CAV2021
11th International Symposium on Cavitation
May 10-13, 2021, Daejon, Korea

Shock-Raindrop Interaction and Potential for Cavitation-Induced Droplet Breakup

Caroline Anderson[1] and Michael Kinzel [1*]

[1]University of Central Florida; Mechanical and Aerospace Engineering

Abstract: Hypersonic vehicles entering the atmosphere have unique challenges including very high loads when interacting with a raindrop. Such loads drive a need to understand droplet breakup mechanisms. In this work, analyses and volume of fluid-based computational fluid dynamics are used to understand breakup mechanisms. At hypersonic speeds, the time scale is very fast with a short distance scale. This leads to conditions where there is not enough thermal energy or time to vaporize the droplet, shear stripping occurs to slow, leaving cavitation as the fastest physics with potential to drive high-speed droplet breakup. The results indicate that studies of rain impact at hypersonic speeds demand and understanding of cavitation and its role in droplet breakup mechanisms.

Keywords: incipient cavitation; hypersonic; droplet

1. Introduction

Rockets, missiles, and atmospheric entry vehicles of the present and near future are subject to weather at speeds in the hypersonic regime, and thus the effect of precipitation is warranted examination to evaluate safe flight conditions. The force which a rain droplet exerts on a body sees an increase that scales with the square of the impact speed (V_∞) and droplet diameter (D). Assuming the droplet does not breakup or slow down substantially after the shock, the impact force, F, peaks with an estimation of

$$F \ = \frac{\rho\pi}{6}\, V_\infty^2 D^2 \tag{1}$$

where ρ_l is the liquid density. In this limiting analysis, the vehicle is subjected to loads exceeding 2kN (Mach 6, 0.1cm raindrop) over a very small area ($\sim D^2$). This upper limit presents a design criteria for flight through rain and understanding the underlying droplet-breakup mechanisms will lead more accurate loads estimates. In this work, the potential role of cavitation as a driver of droplet breakup is evaluated.

Impacting a millimeter-scale droplet at hypersonic speeds leads to forces that can exceed common material yield stresses. Additionally, flight through rain also disrupts boundary layer character and the resulting heat transfer [1,2]. Within the shock layer, these changes alter the bow shock profile, and impacts effective aerodynamic heating and stability. Droplet-vehicle interaction occurs in a series of stages: (1) the droplet is in static atmospheric conditions, (2) the droplet encounters the leading bow shock and crosses into the flow field, and (3) undergoes break up to surface impact. This separation of droplet lifespan into stages serves as a template for the presented computational model. As the droplet crosses the shock wave it enters a sharp change in surrounding fluid conditions: an increase in external and internal pressure forces, surface shear, and in temperature. At high speed, the Reynolds number of the droplet exceed values of 100,000. Thus, the Weber number that compares inertial forces experienced by the droplet to the surface tension force of the droplet, approaches higher ranges. The Weber number ($We = \frac{\rho V^2 D}{\sigma}$) is defined to correlate breakup modes seen in experimentation, and provides insight to droplet shape prior to impact, enabling a prediction of load that will impinge on the vehicle surface. Here, ρ is the density of the liquid phase, V is the relative velocity between droplet and surrounding gas flow, and σ is surface tension coefficient. The conventional regimes suggest breakup in the form of sheet stripping and catastrophic [3], in which the droplet is seen to shatter before material can be removed by sheet stripping in Mach numbers of 3 and higher [4]. In this work, droplet breakup from sheet stripping, boiling, and cavitation is evaluated.

* Corresponding Author: Michael Kinzel, michael.kinzel@ucf.edu

2. Materials and Methods

The study numerically models the process of droplet behavior and breakup through its path from subsonic flow, through post shock flow field, to the vehicle. Modeling the impact of the droplet onto a body presents an issue of scaling. Vehicle bodies are often on the order of meters, and droplets on order of micrometers, creating a disparity that demands an impractical form of meshing and computation. The presented work solves this by breaking up the simulation into stages[5]:

(1) Vehicle scale simulation, in which the hypersonic flow around the body is solved (conservation of mass, momentum and energy of the gas and droplet phases). This effort utilizes high-speed numerics [6], along with high-speed models of one-way, droplet trajectories utilizing high-speed drag models [7] in an Eulerian context are utilized.
(2) The droplet trajectory is extracted and converted to reference frame of the droplet using a Galilean transformation similar to the approach outlined in Ref [8] for studying cavitation models on bubble trajectories.
(3) Using the results from Stage (2), one can focus a computational fluid dynamics (CFD) analysis on the droplet itself. Here, a high-speed, multiphase volume of fluids (VOF) solution at the droplet scale is utilized. The numerics and validation in the context of underwater explosions is presented in Ref [9].
(4) Lastly, cavitation dynamics assessments in the droplet itself are evaluated. Using droplet scale pressure data through time, Rayleigh Plesset (RP) equation [10] can be used to assess cavitation. In this work, the CFD pressure profile is applied to the RP to evaluate the time scale and volume change from incipient cavitation.

