BOOK OF ABSTRACTS

34th European Conference on Liquid Atomization and Spray Systems

ILASS Europe 2026

6–9 September 2026
Lisbon, Portugal

ilass EUROPE

TÉCNICO UNIVERSIDADE DE LISBOA

1 2 9 0 UNIVERSIDADE D COIMBRA

UNIVERSIDADE BEIRA INTERIOR

BOOK OF
ABSTRACTS

34th European Conference
on Liquid Atomization and Spray Systems

ILASS Europe 2026

6–9 September 2026
Lisbon, Portugal

34th European Conference on
Liquid Atomization and Spray Systems

MONDAY

TUESDAY

WEDNESDAY

POSTERS

MONDAY

09:50 – 10:40

Atomization Fundamentals I *Chair: Antonio Lozano*
Impact of Rotor Downwash and Crosswind on Spray Plume Characteristics: A Laboratory Investigation Using Traversal Sampling
J.M.W. Vernon, S. Cooper, N. Emekwuru

Modeling Distortion and Breakup of Free Radial Liquid Films Using a Combined Thin-Film and Rim Model
A. Rouhi, A. Gyurkovich, P. Grimm, C. Mehring

Atomization: Applications I *Chair: Lucio Araneo*
Impact of apex angle of the gas nozzle in governing exit pressure and shock structures in gas atomization
S. Rai, S. Sarkar, S. Bakshi

Uniformization of discharge flow in a rotary atomizer using a cylindrical insert
Y. Seo, J. Lee

Artificial learning methods for sprays *Chair: Stéphane Zaleski*
Aerosol Prediction in Superheated and Flash-Boiling Sprays: Limits and Transferability of Machine-Learning Models
A. Saissi, E. Reardon, H. Hunt, S. Fitzgerald

Fourier positional encoding for high-fidelity Mie scattering neural surrogates
J. Lok, K. Bayik, J. Mei, R. Aviles-Espinosa, P. Birch, C. Crua

11:10 – 12:25

Atomization Fundamentals II *Chair: Carole Planchette*
Effect of Viscosity on the Bag Thickness in Aerodynamic Droplet Breakup
S. Seddiki, I.M. Jackiw

MONDAY

Ultra-High-Speed Investigation of Flash Boiling in Ethanol Droplet Chains Induced by Satellite Droplet Collisions
L. Eich, H. Schmitz, P. Augustin, W. Rohlfs, M. Reddemann

Correlation-Based Turbulent Weber Number and Taylor Reynolds Number in a Fan-Stirred Isotropic Chamber: Hinze's Relation and Breakup Regimes
V.C. Coşar, Y. Simeka, Ö. Ertunç

Experimental methods for droplets & sprays I *Chair: Steven Begg*
Optical Measurement of Liquid Film Thickness in Swirl Atomizers with Correction for Refractive Distortion
K. Anbarcı, M.A. Uzun, Ö. Ertunç

First Characterisation of a typical Aero-engine pre-filmer atomizer through Surface Curvature Analysis
A. Sekar, S. Idlahcen, S.R. Harth, J. Reveillon, B. Duret,
J.-B. Blaisot, F.-X. Demoulin

Flow Regime Analysis of Slit Nozzle Operating Conditions under Vacuum Using CFD and Optical Diagnostics for Powder Aerosol Deposition
M. Ghodbene, H. Auer, C. Franzke, J. Maaß, R. Moos

Numerical methods for sprays I *Chair: Michele Battistoni*
LES investigation of the VOF-LPT coupling method for Dual Gas-Assisted Sheet Atomizers
J. Roeb, O.T. Stein

Holistic Simulation of Laser-Induced Ignition under Flashing Conditions of an Experimental Rocket Combustion Chamber
C. Sessler, J.W. Gärtner, A. Kronenburg, A. Bee, J. Hardi, M. Börner

Large Eddy Simulation of Thixotropic Jets: A priori Analysis of Novel Subgrid-Scale Terms
M. Abdelsayed, M. Klein

MONDAY

14:00 – 15:15

Droplet-wall interaction I *Chair: Michel Gradeck*

On the Non-Deterministic Fingering Event in Single-Drop Impact on Dry Surfaces
M.R. Clemente, M.O. Panão

Role of wettability on coupling between droplet dymamics and boiling during drop impact on heated surface
O. Caballina, T. Potaufeux, A.C. García Wong, T. Czerwiec, G. Castanet

Experimental methods for droplets & sprays II *Chair: Edouard Berrocal*

Near-Field Particle Image Analysis of Liquid Ammonia Sprays from a GDI Injector
Y.-R. Chen, A. Hayakawa, U. Sonawane, C. Hespel, C. Mounaïm-Rousselle

Effect of fluid property on the atomization of perturbed liquid jets
K.K. Yadav, A. Saurabh

Phase-Resolved Imaging of Aperture Plate Oscillation and Jet Timing in a Vibrating Mesh Nebulizer
H. Schmitz, T. Tietz, P.W. Augustin, L. Eich, W. Rohlfs, A.L. Thiebes, C. Cornelissen, M.A. Reddemann

Numerical methods for sprays II *Chair: André Silva*

Validation of the Primary-Secondary Breakup Transition in Close-Coupled Gas Atomization of Molten Metals
S. van Laarhoven, N. Deen, J. Remmer, L. Achelis, U. Fritsching, A. Geppert, G. Finotello

Coupled VOF-LPT Approach to Investigate the Effects of Flow and Nozzle Parameters on Spray Characteristics
M. Piras, G. Sorrentino, R. Ragucci

CFD simulation of automotive multi-hole injector ammonia flow
C. Chasos

15:45 – 17:00

Interfacial and multiphase physics I *Chair: Guillaume de Sercey*
Optical investigation of effect of L/D ratio on transcritical cryogenic sprays
F. Murphy-Hunt, C. Marlow-Eastwood, M. White, C. Crua

Investigation of flash-boiling processes in cryogenic sprays
C.B. Marlow-Eastwood, F. Murphy-Hunt, Y. Liu, K. Vogiatzaki, Y. Shentu, C. Crua

Unified Modeling of In-Nozzle Flow and Aerodynamic–Thermodynamic Eulerian-Lagrangian Breakup for Predicting Flash Boiling Spray Behavior
M. Fagade, F.J. Arguelles, S. King, X. Fang

Spray Dynamics I *Chair: Jan Jedelsky*
A similarity analysis of cryogenic spray characteristics under sub-critical pressure
Y. Liu, J. Li, C.B. Marlow-Eastwood, C. Crua, Y. Shentu, K. Vogiatzaki

Measurement and Modelling of Equivalence Ratio Distribution in Narrow-cone-angle Hollow-cone Hydrogen Jets
G. Bae, J. Lim, Y. Ki, H. Yang, C. Bae, S. Moon

Similarity Study on Turbulence behaviour for Cryogenic Hydrogen Jets Across Thermodynamic Regimes
J. Li, G. Tretola, Y. Liu, K. Vogiatzaki

Experimental And Computational Investigation Of Turbulent Mixing In Under-Expanded Hydrogen Jets For Gaseous Propulsion Applications
F. Duronio, A. Montanaro, A. Di Mascio, A. De Vita

MONDAY

Numerical methods for sprays III *Chair: Anne Geppert*

Spray Cooling with Relative Nozzle–Substrate Motion: A Physics-Based Framework for Transient and Intermittent Heat Transfer
N. Banerjee, M. Marengo, C. Tropea

Numerical simulation of dynamic spray painting on plastic substrate with electrostatically supported high-speed rotary bell
Q. Ye, S. Popp, H. Dubbe, O. Tiedje

Numerical Study of Gas Injection Rate Effects on Viscous Fuel Atomization Using the Volume-of-Fluid Method
A. Marzooq, P.-H. Chen, H. Im

Impact of Rotor Downwash and Crosswind on Spray Plume Characteristics: A Laboratory Investigation Using Traversal Sampling

James M W Vernon, Simon Cooper, Nwabueze Emekwuru*

Engineering Department, Harper Adams University, Newport, TF10 8NB, UK.
*Corresponding author: eemekwuru@harper-adams.ac.uk

Abstract

This study investigates the interaction between drone rotor downwash and spray plumes under varying crosswind conditions to assess their impact on droplet Volume Median Diameter (VMD). A custom-designed laboratory test rig was constructed, featuring a motorized gantry to traverse a single rotor and a hollow-cone hydraulic nozzle at a constant velocity of 0.05 m/s across the focal point of a laser-based droplet sizer. This traversal approach was specifically chosen to allow for a more representative cross-sectional sampling of the spray plume, as typically obtainable in drone sprays, compared to traditional static point measurements, which often suffer from undercounting due to airflow displacement. The research examines how the highly three-dimensional and unsteady rotor wake affects atomization and droplet transport in a controlled environment. Unlike previous studies that focused on flat-fan nozzles, this work evaluates hollow-cone patterns typical of commercial drones. Furthermore, the implementation of a gantry-based traversal system provides a more accurate assessment of droplet populations within the turbulent wake. Statistical analysis using unpaired, two-tailed t-tests ($\alpha = 0.025$) confirmed that rotor downwash significantly impacts VMD across all high crosswind speeds ($p < 0.006$ for most droplet sizes). For very fine droplets in high crosswind, the rotor narrowed the droplet size distribution, reducing the standard deviation from 8.79 to 3.25 microns. Across different spray qualities, the rotor generally decreased average droplet size, with observed spans of variation between rotor-on and rotor-off states ranging from 23.4 to 26.6 microns. These findings suggest that the rotor's aerodynamic influence can stabilize the spray plume and enhance consistency, potentially mitigating environmental drift by counteracting crosswind effects.

