Development of a Computational Framework to Investigate Thermochemistry of Melt Flow in Aerothermal Entry Physics

Tyler D. Stoffel and Savio J. Poovathingal
Department of Mechanical Engineering
University of Kentucky

International Planetary Probe Workshop
July 20, 2021

Objective

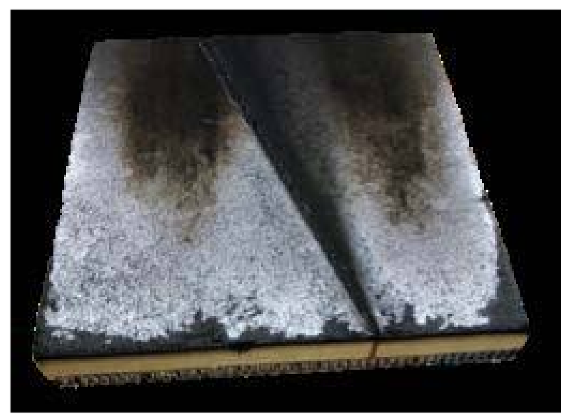
Develop a robust computational framework capable of simulating aerothermodynamic flows subject to the combination of:

- Hypersonic speeds
- High temperatures from shock-layer formation
- Interaction of gas and liquified solid materials

Introduction

During planetary entry, hypersonic vehicles are exposed to drastically high temperatures because of shock-layer formation during which surface temperatures exceed 1500 K. Managing such high-temperatures requires efficient thermal protection systems (TPS) and significant effort is placed into designing TPS materials. Designing a TPS is a major challenge because gas-surface interactions operate on many effects that take place on multiple time and length scales. One of these effects is the **melted flow** of the liquefied TPS material. A powerful tool in computational fluid dynamics (CFD) called the ghost-fluid method exists to resolve multi-phase flows, but has not yet been applied to hypersonic systems. An effort to combine this method with state of the art hypersonic CFD tools could increase the accuracy of crucial computational models for hypersonic flight. Although this work is developed for hypersonic simulations, the framework outlined here has applications in additive manufacturing, combustion, and plasma physics.

(a) Avcoat specimen showing formations from molten silica layers, before (left) and after (right) testing [1].



(b) A section of tested PICA TPS with NuSil surface coating [2].

Figure: Ground-test result images which depict melt flows.

Methods

A liquid-gas system can be properly defined by a level set function denoted as

$$\phi(\boldsymbol{x}, t) \begin{cases} > 0 & \text{if } \boldsymbol{x} \in \text{liquid phase} \\ = 0 & \text{if } \boldsymbol{x} \in \Gamma \\ < 0 & \text{if } \boldsymbol{x} \in \text{gas phase} \end{cases}$$

which is a smooth function of the coordinate vector $\boldsymbol{x}$ and time t, constrained by Γ, the gas-liquid interface [6]. The interface advects by solving

$$\frac{\partial \phi}{\partial t} + \boldsymbol{u} \cdot \nabla \phi = 0$$

Hence the system is defined for both liquid and gas at each node. The ghost-fluid method can then be used to resolve the system's behavior.

