB ignition is simulated, including plume development, overpressure wave propagation and impingement in the flame trench, interaction of SRB and liquid engine plumes, pressure wave propagation past the mobile launch platform towards the vehicle and tower. In the simulation, the vehicle is held fixed and water suppression system effects are ignored.

2 Approach and Results

Ignition and liftoff at the launch site has a large range of spatial scales, requiring a multi-resolution numerical method. The calculation uses a block-structured Cartesian adaptive mesh refinement (AMR) immersed boundary (IB) version of the LAVA code. Boundaries are treated with ghost-cells, similar to [3]. This methodology is capable of automatically generating, refining, and coarsening nested Cartesian volumes. The launch site geometry is highly complex (see Figure 1) and would take months to manually generate structured body-fitted grids. LAVA's AMR-IB method is designed to automatically generate the volume grids from a closed surface triangulation, and dynamically track important flow features as they develop. Accurate and efficient prediction of the IOP waves may be critical for rapid design and mission success of space vehicle launch systems.

Spatial and temporal resolutions used for the present simulations are based on the studies performed in [1] for simplified launch environment test cases. A verification and validation study of the LAVA code in this context is also performed, comparing to flight data and previous simulations of a Space Shuttle launch [2]. Inviscid and off-body RANS approaches will be assessed during the verification and validation.



(a) (b)

(c)

Figure 1: (a) & (b) Pressure signature and plume isosurface. (c) Visualization particles colored by Mach number (top view).

CFD analysis will provide launch pad design teams with estimates of unsteady pressure signatures on the surface of the launch pad and vehicle. These estimates will be used to guide modifications to the plume deflectors, which are located directly beneath the vehicle and serve to direct exhaust away from the pad. In addition, an assessment of plume deflection and structural loadings will be presented. For accurate SPL prediction away from the plume, an acoustic solver is coupled to the CFD to extrapolate IOP waves. A proposed companion paper will focus on acoustic predictions.

References

[1] Housman, J., M. Barad, and C. Kiris. Space-Time Accuracy Assessment of CFD Simulations for the Launch Environment. 29th AIAA Applied Aerodynamics Conference, Honolulu, Hawaii, June 27-30, 2011. AIAA-2011-3650.

[2] 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.

[3] 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.