ADVANCES IN MULTISCALE PREDICTIONS OF TURBULENT SPRAY FORMATION AND IMPINGING JET ATOMIZATION

Project Description

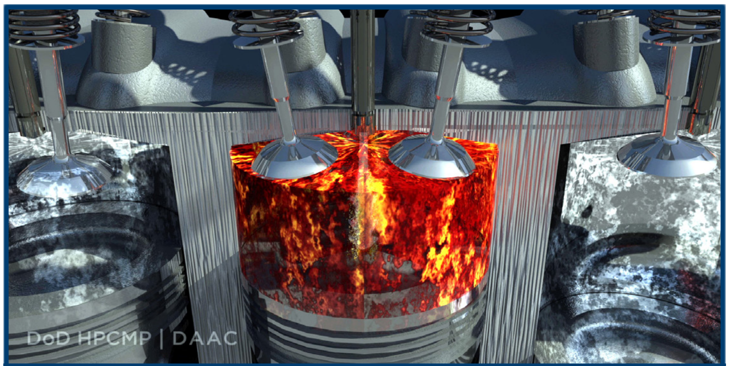
To support the Warfighter, the Army needs to provide reliable and efficient propulsion systems for heavy fuel engine platforms that exclusively rely on direct injection fuel delivery systems. Combat vehicles, such as the Gray Eagle MQ-1C (Figure 1) and the Joint Light Tactical Vehicle (JLTV), are powered by diesel engines running on military JP-8, or F-24 fuels. Technology breakthroughs in engine and fuel conversion efficiencies require a fundamental understanding of key governing phenomena including fuel/air mixture formation due to primary breakup and impinging jet atomization. Despite the relevance of the atomization processes, its modeling is still among the weakest part of practical engineering simulation tools. However, with the recent advances in supercomputing power and numerical algorithms, first-principle continuum simulations of spray atomization are emerging today as a viable tool to investigate the fuel spray and combustion behavior. It is the intent of this project to advance the fundamental tools to quantify and discover the underpinnings of difficult to measure phenomena, in regions including the dense spray and spray/wall dynamics.

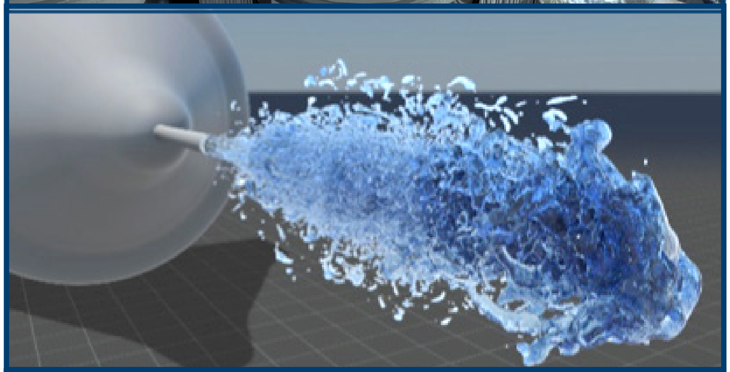
Relevance of Work to DOD

The major impact of this study is the ability to predict the microscale flow physics of atomizing and impinging jets at relevant engine conditions using fundamental principles. The Army can apply these predictive models to the performance of any combustion device that uses spray combustion, with the vetted models being particularly helpful in reducing the experimental steps necessary.

Computational Approach

The methodology for simulating spray primary breakup is based on the solution of the Navier Stokes system of equations coupled to a geometric unsplit interface-capturing method for immiscible fluids, Volume of Fluid. The simulation framework is designed for the distributed computations of unstructured mesh-based methods on HPC systems. The unstructured grids allow an accurate body-fitted representation of the complex curvature internal passageway regions, e.g., internal injectors. In simulating the physics of an impinging jet, a methodology based on smoothed particle hydrodynamics (SPH) is employed. In SPH, a field function (e.g., fluid property) is described by the integral representation method, which is





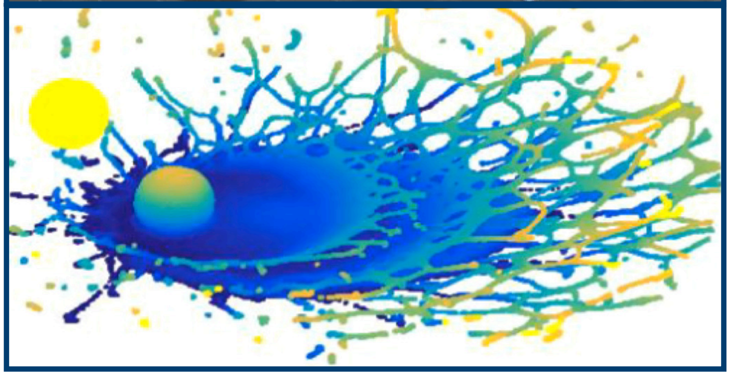


Figure 1: (top) Conceptual animation of Army internal combustion engine and fuel injection process.
Figure 2: (middle) Simulated image visualizing spray primary breakup phenomena from diesel injector.
Figure 3: (bottom) Simulated image describing the interactions of fuel droplets with piston wall.

reformulated based on the use of computational particles. In SPH, the drop, surrounding gas, and the solid wall are discretized into free-moving and/or fixed particles. As a result, it becomes straightforward to track drop deformation and the interface of the liquid and gas.

