Numerical Simulation and Physical Validation of Regolith Ejecta During Plume Surface Interaction

Dutton Webb, Grant Garrison
Advisor: Dr. Bin Xiao

Introduction

The simulation of vacuum plume dynamics involves capturing the intricacies resulting from the transition of continuum flow to free-molecular flow. Rarefied gas flows can cause significant lunar surface erosion and high-velocity dust clouds. Understanding exhaust plume behavior and its interaction with lunar regolith requires simulating both continuum and rarefied gas flow regimes.

Ansys FLUENT, a widely used CFD software, excels in the continuum flow regime by solving the Navier-Stokes equations. However, these equations become unreliable where Knudsen numbers of the flow regimes exceed 0.01. Rarefied gas flows are better solved using the Boltzmann equation, but for large-scale simulations this requires significant computational resources.

To address this, we are integrating the Direct Simulation Monte Carlo (DSMC) method with Ansys FLUENT using custom user-defined functions (UDFs). UDFs can be used to customize and control various aspects of the simulation, such as boundary conditions, material properties, source terms, and solution monitoring and can manifest as complex algorithms. In this case, we are using it to calculate Knudsen number values and to perform DSMC to simulate gas molecules and particle interactions with the rarefied gas regime.

By applying DSMC locally where rarefied gas is present, we can enhance computational efficiency, leveraging the strengths of both methods to more easily simulate and understand the effects of rocket exhaust impingement on the lunar surface. While this method can be scaled for use with high-performance computing clusters, the goal is to perform accurate solutions efficiently on less powerful systems.

Academic Partnership

Ansys® Academic Research Mechanical and

Modeling

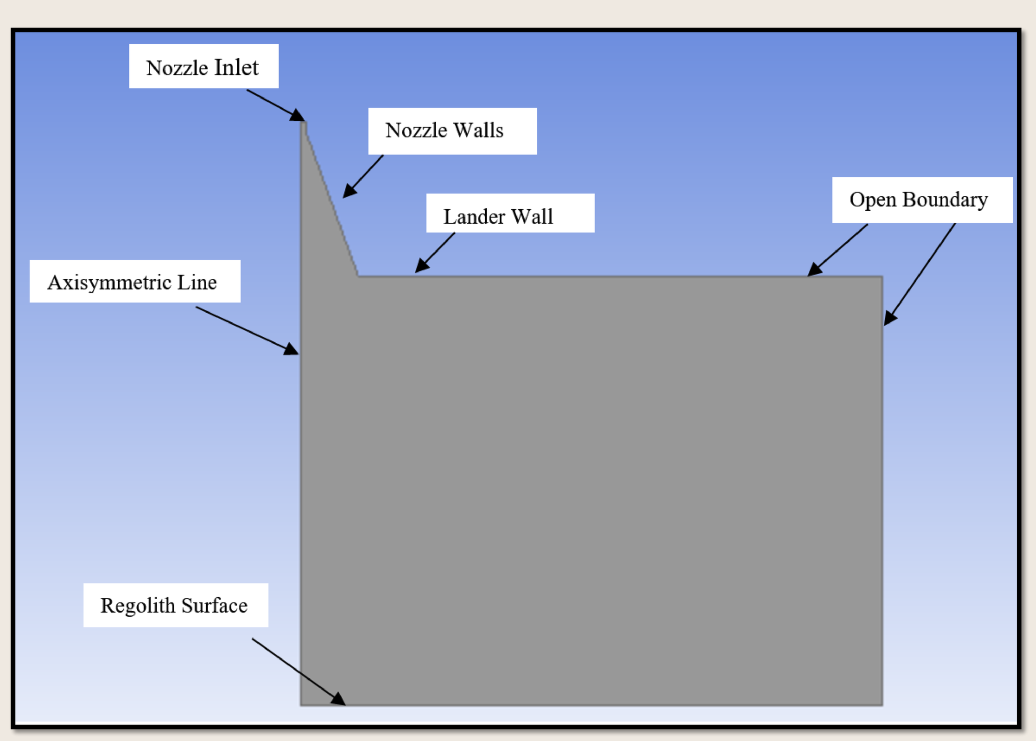


Figure 1. An axisymmetric domain and nozzle geometry approximate the Blue-Origin BE-7 engine at an elevation of 4 meters.