Keywords: Drone spray drift, rotor downwash, droplet VMD, crosswind, traversal sampling

Modeling Distortion and Breakup of Free Radial Liquid Films Using a Combined Thin-Film and Rim Model

Alireza Rouhi*[1], Alexander Gyurkovich[1], Pierre Grimm[1], Carsten Mehring[1]

[1]Institute of Mechanical Process Engineering, University of Stuttgart, Germany
*Corresponding author: Alireza.Rouhi@imvt.uni-stuttgart.de

Abstract
The dynamics of inviscid, radially expanding axisymmetric thin liquid films of finite length are investigated. The nonlinear thin-film model proposed by Gyurkovich et al. [1] and limited to the simulation of semi-infinitely long films in passive ambient gas, is extended to films of finite-length (i.e., finite radial extend) by including a rim model with one-way coupling between the thin film and the rim. The dynamics of the rim and thus the time-varying position of the film edge is calculated by a momentum and mass balance of the rim. The extended model is applied to radially expanding films subject to sinuous (asymmetric) forcing at its origin. Similar to planar and swirling annular/conical films, nonlinear coupling between the sinuous and varicose mode exists. This coupling eventually leads to local film rupture upon which the thin-film simulation terminates in the absence of a rim model. The newly implemented simplified rim model allows to simulate film dynamics beyond this breakup point including also successive ligament shedding and the prediction of the radial film edge position over time. Predicted film lengths are found to be in good agreement with experimental data for low forcing amplitudes. At high disturbance amplitudes the predicted values for film breakup length are significantly influenced by the chosen discretization scheme. In addition, simulations suffer from numerical stability problems in this case.
Keywords: Thin film, radial, inviscid, lubrication theory, rim dynamics, modulation, sinuous

Impact of apex angle of the gas nozzle in governing exit pressure and shock structures in gas atomization

SumitRai[1],SupriyaSarkar[2],ShamitBakshi*[1]

[1]DepartmentofMechanicalEngineering,IITMadras,Chennai600036,India
[2]AlleimaIndiaPvt.Ltd.,Pune411012,India
*Correspondingauthor:shamit@iitm.ac.in

Abstract
Gas-assisted atomization is a widely trusted technique for producing metal powders, as it offers better control over powder characteristics compared to conventional techniques, which is essential for applications such as additive manufacturing, metal injection molding, and spray coating. This process entails the interaction of supersonic inert gas jets with a gravity-driven liquid metal stream falling from a crucible, resulting in melt disintegration and subsequent solidification into metal powder. The apex angle of the gas nozzle is a key geometric parameter influencing the gas flow passage and dictates the position of the focal point. Modification in the apex angle leads to a change in the pressurization condition at the exit of the metal delivery tube (MDT) as well as the structure of shock diamonds downstream of the flow. The present study experimentally compares the influence of semi-apex angles of 18◦ , 12◦ , and 8◦ on the pressure at the tip of MDT for flush position, by maintaining the same exit Mach number in a closed-couple configuration of converging-diverging gas nozzle, for the range of a gas (nitrogen) flow rate of 350–1000 Nm3 /hr. In addition, Schlieren imaging is employed to visualize the shock structures at different apex angles and flow rates. Experimental results indicate that decreasing the apex angle enlarges the aspiration pressure window at the tip of the MDT. Qualitative analysis of the obtained Schlieren images reveals that the shock diamond structure extends farther downstream as the apex angle decreases.
Keywords: gas-atomization, apex angle, metal powder

Uniformization of discharge flow in a rotary atomizer using a cylindrical insert

Yechan Seo[1], Jeekeun Lee*[2]

[1]Department of Mechanical Aerospace Electrical Convergence Engineering, Graduate School
of Jeonbuk National University, Republic of Korea
[2]Department of Mechanical Aerospace Electrical Convergence Engineering, Jeonbuk National University, Republic of Korea
*Corresponding author: leejk@jbnu.ac.kr

Abstract

This study was conducted to address the non-uniform distribution of the discharge flow rate in a rotary atomizer under conditions where the rotational axis is perpendicular to gravity. A cylindrical insert was installed in the fuel supply path of a rotary atomizer, and the fuel distribution characteristics were compared as a function of the insert diameter ratio (α). The fuel distribution within and outside of the rotary atomizer was visualized using a high-speed camera, and droplet size was measured using a phase Doppler anemometry (PDA). An experimental device was constructed to measure the discharge flow rate of a rotary atomizer, and the discharge flow rate was compared at different nozzle orifice positions. Without the cylindrical insert, an arc-shaped wave (ASW) formed on the nozzle cavity wall due to gravity. A local reservoir formed as a result of the ASW, leading to a non-uniform discharge flow rate distribution and deviations in droplet size depending on the orifice position. However, as the insert was installed in the nozzle and α increased, the deviation in the discharge flow rate decreased. Under the condition of $\alpha > 0.5$, the deviation of the discharge flow rate was < 25 %. Swirl vanes on the cylindrical insert further enhanced this effect, but the swirl vane in the opposite direction of the nozzle rotation hindered improvement in fuel distribution.

Keywords: Rotary Atomizer, Cylindrical Insert, Flow Rate Distribution, Spray Characteristics

Aerosol Prediction in Superheated and Flash-Boiling Sprays: Limits and Transferability of Machine-Learning Models

AymaneSaissi*[1],EdmundReardon[1],HughHunt[1],ShaunFitzgerald[1]

[1]CentreforClimateRepair,DepartmentofEngineering,UniversityofCambridge,United Kingdom
*Correspondingauthor:as3814@cam.ac.uk

Abstract
Predicting aerosol size distributions from superheated and flash-boiling sprays is challenging because droplet formation depends on coupled heat transfer, fluid dynamics, and phase change. The reliability of machine learning models under extrapolation to new experiments remains uncertain. This work presents a physics-informed machine-learning framework for predicting aerosol properties from superheated saline sprays that dry into NaCl particles. The predicted quantities are the geometric standard deviation (GSD), the sub-micron volume fraction (VolFrac), the number fraction between 30 and 500 nm, and the thermal energy to generate 1015 particles (Power_1015). To avoid information leakage, test data are never used for feature selection, model tuning, or validation. Performance is evaluated under progressively harder protocols: within-experiment generalization (one experiment partitioned by operating condition), pooled generalization (two experiments combined and randomly partitioned), and cross-experiment generalization (one experiment excluded entirely from training). Within a single experiment, models achieve R2 of +0.75 ± 0.11 for VolFrac and +0.77 ± 0.07 for Power_1015 . When pooled, performance remains positive for VolFrac (+0.70 ± 0.16), Power_1015 (+0.64 ± 0.22), and GSD (+0.20 ± 0.50). However, when trained on one experiment and tested on another, R2 becomes negative for most targets, meaning the models predict worse than the test-set mean. Only VolFrac retains partial transferability in one direction (R2 = +0.55 ± 0.21). Physics-informed features reduce the severity of this failure for VolFrac (−0.34 vs. −4.01 with raw features) and Power_1015 (−1.12 vs. −5.83), but do not resolve it. Standard within-experiment evaluation substantially overstates predictive capability. Cross-experiment benchmarks are essential for reliable deployment of spray machine learning models.
Keywords: flash-boiling spray, machine learning, generalization, dataset shift, aerosol prediction

Fourier positional encoding for high-fidelity Mie scattering neural surrogates

James Lok*[1], Kemal Bayik[1], Jiajun Mei[1], Rodrigo Aviles-Espinosa [1], Phil Birch[1], Cyril Crua[1]

[1] School of Engineering and Informatics, University of Sussex, Brighton, United Kingdom
*Corresponding author: f.lok@sussex.ac.uk

Abstract
Characterising droplet diameter and refractive index in sprays is essential for quantitative diagnostics in multiphase flows. Rainbow refractometry enables non-contact retrieval of these properties from optical scattering patterns, but accurate interpretation typically requires Lorenz–Mie theory. Although physically rigorous, Lorenz–Mie calculations are computationally expensive, requiring the use of simplistic approximations when solving the inversion of typical droplet populations.

Machine-learning surrogates could significantly speed-up scattering computations while providing near-Lorenz-Mie accuracy, but standard multi-layer perceptrons often struggle to reproduce the full Mie-scattering signal. Due to spectral bias, they tend to learn the smooth Airy-like envelope while under-resolving the high-frequency ripples that encode precise droplet information. To overcome this limitation, we propose a Fourier Positional Encoding Multi-Layer Perceptron, FPE-MLP, as a high-fidelity surrogate for Lorenz–Mie scattering in rainbow refractometry.

The model introduces a deterministic Fourier positional encoding layer before the MLP, enabling the network to represent fine ripple structures in the scattering signal. It was trained on 242,121 synthetic Lorenz–Mie rainbows spanning diameters of 80 to 200µm and refractive indices of 1.3–1.5. Compared with a standard MLP, the FPE-MLP reduced mean squared error by more than a factor of 12 and achieved an average R^2 of 0.993. In the test set, 95.1% of individual intensity predictions were within 5% absolute error, while inference was approximately four times faster than the exact Lorenz–Mie solver. FPE-MLP was also found to be faster and more accurate than the Nussenzveig approximation. These results demonstrate the potential of FPE-MLP for real-time inverse retrieval of droplet diameter and refractive index at spray scale.

Keywords: Mie Scattering; Rainbow Refractometry; Deep Learning; Fourier Positional Encoding

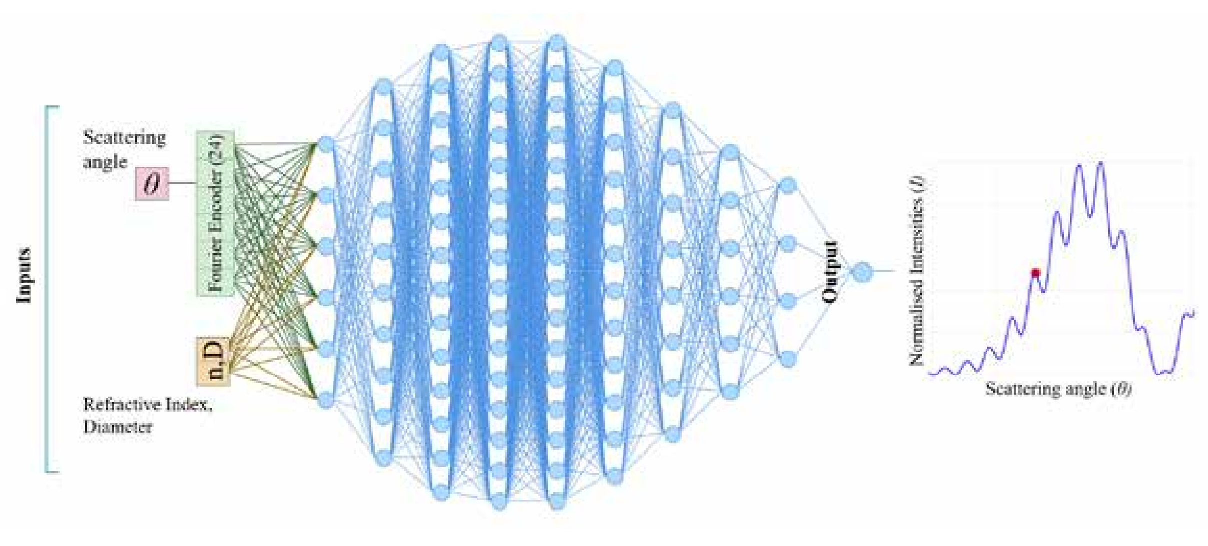


Figure 1. *Schematic of the FPE-MLP architecture. The scattering angle θ is passed through the positional encoder (24 features), while refractive index n and diameter D are concatenated directly. The combined 26-dimensional input is processed through a pyramidal MLP to produce the normalised scattering intensity I_{norm} at the output.*

