A coupled fluid-structure simulation for fracturing ice particles in hypersonic flows

Michael W. Belair, Carlos E. Americo, Ethan H. Huff, and Savio J. Poovathingal

Department of Mechanical and Aerospace Engineering, University of Kentucky, Lexington, KY 40506, USA
Michael.Belair@uky.edu, Saviopoovathingal@uky.edu

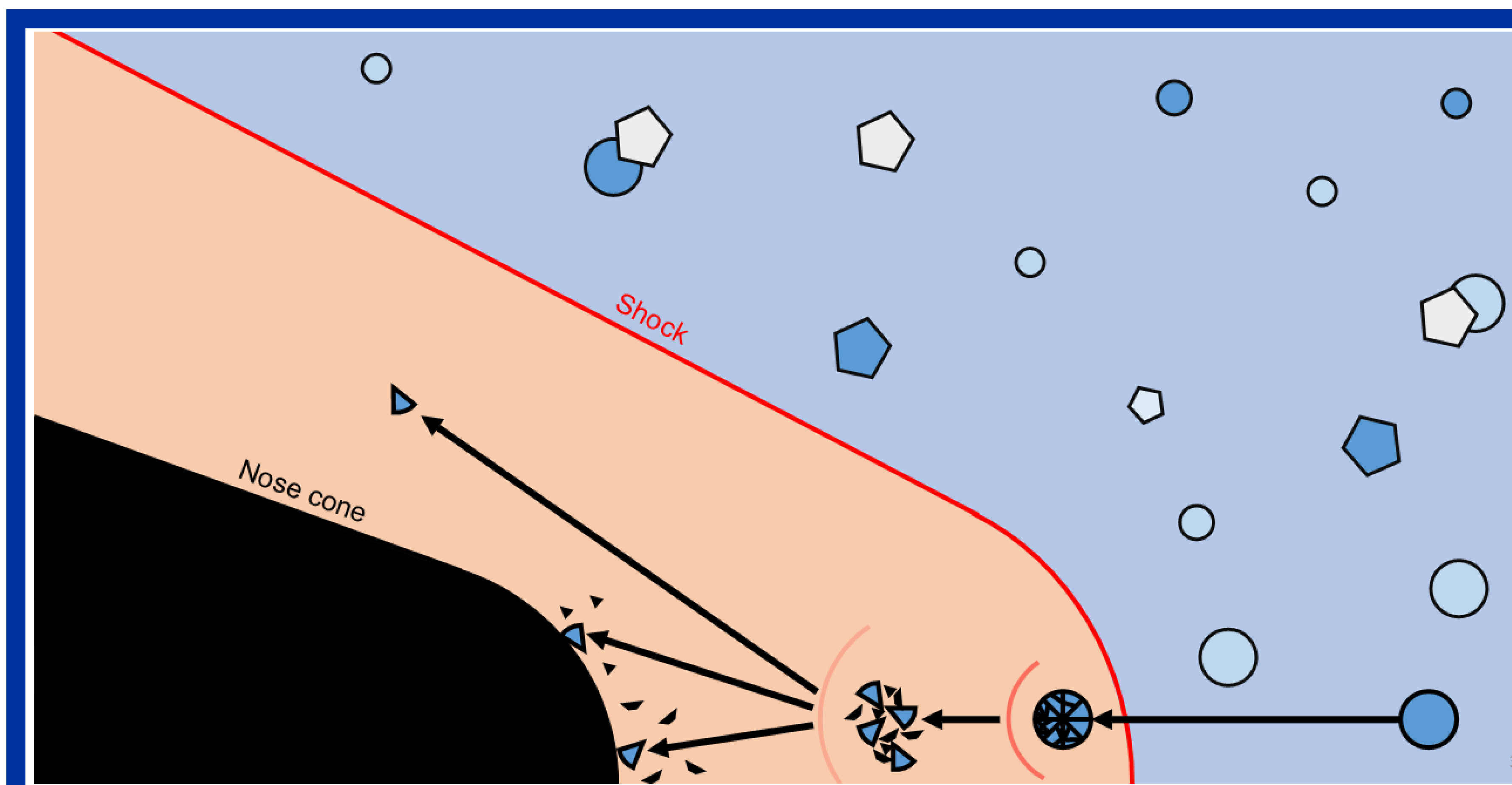
Introduction

Vehicles traveling at hypersonic velocities may pass through clouds in the atmosphere. These clouds, particularly cirrus clouds, lie between 15 and 80 km, contain ice particles and other dust conglomerates. The ice particles pose a potential danger to the vehicle possibly harming the heat shield or can also influence the aerodynamics of the vehicle.