- Nozzle dimensions based on Blue Origin's BE-7 engine
- Inlet pressure derived from thrust specifications and approximated geometry
- Open boundary in far field
- Axisymmetric along central axis of nozzle
- Domain measures 6 meters by 6 meters
- Several particle layers simulating lunar dust ranging from 10 µm to 70 µm along the bottom region of the domain

CFD-DSMC Hybrid Model Algorithm

DSMC
DSMC is a probabilistic method that directly models the statistical behaviors of physical quantities. In this case, the gas molecules are simulated and lunar regolith particles are statistically sampled in the regions of interest. Particle velocities, positions, and collision frequencies are tracked.

The Knudsen Number (K_n)
A dimensionless parameter defined by the ratio of the mean free path (λ) of molecules in the flow and the characteristic length scale (L) of the system. By calculating the Knudsen number for each cell in the domain, DSMC can be applied to a target range of the flow regime. Balancing solve time and solution accuracy is the product of this CFD-DSMC Hybrid method.

For regions where $K_n \leq 0.1$, the Navier-Stokes Equations provide a more efficient solution. In contrast, for regions where $K_n \geq 0.1$, the Boltzmann equation allows for improved accuracy.

Knudsen Value	Type of Model
$K_n \leq 0.001$	Continuum Regime (Navier-Stokes equations)
$0.001 \leq K_n \leq 0.1$	Slip Regime (N-S with velocity-slip and temperature-jump boundary conditions)
$0.1 \leq K_n \leq 10$	Transition Regime (Boltzmann equation)
$K_n \geq 10$	Free-molecular Regime (particle methods such as DSMC)