Effect of Viscosity on the Bag Thickness in Aerodynamic Droplet Breakup

Samy Seddiki and Isaac M. Jackiw

Department of Chemical and Materials Engineering, University of Alberta, Canada

Abstract

For many industrial spray coating processes, the smallest droplets contribute significantly to waste as they are easily carried off-target by ambient air flows. These small droplets are predominantly generated during the rupture of bags formed by thin films during atomization. As the bag ruptures through the formation of holes, its thickness determines the size of the smallest droplets in the distribution. While viscosity is known to dampen fragmentation processes leading to larger average droplet sizes, the effect of viscosity on the bag thickness, and the smallest droplet sizes, is unknown. This phenomenon has been previously studied for aerodynamic droplet breakup of low-viscosity water droplets, where experiments have suggested that the thickness of these bags at rupture is on the order of 1-2 microns and may be independent of Weber number (We) in the range: $We < 30$. In this study, the effect of viscosity on the rupture thickness of the bag in aerodynamic droplet bag breakup is investigated. High-speed video imaging is used to analyse droplet breakup experiments in the bag breakup regime, using aqueous glycerol mixtures exhibiting a range of viscosities: [1 $\mathrm{mPa.s}$ (water); 450 $\mathrm{mPa.s}$ (95% glycerol)]. The retraction velocity of the holes on the ruptured bag is measured and the Taylor-Culick law for retracting films is used to estimate the bag thickness. The results indicate that the viscosity has only a minor effect on the median bag thickness which increases slightly with viscosity (0.7 → 1.1 $\mu\mathrm{m}$) and that there is no apparent dependence on the Weber number.

Ultra-High-Speed Investigation of Flash Boiling in Ethanol Droplet Chains Induced by Satellite Droplet Collisions

LukasEich*[1],HenrikSchmitz[1],PeterAugustin[1],WilkoRohlfs[1,2],ManuelReddemann[1]

[1]InstituteofHeatandMassTransfer,RWTHAachenUniversity,Aachen,Germany
[2]DepartmentofThermalandFluidEngineering,UniversityofTwente,Enschede,Netherlands
*Correspondingauthor:eich@wsa.rwth-aachen.de

Abstract

Improving the predictability of mixture formation in flash-boiling injection systems through numerical modeling requires precise experimental validation data at the single-droplet level. This study introduces a novel methodology to effectively trigger internal vaporization in a droplet chain through controlled droplet collisions. This approach enables the quantitative investigation of flash boiling in free-flying droplets, thereby facilitating the generation of high-fidelity validation data for advanced predictive models. Using a piezo-driven droplet chain generator with superheated ethanol, the excitation frequency is intentionally altered to generate satellite droplets that subsequently collide with the main droplets. These controlled collisions entrap microbubbles that act as nucleation sites, effectively triggering internal flash boiling at a superheat $\Delta T_{sh} \approx 14$ K or $R \approx 1.7$. The highly transient kinematics are captured using ultra-high-speed shadowgraphy at two million frames per second (2 Mfps) and analyzed via a customized volume-detection algorithm. This approach enables the extraction of highly resolved physical properties, including droplet volumes V_d , vapor mass flow rates $\dot{m}_V$, the vapor bubble radius r_V , and its radial growth rate dr_V/dt over time. The evaluated volumetric expansion reveals three distinct phases: an initial acceleration, a subsequent stagnation phase, and a thermally driven renewed acceleration. Future investigations will aim at acquiring a large data ensemble across different degrees of superheat.

Keywords: single-droplet flash boiling, droplet chain, satellite droplet collision, bubble growth, ethanol

Correlation-Based Turbulent Weber Number and Taylor Reynolds Number in a Fan-Stirred Isotropic Chamber: Hinze’s Relation and Breakup Regimes

VeliCanCos¸ar*[1],YeabsiraSimeka[1],ÖzgürErtunç[1],

[1]FluidDynamicsandSprayLaboratory,DepartmentofMechanicalEngineering,Özyeg˘in University,Istanbul,Türkiye
*Correspondingauthor:veli.cosar@ozu.edu.tr

Abstract

Single-droplet secondary breakup was investigated for 2 mm ethanol droplets in a fan-stirred, zero-mean turbulence chamber driven by eight corner-mounted propellers. Planar 2D PIV was used to obtain twopoint velocity correlations and second-order structure functions, while high-speed imaging was used to determine breakup occurrence and mode distribution. The droplet-scale velocity increment was evaluated from the measured structure function at $r = D$ and used to define a turbulent Weber number, W et (D), rather than an imposed mean relative velocity. The turbulent field covered $Re\lambda \simeq 300–680$, with Taylor microscales of $\lambda f = 4.32–5.31$ mm and Kolmogorov scales of $\eta = 78–115$ µm. Breakup probability increased with W et (D) and approached saturation for the strongest constant-speed and sinusoidal protocols. However, breakup occurred at Weber numbers below the classical critical Weber value, indicating that a dissipation-rate-only threshold is insufficient for finite-size droplets in a time-dependent, zero-mean turbulent field. The breakup-mode statistics showed a transition from predominantly vibrational breakup at lower forcing to more frequent bag, multiple-breakup, and unclassified events at higher forcing. The results support a probabilistic description in which breakup occurrence is organized by droplet-scale turbulent forcing, while the final breakup mode also depends on the forcing history during the droplet residence time.

Keywords: Droplet breakup; Two-point velocity correlation; HIT; Secondary atomization

Optical Measurement of Liquid Film Thickness in Swirl Atomizers with Correction for Refractive Distortion

KutayAnbarcı[1],MustafaAykanUzun[1],ÖzgürErtunç*[1]

[1]FluidDynamicsandSprayLaboratory,MechanicalEngineeringDepartment,
Özyeg˘inUniversity,Istanbul,Türkiye
*Correspondingauthor:ozgur.ertunc@ozyegin.edu.tr

Abstract
The liquid film thickness at the orifice of pressure-swirl injectors is a fundamental parameter governing primary atomization and the subsequent droplet size distribution. This research presents a method for measuring film thickness experimentally across distinct injector geometries up to a Reynolds number of 85,000. To capture the high-frequency film dynamics, a high-speed imaging system equipped with a long-distance microscope was employed, utilizing a transparent plexiglass extension specifically machined to fit the orifice exit. A primary technical challenge addressed in this work is the refractive distortion inherent to the circular geometry of the exit; this was successfully mitigated using a customdesigned calibration tool and a dedicated image-processing algorithm that reconstructed the true radial coordinates of the extended orifice as visualized through the plexiglass extension. The resulting dataset provides a comprehensive mapping of film thickness as a function of Reynolds number and is compared with analytical models to evaluate their predictive accuracy. This work presents an optical measurement methodology that combines image correction and automated film-detection algorithms to measure film thickness using high-speed imaging and specialized hardware.
Keywords: Swirl injector, Liquid film thickness, Refractive distortion correction, High-speed imaging

First Characterisation of a typical Aero-engine pre-filmer atomizer through Surface Curvature Analysis

AnbarasanSekar*[1],SaïdIdlahcen[1],StefanRHarth[2],JulienReveillon[1],BenjaminDuret[1], Jean-BernardBlaisot[1],François-XavierDemoulin[1]

[1]UnivRouenNormandie,INSARouenNormandie,CNRSCORIA,F-76000,Rouen,France
[2]Engler-Bunte-Institute,KarlsruheInstituteofTechnology,76131Karlsruhe,Germany
*Correspondingauthor:anbumsr@gmail.com

Abstract

The primary atomization process of a typical aero-engine pre-filmer atomizer is investigated through surface curvature analysis. The recent progress in the surface curvature analysis made possible for the application of the method to two-dimensional or line-of-sight imaging. The experimental data are acquired using back-lit imaging technique with a microscopic arrangement. The effects of external coflow in the prefilmer atomizer are explored while keeping the liquid flow rate constant. The quantitative information, such as length-curvature distribution, direct droplet size distribution, and equivalent droplet size distribution, is derived using the surface curvature analysis for the primary atomization region. A smooth transition of the curvature distribution is observed as the atomization progresses. The atomization processes in the pre-filmer atomizer were found to be highly affected by the external co-flow. A stable surface mean diameter is observed at the end of the primary atomization zone for each case. The surface-to-contour length ratio was also found to be affected by the external co-flow. The equivalent droplet size distribution indicates that the final droplet size distribution trend converges fast at the initial stages of the spray. Moreover, the sensitivity of the length-curvature distribution to the atomization processes shows an excellent way to observe different modes of atomization. This analysis provides a complete quantitative characterization of a typical aero-engine pre-filmer atomizer from the injector exit.

Keywords: Surface Curvature Analysis, Pre-filmer atomizer, Aero-engine

Flow Regime Analysis of Slit Nozzle Operating Conditions under Vacuum Using CFD and Optical Diagnostics for Powder Aerosol Deposition

M. Ghodbene[1,3,*], H. Auer[1], C. Franzke[2], J. Maaß[2], R. Moos[3]

[1]PowerCo SE, 38239 Salzgitter, Germany
[2]IAV GmbH, 10587 Berlin, Germany
[3]Department of Functional Materials, University of Bayreuth, 95447 Bayreuth, Germany
*Corresponding author: maroua.ghodbene@powerco.de

Abstract
Powder Aerosol Deposition (PAD) enables the fabrication of dense ceramic coatings with micrometrescale thickness at room temperature by accelerating ceramic particles in a carrier gas jet and consolidating them on a substrate via kinetic impact. The nozzle operating point critically governs jet stability, particle acceleration, and deposition uniformity; however, suitable operating conditions are still selected empirically, with limited validation via compressible flow physics. In this work, the flow jet of a convergent slit nozzle with an outlet area of approximately 0.5 x 10 mm^2 used in PAD is investigated using a combined experimental–numerical approach. Particle Image Velocimetry (PIV), Schlieren imaging and three-dimensional compressible CFD simulations of gas and gas–particle flow investigates the formation of distinct flow regimes. While PAD is commonly operated at very low chamber pressures below 10 mbar, the present study investigates a broader parameter space, covering chamber pressures from 1 to 100 mbar, normalized volumetric flow rates from 1 to 100 NL min^{-1}, and nozzle pressure ratio (NPR) values between 5 and approximately 150. The simulations reveal two distinct jet flow regimes, ranging from strongly underexpanded jets with pronounced shock cell structures at high NPR values ($\gtrsim$ 40) to near-ideally expanded flows at lower NPR ($\lesssim$ 30). Schlieren imaging confirms the predicted expansion and shock structures, while PIV provides quantitative axial and radial velocity distributions, enabling a physically grounded classification. The analysis demonstrates that stable particle transport and film formation in PAD are associated with specific combinations of chamber pressure, and volumetric flow rate, defining characteristic nozzle operating regimes.
Keywords: aerosol deposition method; vacuum kinetic spray; vacuum cold spray; shadowgraph imaging; schlieren imaging; particle image velocimetry; flow visualization; numerical simulation