The shock wave in front of the vehicle acts as a barrier between the resting ice particle and the high velocity vehicle. When the ice encounters the shock wave, it is subject to a high dynamic pressure and high temperature.

The high pressure from the shock impacting the ice particle may induce fracturing, and the high temperature as the shock passes over the ice particle, the ice particle and melt and evaporate.

An in-house coupled fluid-structure/thermal interaction solver has been developed to study these effects. In this work, a study was performed to determine the numerical parameters that ensure the convergence of unsteady flow as it passes over the ice particle.

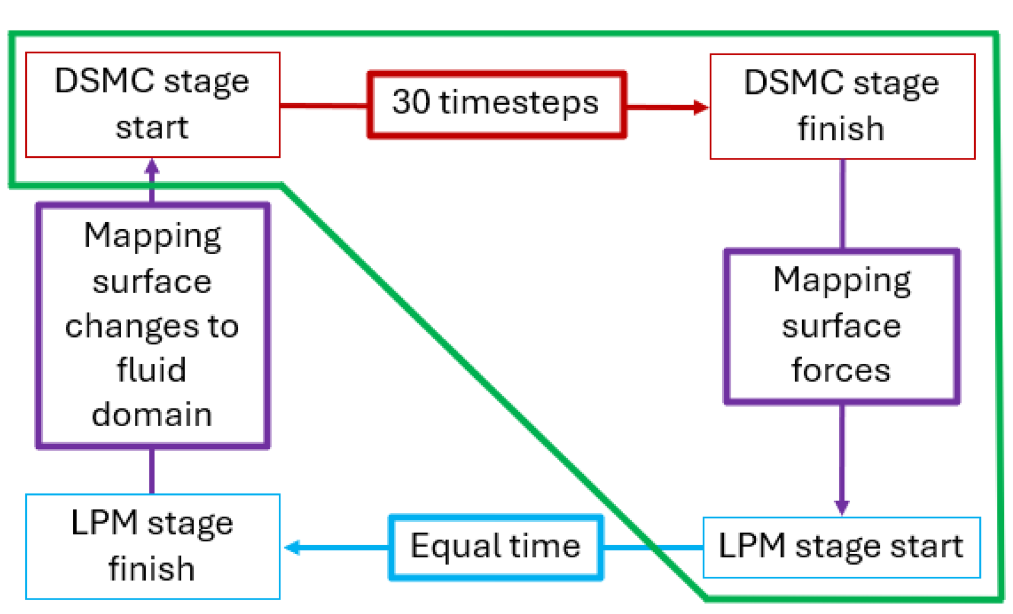


Cartoon of a hypersonic vehicle traveling through a cirrus cloud with the ice fracturing as the shock passes over them.