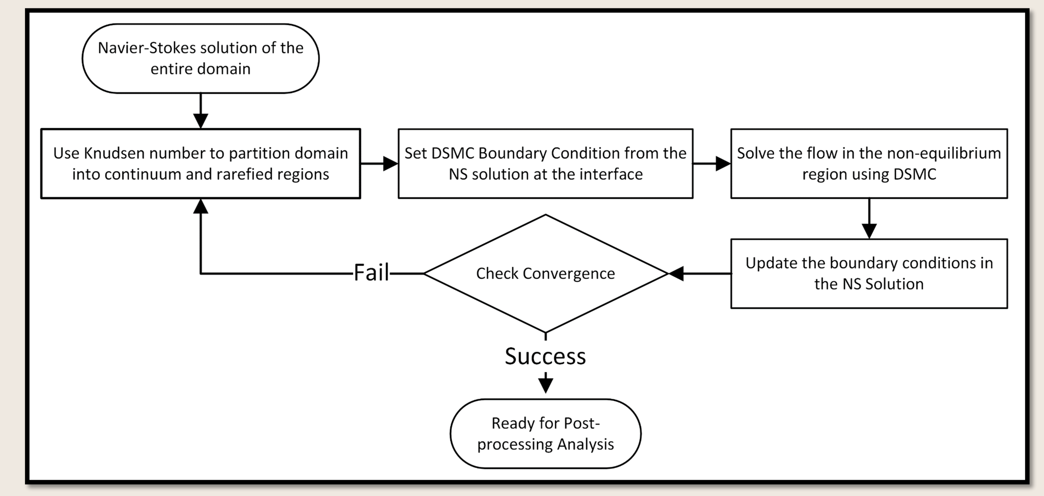


Figure 3. CFD-DSMC Hybrid Method Algorithm

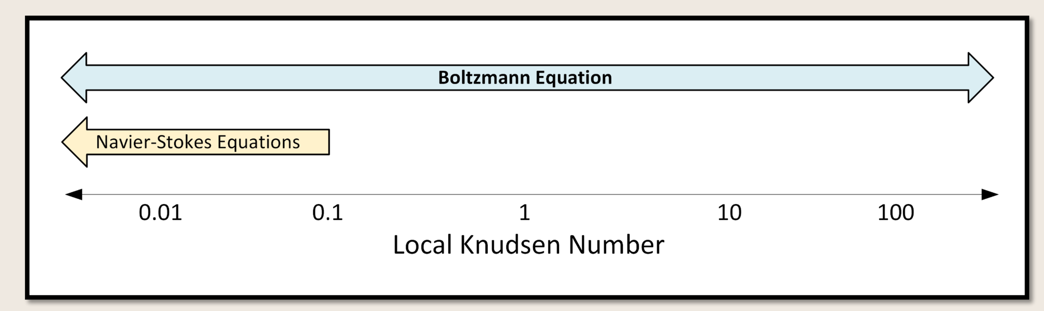


Figure 4. While the Boltzmann equation can be applied across the range of flow regimes, solving the equation with DSMC requires significant computational resources. The CFD-DSMC hybrid method leverages the efficiency of the Ansys FLUENT continuum solver, implementing DSMC only where rarefied gas regime is detected.

Preliminary Results

Four layers of dust cover the regolith surface. The dust particles range in size from 10 micrometers to 70 micrometers, with an average size of 50 micrometers. The open boundary pressures are kept at zero, and the nozzle inlet pressure is set to 3000 psi. The nozzle walls are treated as no-slip walls.

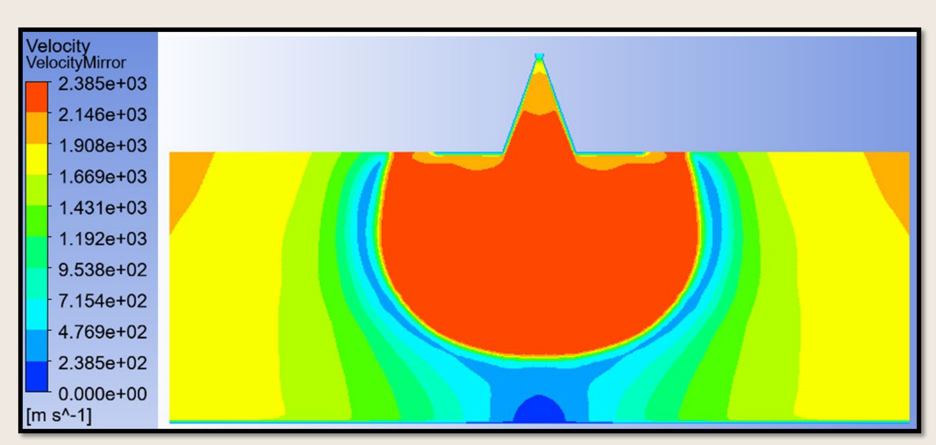


Figure 4. Under high pressure, the plume accelerates through the nozzle's throat. Upon exiting, it expands and slows before reaching the regolith. After impact, the plume reflects and accelerates diagonally on both sides.

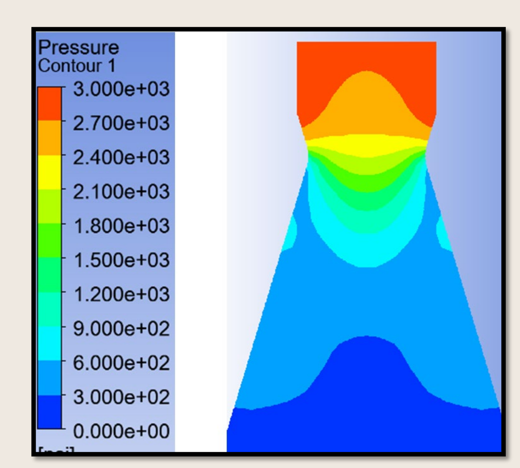


Figure 5. This pressure distribution is typical of fluid flow through a converging-diverging nozzle.