LES investigation of the VOF-LPT coupling method for Dual Gas-Assisted Sheet Atomizers

JuliaRoeb*[1,2],OliverT.Stein[1]

[1]SimulationofReactingThermo-FluidSystems,Engler-Bunte-Institut,KarlsruheInstituteof Technology,Germany
[2]InstituteforTechnicalChemistry,KarlsruheInstituteofTechnology,Germany
*Correspondingauthor:julia.roeb@kit.edu

Abstract
The gasification of plastic waste-based pyrolysis oils in entrained-flow gasifiers has recently received increasing interest. Efficient atomization, providing high surface-to-volume ratios, is essential for rapid conversion of highly viscous liquid feedstocks into high-quality syngas, enabling fast evaporation and complete fuel conversion. Dual gas-assisted sheet atomizers may enhance atomization at higher liquid mass flow rates compared to conventional coaxial gas-assisted systems. However, primary atomization in such configurations exhibits a wide range of breakup morphologies associated with pulsating instabilities [1] which can lead to periodic ligament ejection that delays droplet formation. In our previous large eddy simulation (LES) studies we have shown that the Volume-of-Fluid (VOF) method captures primary breakup of the liquid core and the formation of large ligaments well, even under pulsating conditions [2]. However, since a fully Eulerian description of secondary ligament breakup and droplet formation remains computationally prohibitive, we employ Lagrangian Parcel Tracking (LPT) to model these processes. The present study explores droplet formation under pulsating conditions by modeling the time-dependent transition from ligament breakup to LPT-based droplet injection. The pulsation-driven periodic effects on droplet formation rate and droplet size distribution are assessed. This approach bridges primary breakup captured in VOF-LES with a LPT-based description towards secondary atomization in dual-gas-assisted sheet atomizers.
Keywords: VOF-LES, LPT, primary breakup, secondary atomization

Holistic Simulation of Laser-Induced Ignition under Flashing Conditions of an Experimental Rocket Combustion Chamber

ChristianSessler[1],JanWilhelmGärtner[1],AndreasKronenburg[1],AlexanderBee[2],Justin Hardi[2],MichaelBörner[2]
∗

[1]InstituteforReactiveFlows,UniversityofStuttgart,Germany
[2]InstituteofSpacePropulsion,GermanAerospaceCenter(DLR),Lampoldshausen,Ger.
*Correspondingauthor:christian.sessler@irst.uni-stuttgart.de

Abstract

This study presents an integrated experimental and numerical analysis of laser-induced ignition in a rocket combustor featuring two coaxial injectors. Operating with gaseous methane and cryogenic liquid oxygen (LOX) under flashing conditions, the system is modeled using a compressible, Euler-Euler, one-fluid Reynolds-averaged Navier-Stokes (RANS) approach. To ensure high-fidelity inflow conditions, the numerical domain includes the LOX post and fuel dome, accounting for supersonic fuel dynamics, cryogenic flashing, and two-phase reacting flows. The solver's ability to replicate pre-ignition conditions was validated against experimental Schlieren images. In consideration of the pulsed laser and the highly dynamic inflow conditions, it is imperative to acknowledge that each experimental iteration constitutes a multitude of ignition attempts. In the course of the simulations, the initial three laser pulses were modelled, resulting in a total of 45 simulations. The results show excellent agreement with experimental data, correctly predicting the ignition outcome in 43 out of 45 cases. The analysis reveals that ignition failure is primarily driven by the presence of the liquid phase, as the energy required for evaporation significantly cools the gas phase.

Large Eddy Simulation of Thixotropic Jets: *A priori* Analysis of Novel Subgrid-Scale Terms

Marianne Abdelsayed*, Markus Klein

University of the Bundeswehr Munich, Department of Aerospace Engineering,
Institute of Applied Mathematics and Scientific Computing, Neubiberg, Germany
*Corresponding author: marianne.abdelsayed@unibw.de

Abstract

Thixotropy is a time-dependent shear-thinning fluid property, in which the viscosity decreases due to microstructural breakdown under shear and recovers when the fluid is at rest. Thixotropic fluids can be found in various spray applications, such as spray painting and the atomization of gel fuels. These applications are typically characterized by high Reynolds numbers, which makes direct numerical simulations (DNS) computationally expensive. Large eddy simulation (LES) offers a feasible alternative by resolving only the large scales while modeling the small scales.

In this study, we derive the filtered kinetic equation for a thixotropic fluid based on Moore's model [1] by applying a spatial low-pass filter. The studied flow configuration corresponds to thixotropic jet breakup into still Newtonian gas [2]. The Volume-of-Fluid (VOF) method was employed to capture the interface between the two phases. Since only the liquid phase is thixotropic, we introduce a volume fraction based Favre filter $\widetilde{\phi} = \overline{\alpha\phi}/\overline{\alpha}$ to ensure that the filtered structural integrity $\widetilde{\lambda}$ remains physically meaningful, even if only one of the phases is thixotropic. The filtered kinetic rate equation reads

$$\frac{\partial \bar{\alpha}\widetilde{\lambda}}{\partial t} + \frac{\partial \bar{u}_i \bar{\alpha}\widetilde{\lambda}}{\partial x_i} = \frac{\bar{\alpha}(1-\widetilde{\lambda})}{\tau_\theta} - \beta\dot{\bar{\gamma}}\bar{\alpha}\widetilde{\lambda} - \frac{\partial}{\partial x_i}\tau_{\mathrm{adv},\lambda,i} - \tau_{\mathrm{destruct}}, \tag{1}$$

where $\dot{\gamma} = \sqrt{2\overline{S}_{ij}\overline{S}_{ij}}$ is the resolved shear rate, τ_θ is the thixotropic time-scale and β is the destruction parameter. Two novel subgrid-scale (SGS) terms arise in Eq. 1. The first term describes the SGS advection of the structural parameter $\tau_{\mathrm{adv},\lambda,i} = \overline{u_i\alpha\lambda} - \overline{u}_i\overline{\alpha}\widetilde{\lambda}$, while the second term describes the SGS destruction of the microstructure $\tau_{\mathrm{destruct}} = \beta(\overline{\dot{\gamma}\alpha\lambda} - \dot{\bar{\gamma}}\overline{\alpha}\widetilde{\lambda})$. Note that $\dot{\bar{\gamma}} = f(\overline{u}_i)$.

To investigate the significance of these terms, we perform an *a priori* analysis based on filtered DNS data. Figure 1 shows the order of magnitude of the advection SGS term (left) and the destruction SGS term (right) at the interface, normalized by the resolved advection term, for various filter widths. The results indicate that the destruction SGS term is highly active in the subgrid-scales. Based on a correlation analysis and error estimation, our analysis further indicates that structural models provide a promising closure for this term.

Financial support from the German Research Foundation (DFG) under the project number KL 1456/11-1 (503977784) and helpful discussions with Bernhard Weigand are gratefully acknowledged.

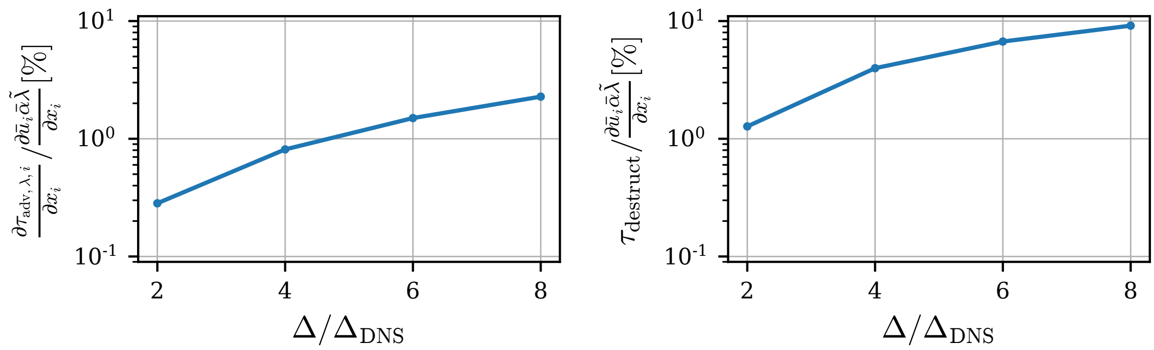


Figure 1. *Order of magnitude of the advection SGS term (left) and the destruction SGS term (right), normalized by the resolved advection term obtained from the filtered DNS, for various filter widths.*

References

[1] F. Moore, "The rheology of ceramic slips and bodies," Transactions of the British Ceramic Society **58**, 470–494 (1959).

[2] M. Abdelsayed and M. Klein, "Effect of thixotropy on primary jet breakup – A DNS study," International Journal of Multiphase Flow **194**, 105486 (2026).