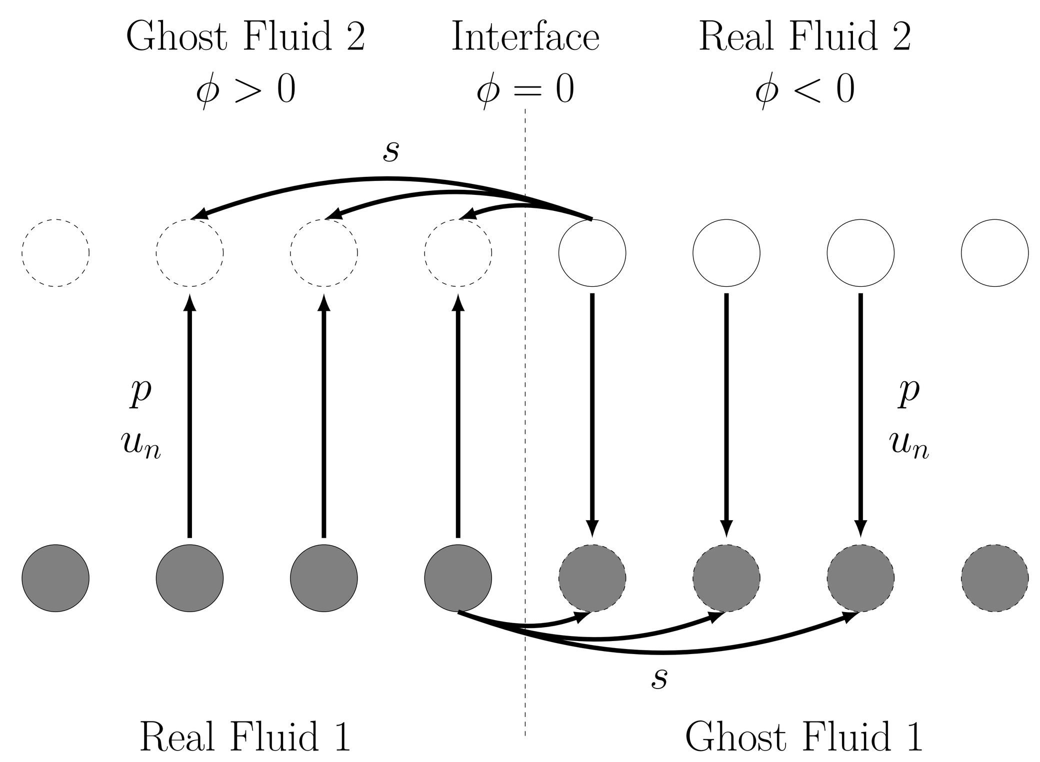


Figure: A CFD stencil depicting the ghost-fluid method [4].

Central Idea

Couple the **ghost-fluid method** with leading edge hypersonic **compressible and incompressible CFD** techniques to build a robust analysis tool for melt flows in entry physics.

Preliminary Results

A preliminary 1D computational experiment simulating an air/liquid drop system based on work from Ref. [3] is shown. A stationary liquid droplet of length $L = 0.2$ m is impinged upon by a shock traveling through surrounding air. The initial conditions are:

- Ambient pressure: $P_i = 98.06$ kPa
- Liquid center position: $x_{L,i} = 0.5$ m
- Liquid density: $\rho_L = 10$ kg/m^3
- Shock position: $x_{S,i} = 0.1$ m
- Pre-shock air density: $\rho_{G_1,i} = 1.583$ kg/m^3
- Post-shock air density: $\rho_{G_2,i} = 2.124$ kg/m^3
- Post-shock air velocity: $u_{G_2,i} = 89.98$ m/s
- Post-shock air pressure: $P_{G_2,i} = 147.4$ kPa