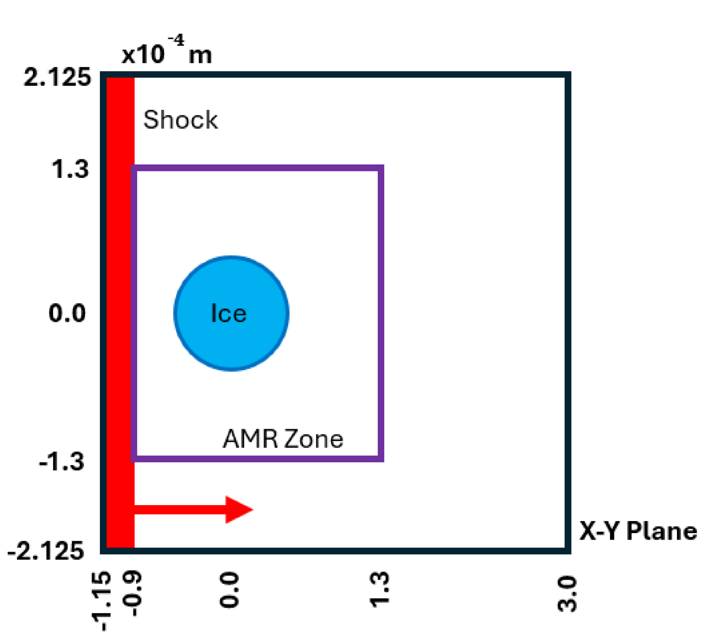
The Model

The fully coupled solver consists of the direct simulation Monte Carlo (DSMC) technique to simulate the fluid domain and the lattice particle method (LPM) to simulate the solid domain of the ice particle.

The coupling consists of four sequential steps. The DSMC simulations are performed for a sampling window, consisting of several time steps. The surface forces from the fluid domain are mapped to the structural boundaries of the ice. These forces are computed for a stage length in the LPM solver that is of equal time to the sampling window size. Finally, the structural state of the solid from LPM is looped back to the DSMC solver.



Coupling Scheme



2D view of the simulation setup

Methods

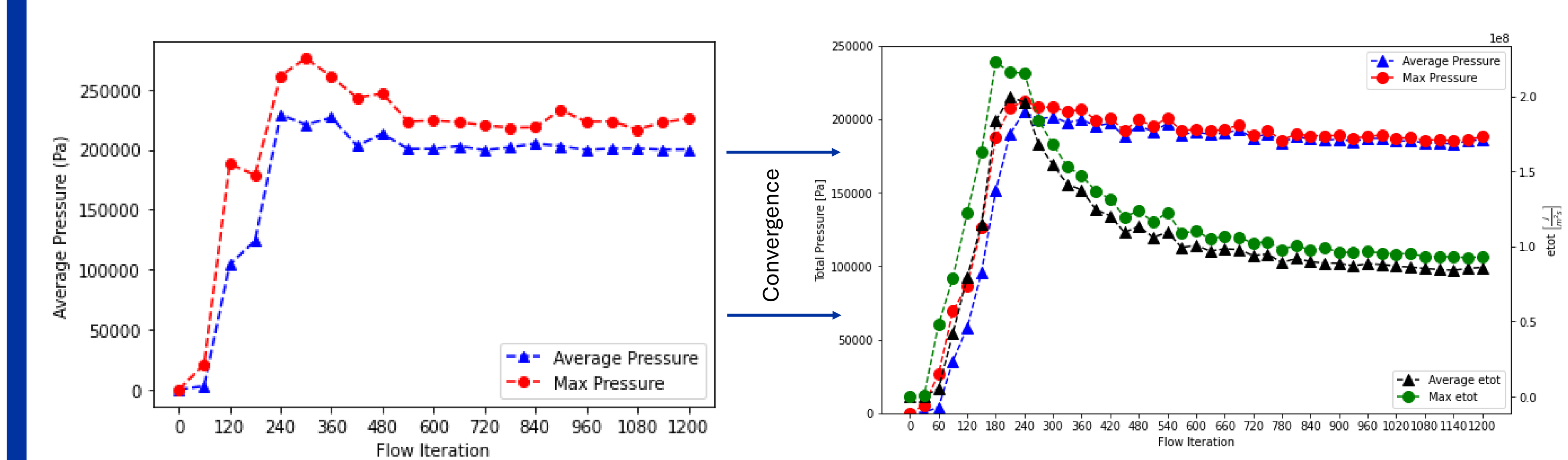
To ensure proper mapping of the forces to the structural boundaries is simulated, an adaptive mesh refinement (AMR) region was created. The cells in the AMR are refined in the shock close to the ice particle in the fluid domain.