On the Non-Deterministic Fingering Event in Single-Drop Impact on Dry Surfaces

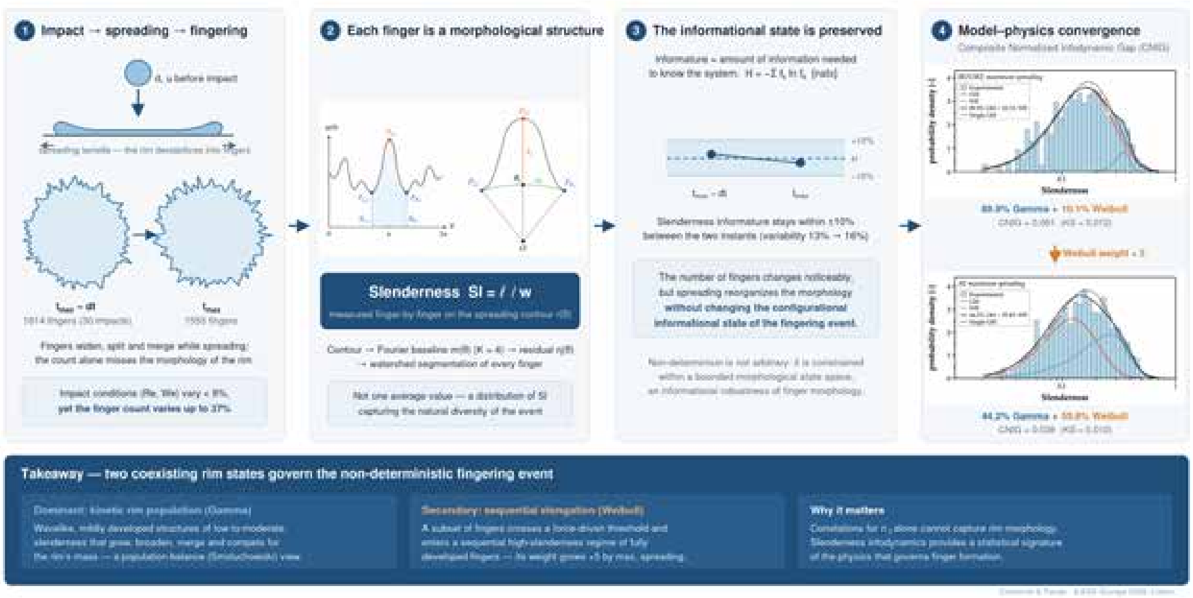
Miguel R. Clemente[1], Miguel O. Panão*[1]

[1] ADAI, Department of Mechanical Engineering, University of Coimbra, Portugal
*Corresponding author: miguel.panao@dem.uc.pt

Abstract

This study investigates the non-deterministic nature of the fingering event during the spreading of a single droplet that impacted a dry smooth surface. Instead of focusing only on the number of fingers, the analysis treats each finger as an evolving morphological structure and characterizes it through its slenderness, which is defined from its individual length and width extracted from the spreading contour. High-speed images from below were processed at two successive stages of the event, namely, the penultimate frame before maximum spreading and the frame at maximum spreading. The resulting finger slenderness distributions were analyzed statistically and through infodynamics. The results show that, although the number of fingers changes noticeably between both instants, the fingering event preserves a similar configurational informational state, while its morphology is reorganized. Model–physics convergence led to a binary mixture of gamma and Weibull distributions at maximum spreading. These results suggest that the non-deterministic nature of the fingering event is governed by a dominant kinetic rim-population process, whereas a secondary subset of fingers evolves into a more developed sequential high-slenderness regime.

Keywords: Drop impact; Fingering instability; Slenderness; Infodynamics; Model–physics convergence



Role of wettability on coupling between droplet dymamics and boiling during drop impact on heated surface

O. Caballina*[1], T. Potaufeux[1], A. C. García Wong[2], T. Czerwiec[2], G. Castanet[1]

[1] Université de Lorraine, CNRS, LEMTA, Nancy, France
[2] Université de Lorraine, CNRS, IJL, Nancy, France
*Corresponding author: ophelie.caballina@univ-lorraine.fr

Abstract

In the present study, the impact dynamics and heat transfer of a single water droplet impinging on a heated solid substrate are investigated experimentally. A high-speed shadowgraphy system is combined with infrared thermography to simultaneously characterize the droplet spreading (fig.1a) and the temperature evolution at the front surface where impact occurs ((fig.1b).

For IR thermography, a semi-transparent sapphire window coated with a high emissive layer is used as a substrate. The front-surface of the substrate is thus observed by the bottom though the sapphire. A well-posed inverse heat conduction problem is solved to reconstruct the local heat flux at the liquid–solid interface with high spatial and temporal resolution (fig.1c). In addition to the inverse heat conduction approach, the temperature field is accurately reconstructed by correcting the infrared camera signal for the intrinsic thermal emission of the sapphire substrate. A radiative model is implemented to discriminate the respective contributions of the emissive coating, the sapphire itself, and any external radiative sources.

Experiments are conducted over a wide range of Weber numbers and surface temperatures, from 80 °C up to 600 °C, on substrates with contrasted wettability obtained from different coatings. Throughout the investigated conditions, the maximum local heat flux density is shown to strongly depend on both surface temperature and wettability. It increases with surface temperature until the transition to the Leidenfrost regime, where the formation of a stable vapor layer suppresses direct liquid–solid contact and drastically limits heat transfer. This transition is found to occur at higher surface temperatures for less wetting surfaces. In addition, increasing the Weber number generally promotes liquid–solid contact, delaying the onset of complete levitation.

The total heat flow transferred during droplet impact results from a non-trivial interplay between local heat flux and the effectively wetted area. While higher surface temperatures enhance the heat flux density, they may simultaneously reduce the contact area due to vapor generation. Conversely, higher Weber numbers can increase the wetted area through enhanced spreading, but may also promote droplet fragmentation, thereby limiting the effective heat transfer. These results highlight the critical roles of wettability, impact inertia, and surface temperature in controlling droplet-scale heat transfer, and provide insight into the mechanisms governing spray cooling efficiency.

Keywords: Spray Cooling, Drop impact, Infra-red thermography imaging, Nucleate boiling, Wettability.

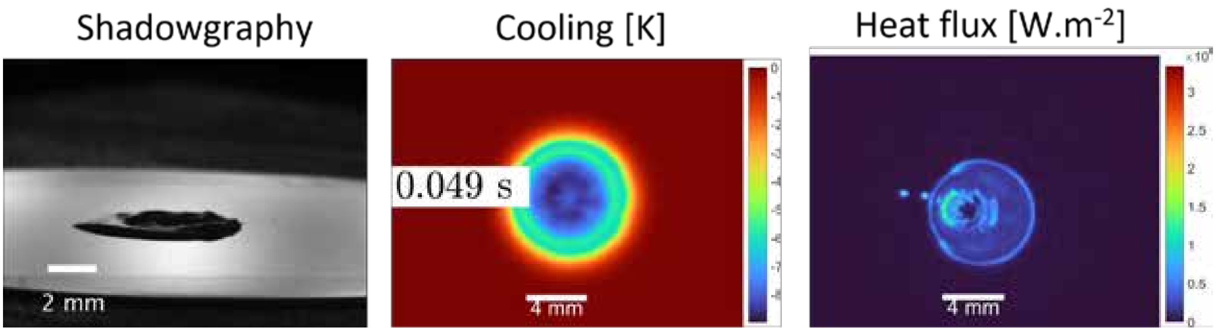


Figure 1. *Impact of a water drop at Weber number* $We = 106$ *on a surface initially at* T_{w0} *=125 °C.*

Near-Field Particle Image Analysis of Liquid Ammonia Sprays from a GDI Injector

Yi-Rong Chen*[1,2], Akihiro Hayakawa[1], Utkarsha Sonawane[3], Camille Hespel[3], Christine Mounaïm-Rousselle[3,4]

[1]Institute of Fluid Science, Tohoku University, Japan
[2] Department of Mechanical Engineering, National Central University, Taiwan
[3]PRISME, Université d'Orléans, France
[4]Institut Universitaire de France, IUF, Paris, France
*Corresponding author: chen.yi.rong.a4@tohoku.ac.jp

Abstract
Liquid ammonia (LNH3) spray combustion offers simplified fuel supply systems and rapid startup capability than gaseous ammonia, making it a promising carbon-free alternative for gas turbine or internal combustion engine applications. However, as a function of the injection conditions, LNH3 can undergo flash boiling, characterized by intense evaporation during injection, when introduced into environments with ambient pressure significantly lower than its saturation pressure. In this study, the flash-boiling atomisation characteristics of LNH3 are investigated using a single-hole gasoline direct injector. Measurements are conducted at 35 mm downstream of the nozzle tip during a stable injection period, where the spray is fully developed. Near-field visualization was achieved using a high-speed camera coupled with microscope optics and Nd:YAG pulsed-laser for illumination, yielding a spatial resolution of 2.3 μm/pixel. More than 200 images from 200 injection events were analysed using particle image analysis to determine particle size distributions across different degrees of superheat conditions. By adjusting the imaging position, spatial distributions of the arithmetic mean diameter and Sauter mean diameter, SMD as functions of radial positions are obtained. An inverse relationship between the degree of superheat and SMD is observed, demonstrating enhanced atomisation at higher superheat levels.
Keywords: Liquid ammonia spray, Flash-boiling, Particle image analysis, Droplet size distribution

Effect of fluid property on the atomization of perturbed liquid jets

KamalKumarYadav*[1],AdityaSaurabh[1]

[1]DepartmentofMechanicalEngineering,IndianInstituteofTechnology,Kanpur,India
*Correspondingauthor:yadavkamal2012@gmail.com

Abstract
This study investigates the effect of mechanical perturbation on the atomization of a cylindrical liquid jet injected into a quiescent gaseous medium. A periodic disturbance at 20 kHz is introduced at the jet inlet using a mechanical oscillator. Experiments are conducted using ethanol–water (70E–30DI) and glycerol–water (60G–40DI) mixtures to examine the effects of surface tension and viscosity on spray behaviour. The Weber number ranges from 5925 to 27,420 for the ethanol–water mixture and from 2350 to 28,890 for the glycerol–water mixture. The ethanol–water mixture undergoes earlier and more irregular breakup, whereas the more viscous glycerol–water mixture maintains a longer liquid core and forms more organized periodic structures. At the highest Weber number for the glycerol–water mixture, the measured structure spacing gives an estimated frequency of 17.6 kHz, which is reasonably close to the imposed 20 kHz frequency and suggests that the observed structures are linked to the mechanical perturbation. The breakup length generally increases with Weber number for both fluids, while the Sauter mean diameter of the spray generally decreases with increasing Weber number. These results show that fluid properties strongly influence the breakup and atomization of mechanically perturbed liquid jets.