The governing equations to describe the fluid motion are discretized into the particle space, instead of grid space that is used in conventional computational fluid dynamics. As a result, SPH has the advantage of reproducing drop deformation and incorporating the drop properties and wall conditions easily.

The research was completed on the Cray-XE6 Garnet DSRC located at the U.S. Army Engineer Research and Development Center. Each simulation used 80 million cell nodes distributed over 5,000 processors and ran for 340 wall-clock hours.

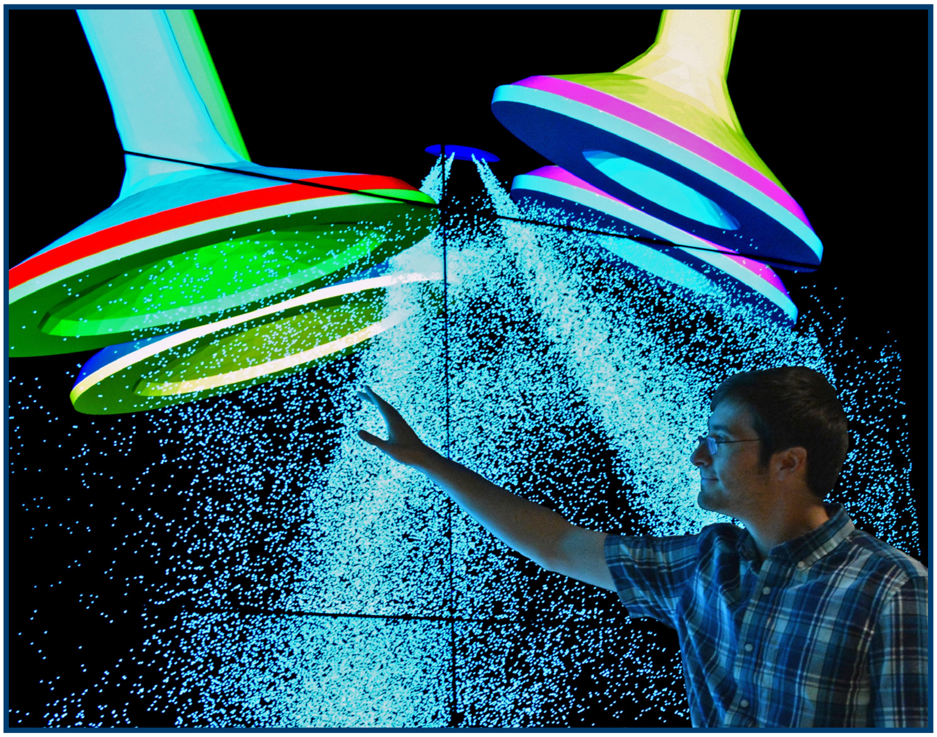
Results

In the simulation, the liquid fuel is ejected from a complex diesel injector geometry that accounts for internal features (Figure 2). The fuel pressure was specified at 150 bar imposing Reynolds and Weber numbers Re = 25,573 and We = 125,806. To specify diesel-type conditions, the chamber gas density is set to ρ = 22.8 kg/m3, by using 100% filled gaseous nitrogen at 303K, and a back-pressure at 20 bar. The simulation initializes with a liquid-filled injector and prescribes a rate-of-injection profile with bulk inflow velocities based from nozzle flow measurements. The simulations provide quantitative detailed information in the optically dense region, 0 < x/d < 30 jet diameters, seamlessly calculating droplet statistics, and fuel/air mixture formation processes. The results provide insights and new understanding of the breakup phenomena and droplet formation process for pulsed-injection diesel sprays.

Figure 3 shows a predicted image during a drop-wall interaction event. A series of diesel drops impact the piston surface at a 45-degree angle. The initial drop diameter is 100 micro meters with a velocity of 50 m/s. The leading drop impinges on the piston surface and creates a liquid film. The subsequent drops impact the film, causing the film to spread and generate liquid ligaments and secondary droplets. These ligaments can further form droplets as time progresses. In a combustion engine, the gas flow will alter the trajectory of fuel drops, ligaments, and secondary droplets. The high-temperature gas and wall in the combustion chamber will also cause the liquid drops and wall films to vaporize and create combustible mixtures. These phenomena require further investigation by coupling the present numerical method with advanced physical models.

Future

Future research efforts will extend the scalability of the tools to enable 1 billion element calculation of the jet atomization process. Further mechanisms will be investigated including nozzle effects, vaporization, compressibility, and cavitation. These research efforts are enabling breakthrough simulations that will provide discoveries and insights of the underpinning phenomena.



Contact Information

Dr. Luis Bravo
Vehicle Technology Directorate
Army Research Laboratory
luis.g.bravorobles.civ@mail.mil 410.278.9719

Co-Investigators

Professor Song-Charng Kong
Iowa State University