Implementation of Surface Tension on a Reacting Flow Solver, PeleLM

Preprint

Hyoungwoo Kim,[1] Mahesh Natarajan,[2] Robert Chiodi,[3] Marc Day,[4] and Dong-hyuk Shin[1]

1 Korea Advanced Institute of Science and Technology
2 NASA Ames Research Center
3 Los Alamos National Laboratory
4 National Renewable Energy Laboratory

Presented at the 23rd Annual Meeting of the Institute for Liquid Atomization and Spray Systems (ILASS)-Asia 2023
Maison Glad Jeju, Korea
October 19-21, 2023

NREL is a national laboratory of the U.S. Department of Energy
Office of Energy Efficiency & Renewable Energy
Operated by the Alliance for Sustainable Energy, LLC

This report is available at no cost from the National Renewable Energy Laboratory (NREL) at www.nrel.gov/publications.

Contract No. DE-AC36-08GO28308

Conference Paper
NREL/CP-2C00-86728
November 2024

Implementation of Surface Tension on a Reacting Flow Solver, PeleLM

Preprint

Hyoungwoo Kim,[1] Mahesh Natarajan,[2] Robert Chiodi,[3] Marc Day,[4] and Dong-hyuk Shin[1]

1 Korea Advanced Institute of Science and Technology
2 NASA Ames Research Center
3 Los Alamos National Laboratory
4 National Renewable Energy Laboratory

Suggested Citation
Kim, Hyoungwoo, Mahesh Natarajan, Robert Chiodi, Marc Day, and Dong-hyuk Shin. 2024. *Implementation of Surface Tension on a Reacting Flow Solver, PeleLM: Preprint*. Golden, CO: National Renewable Energy Laboratory. NREL/CP-2C00-86728. https://www.nrel.gov/docs/fy25osti/86728.pdf.

NREL is a national laboratory of the U.S. Department of Energy
Office of Energy Efficiency & Renewable Energy
Operated by the Alliance for Sustainable Energy, LLC

This report is available at no cost from the National Renewable Energy Laboratory (NREL) at www.nrel.gov/publications.

Contract No. DE-AC36-08GO28308

Conference Paper
NREL/CP-2C00-86728
November 2024

National Renewable Energy Laboratory
15013 Denver West Parkway
Golden, CO 80401
303-275-3000 • www.nrel.gov

NOTICE

This work was authored in part by the National Renewable Energy Laboratory, operated by Alliance for Sustainable Energy, LLC, for the U.S. Department of Energy (DOE) under Contract No. DE-AC36-08GO28308. Funding provided by the U.S. Department of Energy Office of Science, Advanced Scientific Computing Research (ASCR). The views expressed herein do not necessarily represent the views of the DOE or the U.S. Government. The U.S. Government retains and the publisher, by accepting the article for publication, acknowledges that the U.S. Government retains a nonexclusive, paid-up, irrevocable, worldwide license to publish or reproduce the published form of this work, or allow others to do so, for U.S. Government purposes.

Cover Photos by Dennis Schroeder: (clockwise, left to right) NREL 51934, NREL 45897, NREL 42160, NREL 45891, NREL 48097, NREL 46526.

Implementation of Surface Tension on a Reacting Flow Solver, PeleLM

Hyoungwoo Kim[1], Mahesh Natarajan[2], Robert Chiodi[3], Marc Day[4], and Dong-hyuk Shin[1]*

[1] Korea Advanced Institute of Science and Technology, Daejeon, Republic of Korea
[2]NASA Ames Research Center, Mountain View, CA, USA
[3]Los Alamos National Laboratory, Los Alamos, NM, USA
[4]National Renewable Energy Laboratory, Golden, CO, USA

Corresponding author: Dong-hyuk Shin, Email: donghyuk.shin@kaist.ac.kr

Keywords: Multiphase, Volume of Fluid (VOF), Surface Tension, Height Function (HF), PeleLM

1. Introduction

In liquid rocket engines, the fuel is supplied to the combustion chamber in the liquid state though injectors. Such fuel undergoes atomization, vaporization, and combustion processes. To design reliable and efficient injectors, it is required to understand the full processes. This research is part of an effort to develop a full atomization-vaporization-combustion solver from first principles. As an initial step to tackle the atomization process, a multiphase flow solver is under development. For the development, a library of the volume of fluid scheme for multiphase, IRL [1-2], is coupled with a reacting Navier-Stokes equation solver, PeleLM [3]. Furthermore, as the surface tension has considerable effects on spray breakup. surface tension is implemented in the momentum equation using the continuum surface force model (CSF) [4] and the improved height function (HF) technique [5].

2. Numerical Methods

In the coupled PeleLM-IRL solver, the scalar field α (volume fraction of fluid) is defined. If α is 0, there is no tracking fluid inside the cell, and if α is 1, the cell is full with the tracking fluid. Instead of solving the continuity equation of density, a flux of α is calculated with IRL in the coupled PeleLM-IRL solver. Then, the next time step α^{n+1} is updated with computed flux. The density is estimated by $\rho = \rho_l \alpha + \rho_g (1-\alpha)$, where the subscripts l and g denote liquid and gas, respectively.

In the CSF model [4], the surface tension volume force is calculated by $\vec{F} = \sigma \kappa \vec{n}$, where σ is the surface tension coefficient, κ is the curvature which is calculated by the improved HF technique [5], and $\vec{n}$ is the interface normal vector which is obtained from the gradient of α. Finally, the surface tension is introduced in the momentum equation by the external force term.

3. Verification Results

To verify the implementation of surface tension, two test cases were conducted. In both test cases, the liquid density is 1,000 kg/m^3 and the gas density is 1 kg/m^3. The dynamic viscosity coefficient is set to 0.01 N·s/m^2 and the surface tension coefficient is 0.07 N/m.