Keywords: Atomization, surface tension, viscosity, liquid jet, mechanical perturbation, spray structure, breakup length, droplet size, Weber number

Phase-Resolved Imaging of Aperture Plate Oscillation and Jet Timing in a Vibrating Mesh Nebulizer

HenrikSchmitz*[1],TianaTietz[2],PeterWilhelmAugustin[1],LukasEich[1],WilkoRohlfs[1,4],Anja LenaThiebes[2],ChristianCornelissen[2,3],ManuelArminReddemann[1]

[1]InstituteofHeatandMassTransfer,RWTHAachenUniversity,Germany
[2]DepartmentofBiohybridandMedicalTextiles(BioTex)AME-InstituteofAppliedMedical Engineering,HelmholtzInstitute,RWTHAachenUniversity,Germany
[3]ClinicforPneumologyandInternisticIntensiveMedicine(MedicalClinicV),MedicalFaculty, RWTHAachenUniversityHospital,Germany
[4]FacultyofEngineeringandTechnology,UniversityofTwente,theNetherlands
*Correspondingauthor:hschmitz@wsa.rwth-aachen.de

Abstract
This study presents an ultra-high-speed microscopy methodology, combined with a displacement-tracking algorithm to resolve microscale fluid dynamics in vibrating mesh nebulizers. By mapping the mechanical mesh oscillation phase with liquid jet ejection timing, this technique can address the theoretical debate between macroscopic inertial pumping and microscale acoustic compression. As a proof of concept, the methodology is applied to an exemplary jet ejection event, successfully demonstrating the capability of the method to relate mesh kinematics and fluid ejection. Furthermore, the high-resolution recordings capture phenomena previously inaccessible to direct observation in microscopic scale aperture plates. The jet's disintegration via Plateau-Rayleigh instabilities is shown and they confirm the predicted existence of jet-droplet collisions. The recordings show that individual jets initially break up without interacting with neighboring jets upon exiting the aperture, justifying the single-aperture assumption in numerical models. Contrary to the clean nozzle exit typically assumed, an oscillating liquid film on the aperture is observed.
Keywords: Vibrating mesh nebulizer, acoustic pumping, ultra high-speed microscopy, jet breakup

Validation of the Primary-Secondary Breakup Transition in Close-Coupled Gas Atomization of Molten Metals

StefanvanLaarhoven*[1],NielsDeen[1],JorisRemmer[1],LydiaAchelis[2],UdoFritsching[2],Anne Geppert[3],GiuliaFinotello[1]

[1]PowerandFlow,DepartmentofMechanicalEngineering,EindhovenUniversityof Technology,TheNetherlands
[2]ParticlesandProcessEngineering,FacultyofProductionEngineering,Universityof Bremen,Germany
[3]LeibnizInstituteforMaterialsEngineering-IWT,Bremen,Germany
*Correspondingauthor:s.j.w.v.laarhoven@tue.nl

Abstract
Close-coupled gas atomization is widely used for producing metal powders for additive manufacturing. The primary and secondary breakup mechanisms during atomization remain difficult to characterize. In existing multiscale models, including the Computational Fluid Dynamics (CFD) framework of Thuy-Petrov et al. [1], the coupling plane (the axial location where the simulation switches from primary to secondary atomization) is typically prescribed based on criteria such as droplet size and sphericity. However, this coupling plane has never been directly validated against experiments with real molten metal. In this study, we combine CFD simulations and high-speed imaging of molten steel atomization to identify and verify the physical location of the coupling plane. The gas-liquid interface is resolved using a Volume of Fluid approach and Adaptive Mesh Refinement, while turbulence is captured using Large Eddy Simulation. Experimental data is provided by the University of Bremen and Leibniz Institute of Materials Engineering IWT [2, 3], obtained using side-view high-speed optical imaging with cameras operating at up to 29 kfps. Breakup length is extracted from time-resolved simulations as a time-dependent quantity, from which statistical distributions are constructed and compared directly with experimental results. This approach enables a quantitative assessment of breakup length statistics from both experiments and simulations. In this work, the methodology is demonstrated and a preliminary cross-case comparison is presented. A fully matched validation of the coupling plane position will be reported in future work once the simulation of the target case is completed.
Keywords: Close-coupled gas atomization, Primary breakup, Breakup length, Interface length

Coupled VOF-LPT Approach to Investigate the Effects of Flow and Nozzle Parameters on Spray Characteristics

MarcoPiras[1],GiancarloSorrentino*[1],RaffaeleRagucci[1]

[1]NationalResearchCouncil,STEMS-CNR,viaMarconi4,80125,Naples,Italy
*Correspondingauthor:giancarlo.sorrentino@stems.cnr.it

Abstract
This study investigates liquid-jet-in-crossflow (LJICF) atomization to analyze the role of the injector design on primary breakup using a coupled OpenFOAM solver. The numerical methodology combines a Volume of Fluid (VOF) formulation for the near-nozzle interface dynamics with a two-way-coupled Lagrangian particle tracking (LPT) approach for the downstream dispersed phase. The investigation is structured in two stages: a preliminary statistical validation of the spray characteristics using a RANSbased framework, followed by a high-fidelity Large-Eddy Simulation (LES) employing the WALE subgridscale model and Adaptive Mesh Refinement (AMR) to accurately resolve the liquid core morphology. To isolate the effect of nozzle geometry, the LES analysis focuses on purely hydrodynamic interactions and compares a standard circular nozzle with a transversely oriented elliptical nozzle. The analysis indicates that the circular jet rapidly flattens into a horseshoe-like shape, which promotes fragmentation, whereas the elliptical jet shows reduced lateral expansion. Driven by strong localized capillary forces, the elliptical configuration maintains structural coherence and significantly delays the onset of catastrophic primary breakup. The resulting VOF-LPT workflow provides a robust tool to capture geometry-induced atomization trends, highlighting the potential of non-circular nozzles to control crossflow-driven sprays in advanced propulsion systems.
Keywords: Spray in crossflow, VOF-LPT coupling, RANS, OpenFoam

CFD simulation of automotive multi-hole injector ammonia flow

Charalambos Chasos*[1]

[1]FredU-CFD-LAB, Mechanical Engineering Department, Frederick University, Cyprus
*Corresponding author: c.chasos@frederick.ac.cy

Abstract
Ammonia is considered as a very promising and readily available solution for decarbonisation of the transport sector and reaching net-zero emissions by the year 2050. There is a great challenge to fully model ammonia liquid fuel injection, in order to adapt ammonia in vehicle and marine internal combustion engines. The first objective of the present work was to address ammonia computational fluid dynamics (CFD) modelling with emphasis on ammonia properties and the trans-critical liquid ammonia behaviour which imposes highly complex atomization, evaporation and boiling phenomena within combustion chambers at varying conditions and intense turbulence fields. The second objective was to design a typical automotive multi-hole injector and carry out CFD simulations of ammonia injector internal flow and the emerging spray jet flow. The final objective was to identify the injection flow patterns of liquid and gas phase ammonia within the injector and at the vicinity of the nozzle exit, as well as compare jet characteristics, notably liquid breakup length and jet spreading angle against semi-empirical model data in the absence of experiments. The methodology employed in the present work included the computer-aided design (CAD) of the multi-hole injector, the generation of the computational mesh and the CFD simulations performed with the STAR-CD CFD code. Simulations were carried out for constant injection pressure 120 bar at increasing temperature and pressure of the constant-volume chamber. The results revealed that ammonia flow is highly complex, including ammonia and gas turbulent flows which strongly interact and produce a deflected spray jet from the nozzle.

Keywords: Liquid ammonia, multi-hole injector, constant-volume chamber, atomization, CFD simulation.

Introduction
For the modelling and setup of realistic ammonia injection, mixing and combustion simulations, it is required to fully characterize ammonia properties in liquid and gaseous form, as well as accurately predict the complex ammonia injection, air/fuel mixing and turbulent phenomena for wide range of operating conditions. To this end, the present work addressed the highly turbulent liquid and gaseous ammonia injection and air flow phenomena occurring in a constant volume chamber resembling internal combustion engine cylinder conditions.

Methodology
The automotive multi-hole injector, which was designed in the present work using CAD software, was selected from literature, for which the geometry details, injection pressure and duration were available. The Eulerian methodology with the volume of fluid (VOF) method and the Reynolds Averaged Navier-Stokes (RANS) framework for turbulence treatment were employed for the two-phase flow simulation of the internal and external injector flow of the atomizing liquid jet emerging from the nozzle.

Results

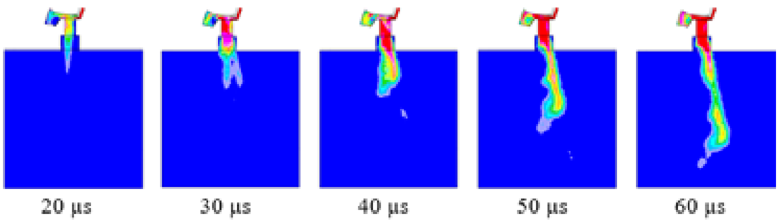


Figure 1. *Ammonia VOF field predictions at elapsed time after start of injection.*

Optical investigation of effect of L/D ratio on transcritical cryogenic sprays

FergusMurphy-Hunt*[1],CampbellMarlow-Eastwood[1],MartinWhite[1],CyrilCrua[1]

[1]EnergyandMaterialsEngineeringResearchCentre,UniversityofSussex,UK
*Correspondingauthor:f.murphy-hunt@sussex.ac.uk

Abstract
Prediction of the onset and severity of flash boiling in cryogenic sprays is important as it significantly impacts spray morphology, mixing behaviour, and heat transfer. The influence of nozzle length-to-diameter ratio (L/D) on flash boiling behaviour under transcritical conditions is a sparsely researched area with limited experimental data. Near-nozzle observations were successfully obtained using long-distance shadowgraph imaging across nozzle L/D ratios of 1, 3, and 5 under subcritical, transcritical, and supercritical upstream conditions. Cone angle was found to increase generally with L/D ratio, and was highly sensitive to upstream fluid temperature near the critical point, with a 7 K reduction producing a 16° cone angle increase for L/D = 3. Turbulent margin thickness near the nozzle also increased with L/D ratio. The applicability of superheat degree (Ts) as a predictive metric is assessed against the dataset and found inapplicable in its standard form at supercritical conditions. The Widom line is proposed as a substitute for saturation boundary to calculate a modified superheat degree (Ts∗).This is evaluated as a predictive metric for pseudo-flash-boiling severity under supercritical conditions.
Keywords: Cryogenics, LN2 , Transcritical mixing, Flash boiling, L/D ratio

Investigation of flash-boiling processes in cryogenic sprays

Campbell Brocc Marlow-Eastwood[1], Fergus Murphy-Hunt[1], Yuzhao Liu[2], Konstantina Vogiatzaki[2], Yekuan Shentu[3], Cyril Crua[1]

[1] Energy and Materials Engineering Research Centre, University of Sussex, Brighton, UK
[2] Department of Engineering Science, University of Oxford, Oxford, UK
[3] Castrol Ltd, Technology Centre, Whitchurch Hill, Pangbourne, Berkshire, RG8 7QR, UK
Corresponding author: C.Marlow-Eastwood@sussex.ac.uk

Abstract

Efficient refuelling of LH_2 HGVs rely on rapidly cooling the vehicles fuel tank to prevent over pressurisation and minimise boil-off losses. However, the initial state of the injected fluid and receiving vessel can drastically change the thermodynamics during this cycle. As cryogenic liquids are stored above atmospheric pressure to minimise boil-off losses during storage, the bulk fluid remains at a saturation pressure above standard atmospheric conditions, ensuring flash-boiling when injected into lower-pressure environments. Flashing can drastically change the atomisation, vapour mass fraction and spray dispersion angle, which influence initial tank cool-down during filling processes. This can present challenges during fluid transfer or refuelling cycles for no-vent LH_2 vehicles, which rely on rapid tank cooling. In this study, LN_2 was used as a surrogate for LH_2 to investigate the degree of flash-boiling during the initial stages of a tank filling cycle. A saturation-environment pressure ratio (R_p) range of 1 – 3.9 was selected under two different atomisation regimes to represent typical LH_2 refuelling conditions. To account for the differences in thermofluid properties between LH_2 and LN_2, the reduced temperature relative to critical point (T_r), Jakob number (J_a) and nucleation parameter (χ) were matched independently to evaluate flash-boiling and surrogate fluid similarity. A combination of shadowgraphy and PIV analysis was used to characterise each spray condition at different R_p. Physical parameters such as the length-to-diameter ratio (L/D) and mass flow rates were matched based on sLH_2 tank geometry. The results from this study will be used to inform LH_2 refuelling strategies and help to further validate CFD models.