For the study, a Mach 6 ground level extreme case is simulated to create refinement criteria for all realistic Mach and altitude conditions. The DSMC numerical parameters simulated consist of number of simulated particles, number of cells refined in the flow direction, and number of off-direction cells refined for each loop. The refinement criteria was set as the local Knudsen number, being the mean free path divided by the cell length.

Number of simulator particles	AMR in-plane refinement	AMR out-of-plane refinement	Local Knudsen refinement	Window size
8.250 million	4	1	0.60	10
16.50 million	8	2	0.70	20
24.75 million	16		0.78	30
33.00 million			0.80	60
41.25 million			0.90	
66.00 million			1.00	
			1.10	

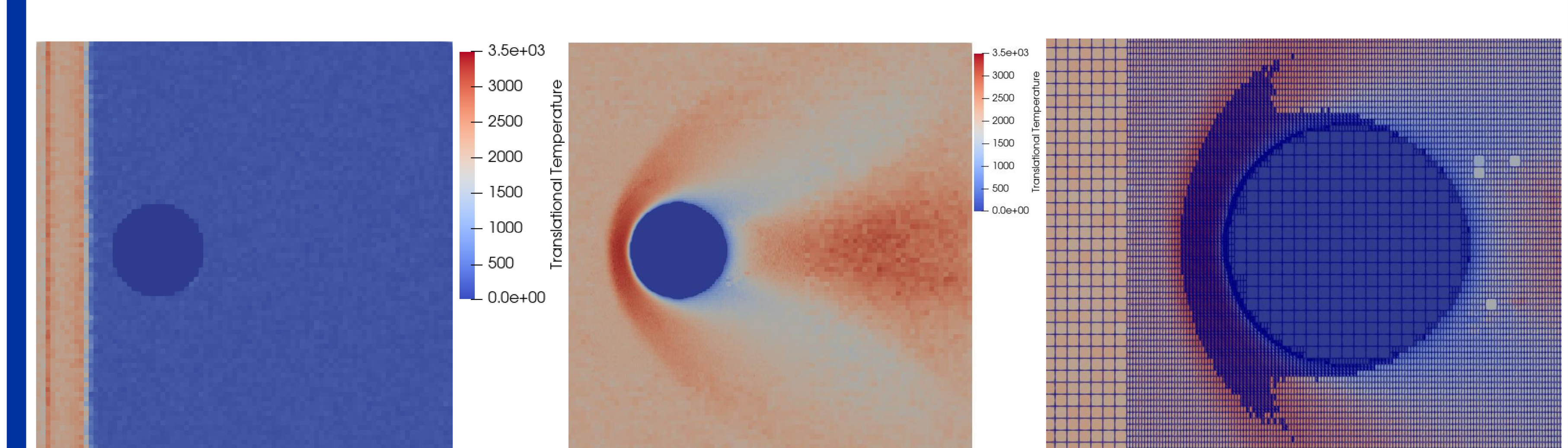
The different parameters were tested independently for a converged solution.

Results



Initial case - 8.25 million particles, 4 in-flow, 1 out-flow, 0.78 Kn, Window of 60

Converged case - 33 million particles, 4 in-flow, 2 out-flow, 0.78 Kn, Window of 30



Initial setup of the simulation. Structured mesh with equal cell sizes across the domain.

Converged simulation visual of the shock development around the ice particle

A zoomed image of the AMR region with a stronger refinement near the ice particle, and where the bow shock lies.

Conclusions

It was determined that the window size of 30 timesteps, 33 million simulation particles, an AMR criteria of a local Knudsen number of 0.78 yielding 4 in-plane refinements and 2 out-of-plane refinements was required for convergence of the simulations. The AMR region shows proper cell refinement for the shock development over the ice particle.

Future Work

With the DSMC convergence complete, the focus diverts to the LPM solver. Parameter testing and convergence on the LPM will be completed to ensure proper physical, structural, and fracturing changes occur prior to the validation testing of the fully coupled fluid-structure interaction solver.

Acknowledgements

This material is supported by the Office of the Under Secretary of Defense for Research and Engineering under award number FA9550-22-1-0342. This research was performed with the support of computational resources from the University of Kentucky Center for Computational Sciences.