3.1 Verification 1: Stationary circular droplet

The objective of the test case is to confirm that the surface tension implementation leads to the correct pressure difference inside a circular droplet. As shown in Fig. 1a, a circular droplet of the radius of 1 mm is placed on the simulation domain as an initial condition, then unsteady simulations were conducted on three different grid resolutions (Δx=Δy=23, 47, and 94 μm).

Fig. 1b shows the resulting pressure distribution along the centerline on the fine grid. The pressure inside the circular droplet is fairly uniform, and the average pressure is 66, 67,1 and 67.5 Pa for the coarse, medium, and fine grids respectively. Considering the theoretical value of 70 Pa, the fine grid solution provides a small error of 3.6 %.

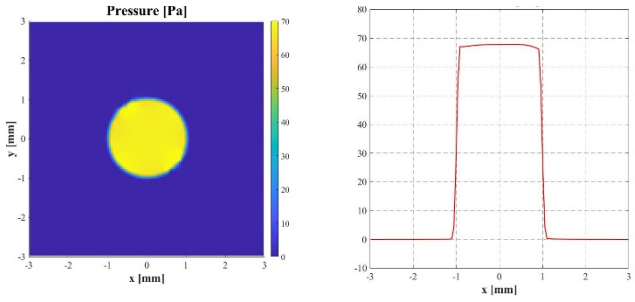

Fig. 1. The result of the stationary droplet test: (a) The pressure field. (b) The pressure along the y = 0 line.

3.2 Verification 2: Oscillating elliptical droplet

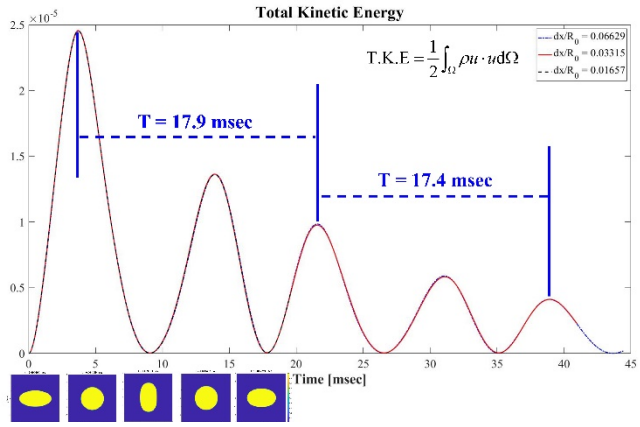

Fig. 2. The total kinetic energy of oscillating droplet test case with various grid sizes.

An elliptical droplet is given by $x^2/4 + y^2 \leq 1$, whose units are in mm. Fig. 2 shows the evolution of the total kinetic energy (T.K.E). Rayleigh [6] derived the analytic solution for the oscillation period when the amplitude is small. The analytical solution value is 16.3 msec, while the first and second periods are 17.9 msec and 17.4 msec, respectively from the simulation. The differences (9.8% and 6.7%) are decreasing over time as the amplitude becomes smaller.

4. Conclusion

The surface tension is implemented in the coupled PeleLM-IRL solver using the continuum surface force model and the improved height function technique.

Two validation cases were performed. Firstly, a stationary circular droplet test was conducted. The resulting pressure difference inside the droplet is similar to the theoretical value with a 3.6 % difference. Secondly, an oscillating elliptical droplet test was conducted. Total kinetic energy (T.K.E) is used to quantify the oscillating period. The amplitude of T.K.E decreases due to viscosity and the period is similar to the analytical solution [6] with a 9.8 % error.

In future work, the liquid jet atomization will be simulated using the developed solver. Furthermore, vaporization physics will be implemented.

Acknowledgments

This work was supported by Korea Research Institute for defense Technology planning and advancement (KRIT) grant funded by the Korea government (DAPA(Defense Acquisition Program Administration)) (No. KRIT-CT-22-030, Reusable Unmanned Space Vehicle Research Center, 2023), Korea Institute of Energy Technology Evaluation and Planning(KETEP) grant funded by the Korea government(MOTIE)(20214000000310, Energy Innovation Research Center for Carbon-Neutral High-Efficiency Gas Turbine Combustion Technology), the Exascale Computing Project (17-SC-20-SC), a collaborative effort of the U.S. Department of Energy Office of Science and the National Nuclear Security Administration, and the US Department of Energy through the Los Alamos National Laboratory. Los Alamos National Laboratory is operated by Triad National Security, LLC, for the National Nuclear Security Administration of U.S. Department of Energy (Contract No. 89233218CNA000001).

5. Reference

1. Owkes, M., and Desjardins, O., “A computational framework for conservative, three-dimensional, unsplit, geometric transport with application to the volume-of-fluid (VOF) method,” Journal of Computational Physics, pp. 587-612, 2014.

2. Chiodi, R., and Desjardins, O., “General, robust, and efficient polyhedron intersection in the Interface Reconstruction Library,” Journal of Computational Physics, 110787, 2022.

3. Day, M. S., and Bell, J. B., “Numerical simulation of laminar reacting flows with complex chemistry,” Combustion Theory and Modelling, pp. 535-556, 2000.

4. Brackbill, J. U., Kothe, D. B., and Zemach, C., “A continuum method for modeling surface tension,” Journal of Computational Physics, pp. 335-354, 1992.

5. Lopez, J., Zanzi, C., Gomez, P., Zamora, R., Faura, F., and Hernandez, J., “An improved height function technique for computing interface curvature from volume fractions,” Computer Methods in Applied Mechanics and Engineering, pp. 2555-2564, 2009.

6. Rayleigh, L., “On the capillary phenomena of jets,” Proc. R. Soc. London, pp. 71-97, 1879.