Keywords: Cryogenics, flash-boiling, LH_2 mobility, optical analysis, surrogate study, tank filling.

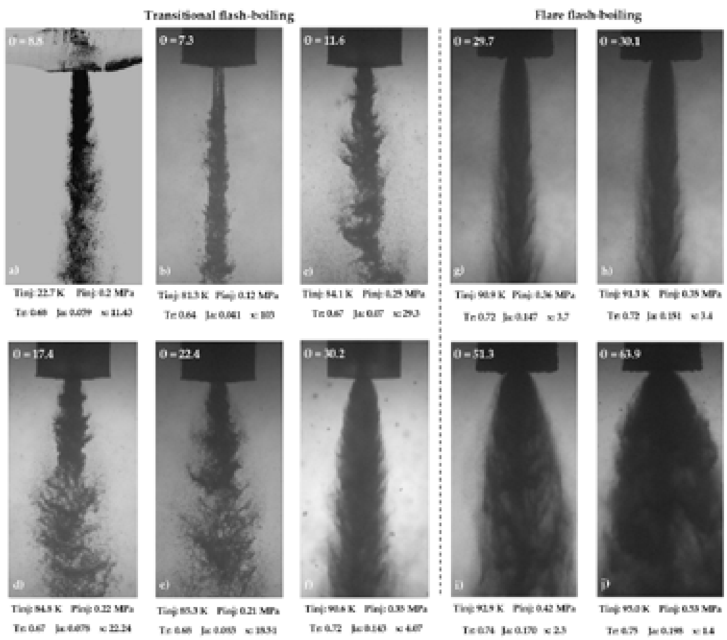


Figure 1. *Shows the experimental setup with high-speed camera, LED backlight and a flashing LN_2 spray with the spray dispersion angle.*

Unified Modeling of In-Nozzle Flow and Aerodynamic–Thermodynamic Eulerian-Lagrangian Breakup for Predicting Flash Boiling Spray Behavior

Mofetoluwa Fagade, Frans Joseph Arguelles, Sean King, XiaoHang Fang*

Department of Mechanical and Manufacturing Engineering, University of Calgary, Calgary, Alberta, Canada
*Corresponding author: xiaohang.fang@ucalgary.ca

Abstract

Flash boiling sprays present unique challenges in internal combustion engine modeling, as aerodynamic and thermodynamic instabilities interact throughout the atomization process. Under conditions of elevated fuel temperature or reduced ambient pressure, the injected fuel undergoes rapid phase change, initiating effervescent processes within the liquid fuel. Although flash boiling physics occur simultaneously with aerodynamic breakup, most numerical studies treat these mechanisms sequentially and often neglect in-nozzle effects. This work introduces a novel framework that incorporates in-nozzle nucleation with a thermodynamic breakup formulation embedded directly within the classical Kelvin–Helmholtz and Rayleigh–Taylor (KH–RT) model. The framework dynamically selects the dominant breakup mechanism based on local injection and ambient conditions, allowing the governing process to evolve naturally as the spray develops. Bubble dynamics are modeled using the incompressible Rayleigh–Plesset equation, integrated numerically at each timestep. The outward pressure generated by bubble growth competes with the surface tension forces that stabilize the droplet, and thermodynamic breakup is initiated when this pressure exceeds the restoring force. The resulting radial velocity is transferred to child droplets upon breakup. The proposed approach is implemented within an Eulerian–Lagrangian spray solver and evaluated under Engine Combustion Network flash boiling conditions. Results demonstrate the model's ability to capture characteristic flash boiling spray behavior, including spray collapse and plume merging at high superheat conditions, and reasonable agreement with experimental penetration lengths and schlieren imaging. By combining in-nozzle nucleation physics with coupled thermal and aerodynamic breakup, the framework provides a physically consistent foundation for simulating flash boiling sprays across a range of operating conditions.

Keywords: Spray G, Breakup Modeling, Flash Boiling, Bubble Growth, In-Nozzle Flow

A similarity analysis of cryogenic spray characteristics under sub-critical pressure

YuzhaoLiu[1],JinhongLi[1],CampbellBroccMarlow-Eastwood[2],CyrilCrua[2],YekuanShentu[3], KonstantinaVogiatzaki*[1]

[1]ThermalPropulsionSystemsResearchGroup,CryogenicFluidDynamicsLab,Department ofEngineeringScience,UniversityofOxford,Oxford,UK
[2]EnergyandMaterialsEngineeringResearchCentre,UniversityofSussex,Brighton,UK
[3]CastrolLtd,TechnologyCentre,Pangbourne,UK
*Correspondingauthor:konstantina.vogiatzaki@eng.ox.ac.uk

Abstract
Hydrogen has been widely regarded as a promising alternative fuel for zero emission future mobility. Understanding cryogenic hydrogen spray morphology and heat transfer characteristics is critical for the production, distribution, dispensing and end use for mobility applications that offer performance parity to conventional liquid fuels. However, due to the handling complexity and limited availability of hydrogen, direct optical experiments using cryogenic hydrogen remain challenging. Consequently, surrogate fluids are commonly employed to replicate hydrogen spray behaviour in experimental studies. In this context, the present study conducts a theoretical and numerical investigation exploring the potential of establishing rigorous similarity criteria between different cryogenic fluids and more specifically cryogenic hydrogen and helium/nitrogen sprays under subcritical pressure. The analysis focuses on the key governing non-dimensional parameters and their influence on spray morphology and mass transfer. In the theoretical section, the relevant dimensionless numbers, their physical relationships, and the methodology for constructing similarity cases between different fluids are presented. Subsequently, numerical simulations are conducted to investigate these cases. The simulation part of the work employs a newly developed OpenFOAM solver, coolFoam, which uses a volume-of-fluid (VOF) approach and provides a unified framework capable of handling cryogenic flows across different phase regions. Spray data are obtained under various operating conditions. The similarity of spray tip penetration evolution, spray angle, and mass transfer characteristics are analyzed theoretically. The results indicate that similarity theory could establish relationships among different cryogenic fluids, enabling more readily available cryogenic fluids to act as surrogates for expensive and hazardous fluids like hydrogen while preserving key thermodynamic and fluid-dynamic behaviours. This research outcome will provide valuable references for optimizing and designing more efficient test conditions for cryogenic applications.
Keywords: cryogenic spray, hydrogen, similarity, OpenFOAM, phase change, sub-critical

Measurement and Modelling of Equivalence Ratio Distribution in Narrow-cone-angle Hollow-cone Hydrogen Jets

Gyuhan Bae[1], Juwan Lim[1], Youngmin Ki[2], Heetae Yang[2], Choongsik Bae[2], Seoksu Moon*[1]

[1]Department of Mechanical Engineering, Inha University, Republic of Korea
[2]Department of Mechanical Engineering, Korea Advanced Institute of Science and Technology, Republic of Korea
*Corresponding author: ss.moon@inha.ac.kr

Abstract
Understanding the mixing characteristics of narrow-cone-angle hollow-cone hydrogen jets is essential for optimizing hydrogen direct-injection strategies over a wide range of combustion modes, including stratified combustion. In this study, the equivalence ratio distribution of a narrow-cone-angle hollowcone hydrogen jet is experimentally measured from the near-nozzle supersonic jet sheet development region to the downstream region after jet collapse. Laser-induced breakdown spectroscopy (LIBS) is employed to quantify the local equivalence ratio in a non-reacting hydrogen jet within a constant-volume chamber, while Schlieren imaging is simultaneously used to visualize jet development. The measured axial equivalence ratio first increases in the upstream region, reaches a local peak near the jet sheet merging location, and then gradually decreases downstream. Meanwhile, the radial equivalence ratio exhibits a Gaussian-like profile with a maximum at the jet centre. These results indicate that the upstream mixing process is strongly affected by hollow-cone jet collapse, while the downstream jet behaves similarly to a turbulent round jet but with a more gradual axial decay due to enhanced near-nozzle entrainment. Based on the measured jet characteristics, a lightweight one-dimensional analytical model is proposed by combining a conventional spray-based equivalence ratio model with a pseudo-jet concept to account for the wider cross-sectional area of the hollow-cone jet. The proposed model reasonably reproduces both the axial and radial equivalence ratio distributions, yielding coefficients of determination of 0.862 and 0.902, respectively. The results demonstrate that the proposed approach provides an effective and computationally efficient means of predicting the equivalence ratio distribution of hollow-cone hydrogen jets.
Keywords: Hydrogen direct injection engine, Hollow-cone hydrogen jet, Equivalence ratio distribution

Similarity Study on Turbulence behaviour for Cryogenic Hydrogen Jets Across Thermodynamic Regimes

JinhongLi[1],GiovanniTretola[1],YuzhaoLiu[1],KonstantinaVogiatzaki*[1]

[1]ThermalPropulsionSystemsResearchGroup,CryogenicFluidDynamicsLab,Department ofEngineeringScience,UniversityofOxford,UK
*Correspondingauthor:konstantina.vogiatzaki@eng.ox.ac.uk

Abstract
Hydrogen is increasingly considered a clean energy carrier for propulsion, combustion, and powergeneration systems, where high-speed injection controls entrainment, mixing, flame stabilisation, and safety-relevant dispersion. Under cryogenic conditions, these processes are strongly affected by realfluid thermodynamics, particularly across transcritical and supercritical regimes where pseudo-boiling effects and sharp property variations may occur. In this work, large-eddy simulations (LES) of turbulent cryogenic hydrogen jets are performed for a range of injector temperatures spanning dense-fluid, transcritical, and gas-like supercritical conditions. The effects of injection temperature, ambient conditions, and density ratio on jet morphology, penetration, spreading, and turbulent mixing are examined. Similarity across thermodynamic regimes is assessed using dimensionless scaling parameters and turbulence diagnostics. Selected cases are also simulated in both planar and fully three-dimensional configurations to evaluate the suitability of reduced-dimensional simulations. The results show that injection temperature strongly affects the near-field jet structure, producing transitions from liquid-spray-like or dense-fluid behaviour to gaseous-jet-like mixing as the flow moves away from the pseudo-boiling region. Jet penetration length scales primarily with the density ratio between the injected hydrogen and the ambient gas, consistent with classical variable-density jet behaviour. Planar simulations predict earlier breakup and heating than 3D LES, but capture similar breakup and mixing trends once initiated.