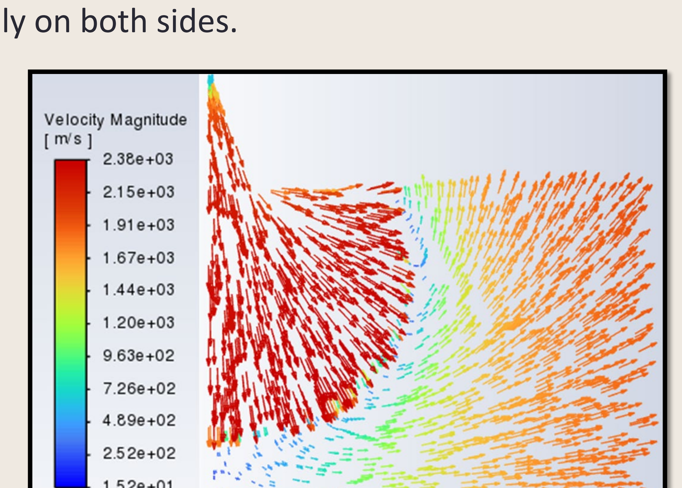


Figure 6. The plume extends along the nozzle wall, slowing before reaching the regolith. On impact, it rebounds and accelerates diagonally sideways.

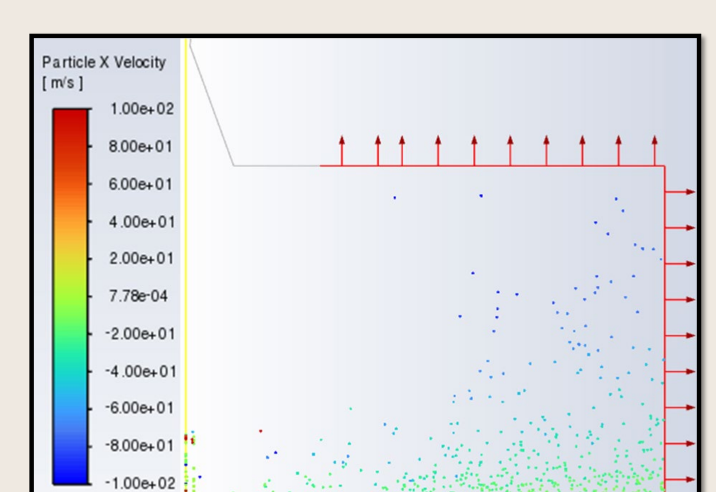


Figure 7. The dust particles on the regolith surface are propelled sideways and upwards by the plume flow, gaining acceleration as they rise.

Physical Testing

Experimental Setup

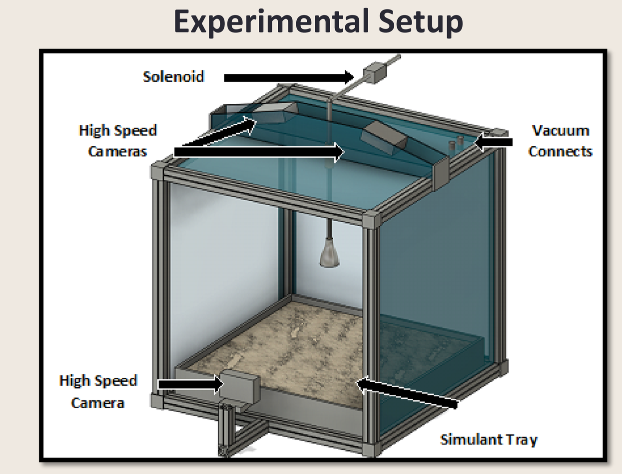


Figure 8. Testing chamber measuring the plume surface interactions

Particle Image Velocimetry (PIV) System

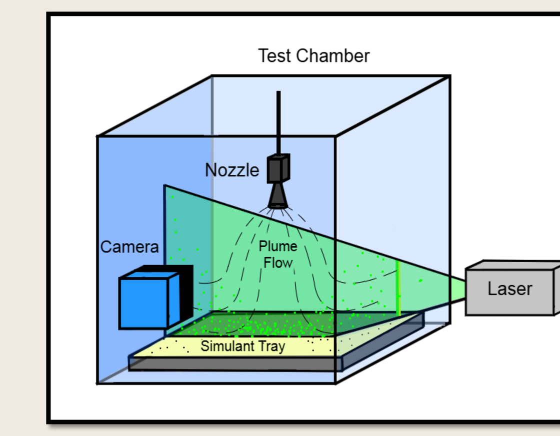
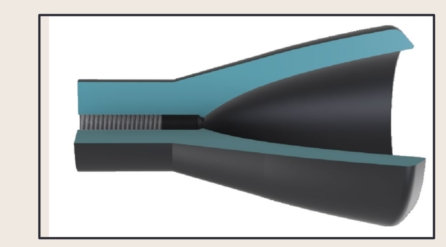
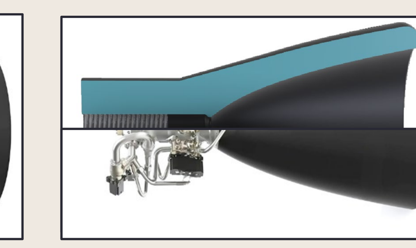