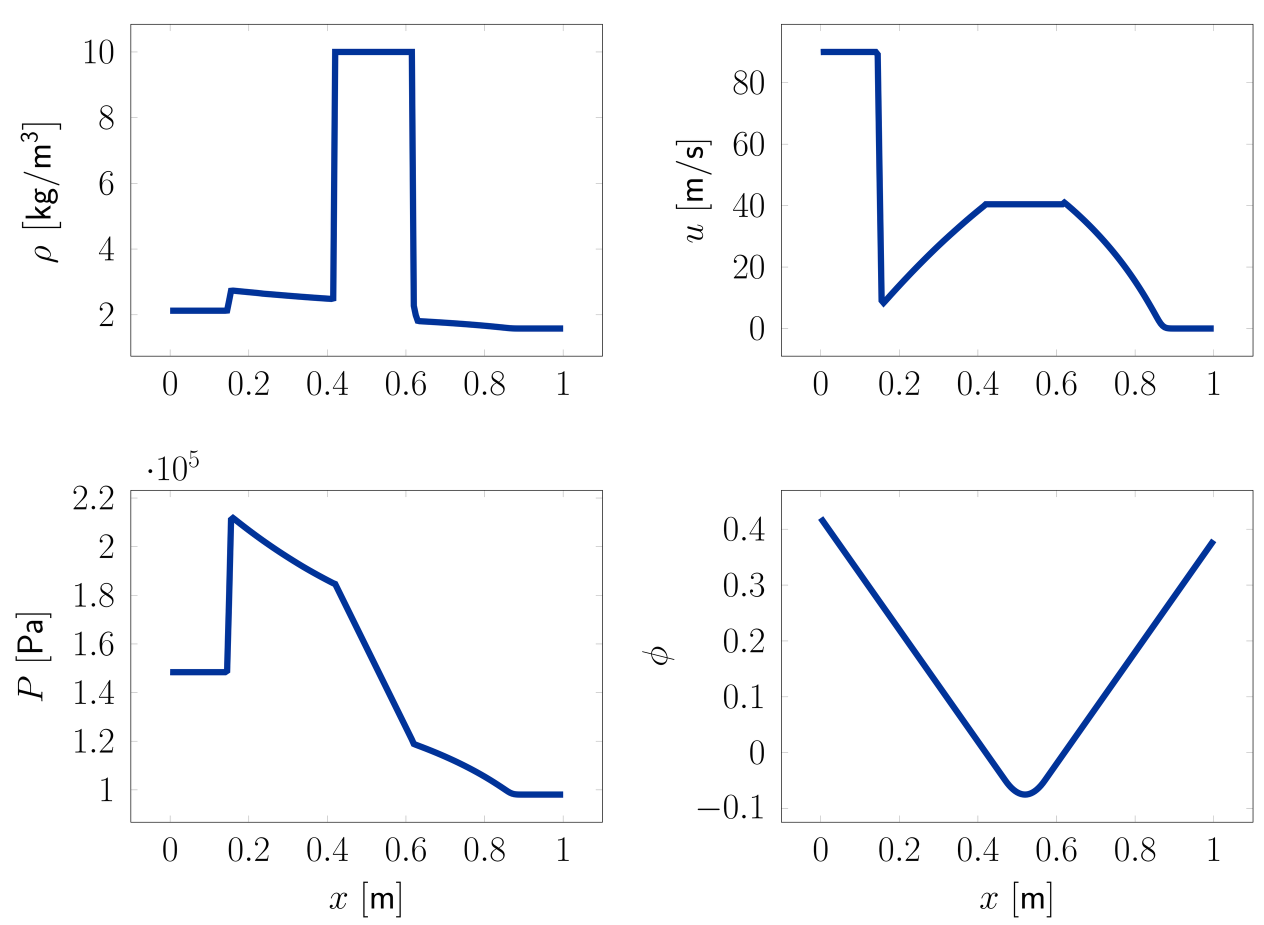


Figure: Density, velocity, pressure and level set function results from the air/liquid drop simulation at $t = 1.75 \times 10^{-3}$ s.

Continuing Work

Moving forward, this project aims to implement:

- Viscous effects, surface tension, and thermal conductivity by solving the full Navier-Stokes equations
- Nonequilibrium chemistry analysis
- Two-wave multiphase interface coupling

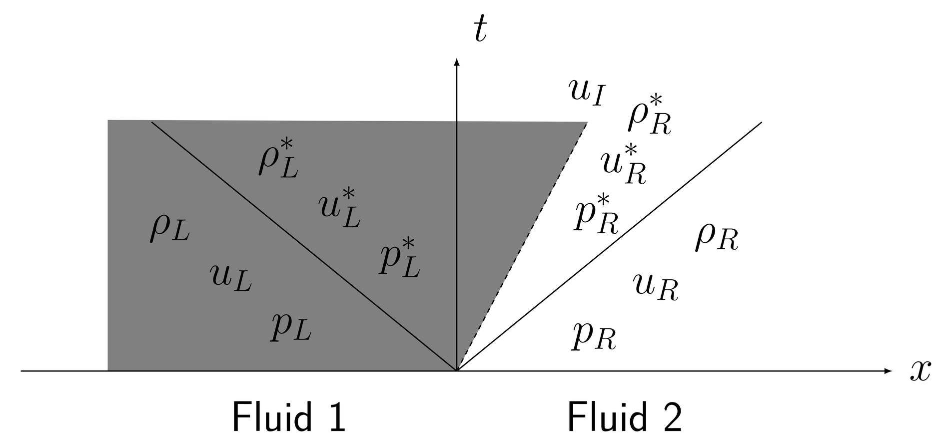


Figure: Wave diagram of the two-wave Riemann probelm for the ghost-fluid method [5].

References

[1] A. Alunni and T. Gokcen. *46th AIAA Thermophysics Conference.* (2016), p. 3534.
[2] M. D. Barnhardt et al. International Planetary Probe Workshop. (2018).
[3] R. Caiden et al. *Journal of Computational Physics* 166.1 (2001), pp. 1–27.
[4] R. P. Fedkiw et al. *Journal of computational physics* 152.2 (1999), pp. 457–492.
[5] R. W. Houim. 2011.
[6] S. Lin et al. *Journal of Computational Physics* 380 (2019), pp. 119–142.

Acknowledgements

- We would like to thank the University of Kentucky Center for Computational Sciences and Information Technology Services Research Computing for their support and use of the Lipscomb Compute Cluster and associated research computing resources.
- This material is based on work supported by NASA Kentucky under NASA award number: 80NSSC20M0047