The CFD aspects (stages 1 and 3) are solved in the context of the commercial code, Star-CCM+[11]. Stages (2) and (4) are developed in the context of Matlab [12].

3. Results

For results, we present analyses of competing mechanisms (boiling), followed by validation of shock-liquid interactions, and an assessment of a $M = 6$ droplet interaction.

3.1. Evaluation of Breakup Regimes

Driven by a high temperature post a shock ($T \sim 2400K$ at $M = 6$), there is a strong potential for boiling. The ultimate question is, are boiling processes more important than cavitation processes? Hence, boiling is evaluated through two approaches. The first evaluates the energy available to boil a droplet, which is estimated as the energy within the droplet stream tube from the shock to the leading edge calculated within the flow solution field. In this work we find that there is only 0.66% of the energy need to drive boiling, based on the vaporization energy of the raindrop. Recall, hypersonic speeds involve shocks that are tightly attached to the body yielding short distances, hence, lack of energy available to boil the droplet prior to impact. Alternatively, we also evaluate the time scale associated with boiling compared to that associated with impact by evaluating the timescale via Sherwood Mass Transfer relation, approximated for droplets

* Corresponding Author: Michael Kinzel, michael.kinzel@ucf.edu

by the Frossling correlation[14]. Such an approach indicates that roughly 0.7% of the droplet can boil prior to impact. Such assessments imply that boiling is a longer timescale than droplet impact time.

3.2. CFD Validation

With previous validation of underwater explosions [9], we perform an additional validation in the context of a shock (Mach 2.4) in air striking a column of water [13]. Results are provided in Fig. 1. Parts (a)-(c) indicates the traveling shock with the density gradient (upper half) and the absolute pressure (lower half). Note in Part (c), the underpressure (-8bar) at the aft end of the water column. Such a feature is observed experimentally and yields potential for cavitation. Parts (d)-(e) indicate comparisons of pressure measurements. Note that the present CFD model does not account for cavitation, hence, may be one cause for the overshoots with respect to the measures. Gauging the overshoots, the droplet is re-evaluated with a gas bubble of size equivalent to airborne dust. Initiated form is seen Fig.2. The gas bubble collapses in the wake of the compression wave, and following the pressure drop of the rarefaction wave, expands. Noted is the local compression wave to the bubble curvature. The fluid solution is compared to the chosen cavitation model. Lastly, the one-way coupled assessments of the RP is evaluated in Part (f). This result indicates a strong likeliness for cavitation by bubble growth of a single particle seed of dust increasing volume up to 250% over a time period of 0.1 microseconds.

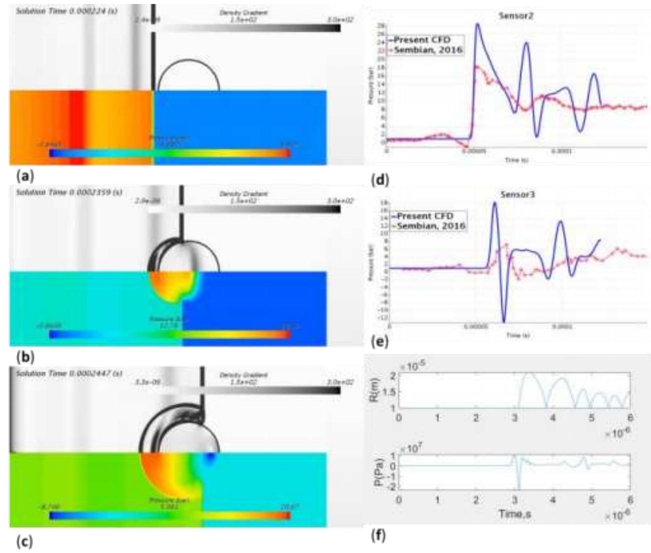

Figure 1. A sample solution of the prediction of the Sembian experiments [13]. In (a)-(c) are time evolutions where in (3) we can see a underpressure due to the shock reflection at the aft end of the cylinder. In (d) and (e) are comparisons to measured pressures. In (f) is the application of RP.