Figure 9: PIV System illuminates particles in plume flow



Cold Gas Thruster Nozzle Section View

Blue Origin's BE-7 Engine



Nozzle Contour Comparison, 12:1 scale

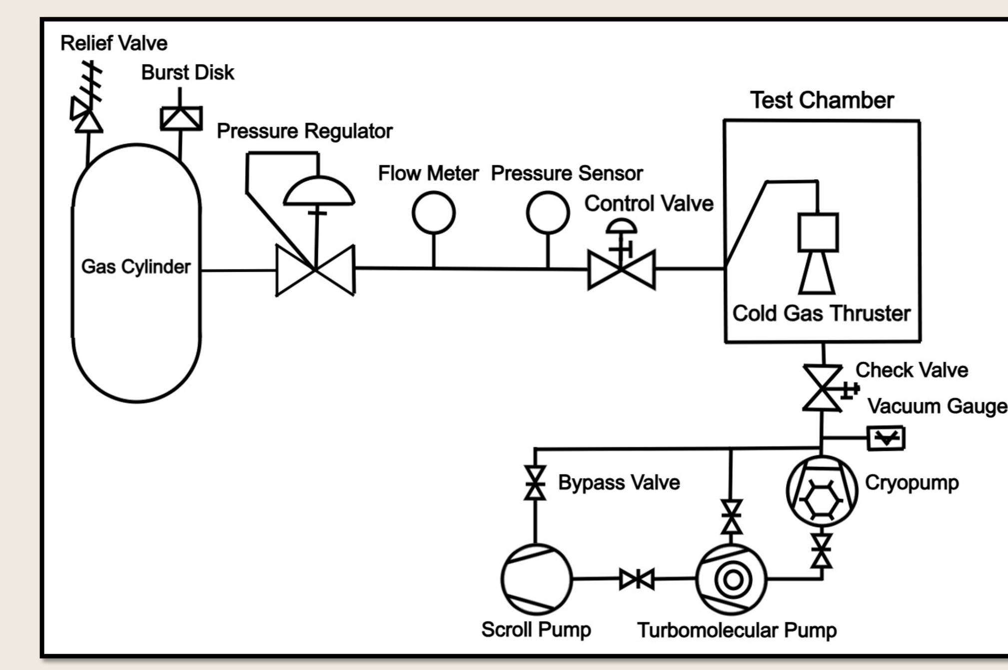


Figure 10. High-Pressure Gas Supply and Vacuum System

Figure 10 High-speed camera Mikrotron EoSens 4.0MCX6
566 fps standard resolution
38,502 fps reduced resolution

Data Acquisition

1. Standard DAQ systems can effectively record pressure data measured by sensors within the test chamber.
2. Image acquisition hardware interfaces with high-speed cameras to capture and store images depicting the flow field
3. specialized Particle Image Velocimetry (PIV) software processes these images, extracting valuable velocity information.
4. By correlating tracer particle movement across consecutive frames, this software computes velocity vectors within the flow field
5. specialized image analysis software is employed to track individual particle trajectories over time, allowing for the calculation of particle velocities, trajectories

Conclusion

1. The psi phenomenon associated with the lunar lander's landing process is simulated using Ansys FLUENT.
2. The Discrete Phase Model (DPM) is utilized to track particle trajectories, while the Direct Simulation Monte Carlo (DSMC) model is employed to capture the plume flow within the rarefied area.
3. Testing devices, instruments, and procedures for future physical validation tests are developed.
4. Preliminary numerical results of plume fluid flow and dust particle trajectories indicate that the current FLUENT model accurately reflects nozzle flow and plume surface interactions.