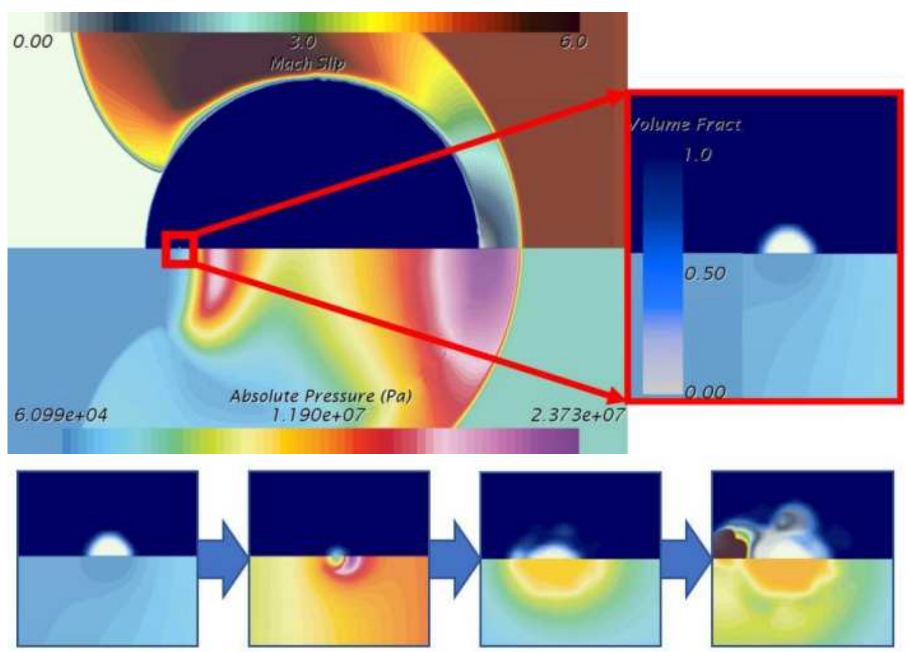

Figure 2. View of gas bubble seed within Droplet. Chronological response of gas bubble shrinkage and growth following interaction with internal pressure waves.

* Corresponding Author: Michael Kinzel, michael.kinzel@ucf.edu

However, the coupling and resulting droplet break up is not directly evaluated which demands coupled assessments.

3.3. CFD Evaluation of Cavitation

In order to evaluate the importance of cavitation over conventional droplet breakup modes (sheet stripping and catastrophic breakup), a prediction of a 1mm droplet impacting a leading edge of a wing traveling at Mach 6 is evaluated. Results are indicated in Fig. 3. Note that the droplet itself is near the standoff distance of the shock itself. As indicated in the plots, the shock persists through the water at roughly the same speed of the droplet. When reflected off the back of the droplet and concentrated by droplet curvature, the strong underpressure occurs following the rarefaction wave. This will drive cavitation, which occurs prior to any formation of sheet stripping or boiling. Such a results further indicates that cavitation phenomena is the fastest physical process to occur and likely to dominate loading when high-speed vehicles interact with raindrops. The measured underpressure is expected to increase with shock strength, increasing cavity growth and impact in droplet demise.

4. Conclusions

In the present effort, analyses of droplet breakup modes associated with high-speed vehicles are assessed. We evaluate the importance of boiling. It was found that the shock standoff distance is too short and that the impact times is too fast for boiling to readily occur prior to impact. Additionally, CFD is utilized to understand the importance of shear-driven breakup such as sheet stripping. It is found that these processes are also relatively slow. The potential for Cavitation conversely is quite strong. This is driven by a strong pressure wave in the droplet forming with the interaction of the shock. This wave travels through the droplet at roughly the sound speed in water (which is roughly equivalent to a Mach 5 shock in air). When the wave reflects from the back surface, cavitation is likely to occur and provides a potentially fast breakup mechanism.

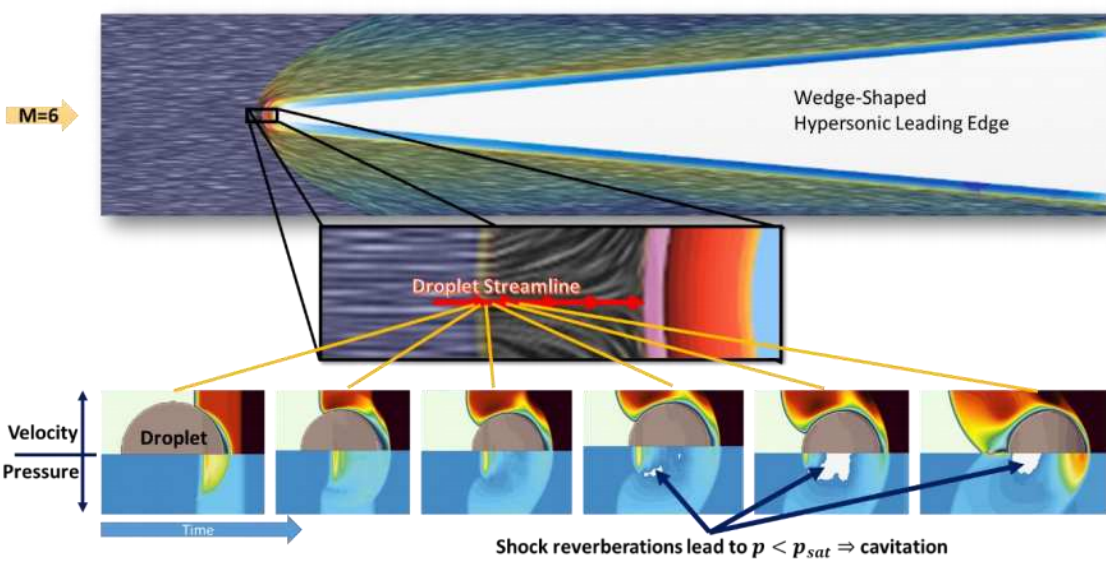

Figure 3. Flow and droplet trajectories over a leading edge at Mach 6 based on the DARPA MACH leading edge specifications. The lower plot indicates droplet-scale (Stage 3) calculations showing the potential for cavitation prior to any stripping mechanism.

* Corresponding Author: Michael Kinzel, michael.kinzel@ucf.edu

References

[1] Moylan, B. E. *Raindrop Demise in a High-Speed Projectile Flowfield*. The University of Alabama in Huntsville, 2010.

[2] Moylan, B., Landrum, B., and Russell, G. "Investigation of the Physical Phenomena Associated with Rain Impacts on Supersonic and Hypersonic Flight Vehicles." *Procedia Engineering*, Vol. 58, 2013, pp. 223–231. https://doi.org/https://doi.org/10.1016/j.proeng.2013.05.026.

[3] Pilch, M., and Erdman, C. A. "Use of Breakup Time Data and Velocity History Data to Predict the Maximum Size of Stable Fragments for Acceleration-Induced Breakup of a Liquid Drop." *International Journal of Multiphase Flow*, 1987. https://doi.org/10.1016/0301-9322(87)90063-2.

[4] Theofanous, T. G., Li, G. J., Dinh, T. N., and Chang, C.-H. "Aerobreakup in Disturbed Subsonic and Supersonic Flow Fields." *Journal of Fluid Mechanics*, Vol. 593, 2007, pp. 131–170. https://doi.org/DOI: 10.1017/S0022112007008853.

[5] Turner, J. E., Kinzel, M. P., and Cavainolo, B. A Multistage Approach To Computational Modeling of Large Droplet Breakup. In *AIAA Scitech 2020 Forum*, American Institute of Aeronautics and Astronautics, 2020.

[6] Liou, M. S. "A Sequel to AUSM: AUSM+." *Journal of Computational Physics*, Vol. 129, No. 2, 1996, pp. 364–382. https://doi.org/10.1006/jcph.1996.0256.

[7] Loth, E. "Compressibility and Rarefaction Effects on Drag of a Spherical Particle." *AIAA Journal*, Vol. 46, No. 9, 2008, pp. 2219–2228. https://doi.org/10.2514/1.28943.

[8] Kinzel, M. P., Lindau, J. W., and Kunz, R. F. An Evaluation of CFD Cavitation Models Using Streamline Data. 2018.

[9] Esplin, J. *Bulk Cavitation Extent Modeling: An Energy-Based Approach*. The Pennsylvania State University, 2016.

[10] Plesset, M. S., and Prosperetti, A. "Bubble Dynamics and Cavitation." *Annual Review of Fluid Mechanics*, 1977. https://doi.org/10.1146/annurev.fl.09.010177.001045.

[11] CD-Adapco, and Siemens. StarCCM+.

[12] Schreiber, R. "MATLAB." *Scholarpedia*, 2007. https://doi.org/10.4249/scholarpedia.2929.

[13] Sembian, S., Liverts, M., Tillmark, N., and Apazidis, N. "Plane Shock Wave Interaction with a Cylindrical Water Column." *Physics of Fluids*, 2016. https://doi.org/10.1063/1.4948274.

[14] Trontin, P., and Villedieu, P. "Heat and Mass Transfer Analogies for Evaporation Models at High Evaporation Rate." *Transactions of Japanese Society for Medical and Biological Engineering*, Vol. 51, No. SUPPL., 2013. doi:10.2514/6.2014-2062.Corporation, A. "Prepared for Space and Missile Interim Report." No. August, 1975.

* Corresponding Author: Michael Kinzel, michael.kinzel@ucf.edu