Keywords: Hydrogen, similarity, OpenFOAM, phase change, trans-critical, pseudo-boiling, large-eddy simulation (LES); cryogenic turbulence

Experimental And Computational Investigation Of Turbulent Mixing In Under-Expanded Hydrogen Jets For Gaseous Propulsion Applications

FrancescoDuronio*[1,2],AlessandroMontanaro[2],AndreaDiMascio[1],AngeloDeVita[1]

[1]DipartimentodiIngegneriaIndustrialeInformazioneediEconomia-UniversitàdegliStudi dell'Aquila,Italia
[2]ConsiglioNazionaledelleRicercheIstitutodiScienzeeTecnologieperl'Energiaela MobilitàSostenibili,CNR-STEMS,Napoli,Italia
*Correspondingauthor:francesco.duronio@univaq.it

Abstract
Direct injection (DI) of gaseous fuels, such as hydrogen, commonly produces under-expanded jet flows. A thorough understanding of these jets is essential to the design and optimization of DI-based propulsion systems, as the injection process strongly influences mixture formation, combustion efficiency, and pollutant emissions. Consequently, both injector design and the resulting jet dynamics require detailed investigation to ensure effective air–fuel mixing under realistic operating conditions. This work presents an integrated experimental and numerical study of hydrogen jets relevant to advanced combustion concepts. The injector examined generates singular under-expanded jets, whose spatial evolution was captured using schlieren visualization. These experimental measurements were used to assess and validate a computational framework, enabling an in-depth analysis of injection and jet development. Given the complexity of under-expanded jet flows, computational modeling plays a key role in improving physical understanding and supporting injector design and optimization. To this end, a high-order CFD methodology was employed to accurately capture the characteristic features of under-expanded hydrogen jets. Computational costs were reduced through state-of-the-art GPU-based parallel computing techniques. Additionally, a simplified, in-house boundary-condition strategy was developed to emulate the internal nozzle flow across a range of pressure ratios. The predictive capability of the numerical approach was evaluated by comparing with schlieren experiments, showing fairly good agreement between simulations and measurements. The numerical results enabled a detailed characterization of jet structure and morphology across various operating conditions, offering insight into the associated flow-field features. Finally, mixture quality was quantified using probability density functions, providing an objective measure of turbulent mixing for the different injection regimes considered. The large-eddy simulations performed in this study provide valuable guidance for the development of hydrogen injectors and, more broadly, gaseous-fueled propulsion systems.
Keywords: Hydrogen injection; Turbulent mixing; Schlieren Jet Characterization

Spray Cooling with Relative Nozzle–Substrate Motion: A Physics-Based Framework for Transient and Intermittent Heat Transfer

NilojenduBanerjee*[1],MarcoMarengo[1],CameronTropea[2]

[1]DepartmentofCivilEngineeringandArchitecture,UniversityofPavia,viaFerrata3,27100, Pavia,Italy
[2]InstituteofFluidMechanicsandAerodynamics,TechnicalUniversityofDarmstadt,64287, Darmstadt,Germany
*Correspondingauthor:marco.marengo@unipv.it

Abstract

Spray cooling with relative motion between a spray and a heated substrate introduces a distinct form of intermittency that arises from the translation of the spray footprint, rather than from temporal modulation of the liquid supply. In such configurations, intermittency is generated either by a moving nozzle sweeping over a stationary surface or by a moving surface passing beneath a fixed nozzle, leading each surface location to undergo repeated cycles of droplet impingement and thermal recovery. These modes of operation are representative of a wide range of practical systems, including rotating machinery, translating substrates, and scanning spray-based thermal processing. In the present work, a physics-based modeling framework is developed to describe the resulting transient heat transfer under relative nozzle–substrate motion. The approach resolves the local droplet number flux associated with a translating spray footprint and accounts for modifications to droplet impact kinematics induced by relative motion, while explicitly distinguishing between nozzle velocity and surface velocity as independent contributors to the relative speed. Heat transfer from individual droplets is modeled using regime-dependent formulations for film [1], transition [2], and nucleate boiling [3]. However, the operating conditions examined in the present study correspond predominantly to the nucleate boiling regime, and the reported results therefore focus on nucleate boiling behavior. The spatio-temporal evolution of surface temperature is obtained from a coupled transient conduction model within the substrate. The framework is constructed to capture the temperature variations arising from repeated exposure and recovery as the spray traverses the surface and is expected to reveal that the effects of spray motion may be represented as an effective intermittent spray characterized by a spatially dependent duty cycle and cycle frequency governed by the spray footprint and relative motion. The formulation is readily extended to arbitrary nozzle trajectories and velocities and enables investigation of the inverse problem, namely, determining the nozzle or surface motion required to maintain the substrate temperature within prescribed bounds. The proposed approach provides a unified basis for analyzing and designing spray cooling systems involving relative motion.
Keywords: spray cooling, intermittency, duty cycle, frequency

Numerical simulation of dynamic spray painting on plastic substrate with electrostatically supported high-speed rotary bell

Q. Ye*[1], O. Tiedje[1], S. Popp[2], H. Dubbe[2]

[1]Fraunhofer Institute for Manufacturing Engineering and Automation, Stuttgart, Germany
[2]University of Applied Sciences Esslingen, Esslingen, Germany
*Corresponding author: qiaoyan.ye@ipa.fraunhofer.de

Abstract

Electrostatically supported high-speed rotary bells are widely used in high-quality organic coating applications, due to their high paint transfer efficiency. While numerical simulations have become an established tool for understanding, developing, and optimizing electrostatic spray processes on grounded metallic workpieces, plastic coating remains significantly more challenging. Due to the low electric conductivity of plastic substrates, surface charges can accumulate during the coating process, leading to electrostatic repulsion of subsequently arriving droplets and to strongly non-uniform electric field distributions.

This contribution presents a numerical study of electrostatically assisted plastic coating, with water-based paint material. Transient CFD simulations with static and dynamically moving bell are performed using a dynamic moving mesh approach in the commercial CFD code ANSYS Fluent. A dynamic droplet charge model is developed based on experimentally measured electric current data. Furthermore, a time-dependent effective grounding condition is introduced to account for the evolving wet paint film on the non-conductive plastic surface, which influences the local electric field and droplet deposition. The final film thickness distribution as well as transfer efficiency on the workpiece are compared with experimental results. The presented approach provides detailed insight into the governing physical mechanisms of electrostatic plastic coating and forms a robust basis for future reduced-order or data-driven models aimed at accelerated process optimization.

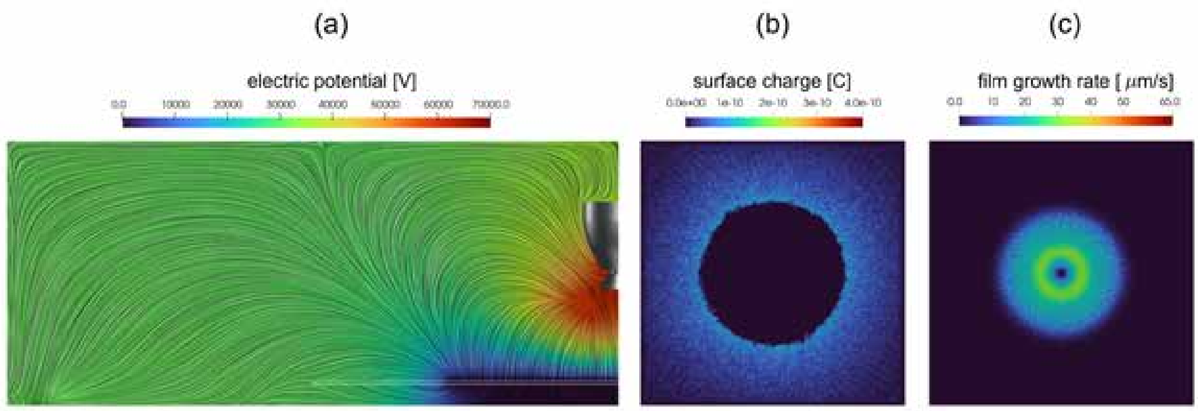


Figure 1. *Electric potential and electric field lines at central cross-section (a), surface charge (b) and film growth rate (c) on plastic substrate surface for a static atomizer.*

Numerical Study of Gas Injection Rate Effects on Viscous Fuel Atomization Using the Volume-of-Fluid Method

AliMarzooq*[1],Po-HanChen[1],HongIm[1]

[1]CleanEnergyResearchPlatform(CERP),PhysicalScienceandEngineering(PSE) Division,KingAbdullahUniversityofScienceandTechnology(KAUST),Thuwal,SaudiArabia
*Correspondingauthor:ali.marzooq@kaust.edu.sa

Abstract

Atomization of viscous fuels such as vacuum residue oil (VRO) is critical for entrained-flow gasification, but high viscosity suppresses interfacial deformation and delays breakup. Companion cold-flow experiments revealed a non-monotonic droplet-size response to increasing gas injection rate: droplet size decreased initially but increased again at the highest gas rate, despite higher gas velocity and Weber number. Here, large eddy simulation (LES) coupled with a geometric volume-of-fluid (VOF) method is used to isolate the effect of gas injection on primary atomization of a viscous VRO-like liquid. A controlled numerical operating path is considered in which the liquid injection velocity is fixed while the gas injection rate is increased. The LES–VOF framework, using reconstructed distance function (RDF) interface reconstruction and isoAdvector, resolves the evolving gas–liquid interface, ligament formation, and breakup. In contrast to the experiments, the simulations show monotonic aerodynamic refinement: increasing gas mass flow rate shortens the intact core, produces finer ligaments, and decreases droplet-size metrics from Case A to Case C. These results indicate that the experimentally observed non-monotonic droplet-size trend is not caused by increased gas injection alone, but by coupled changes in the liquid-side operating condition and the associated breakup evolution.

Keywords: High-viscosity sprays, liquid fuel atomization, LES, VOF, AMR, entrained-flow gasifiers

TUESDAY

09:50 – 10:40

Industrial applications *Chair: Jérôme Bellettre*
Revealing Dispersion Dynamics to Advance CBRN Defense
A. van Wuijckhuijse

Reduced Order Assessment of Rain Droplet Impact, Wet Film Attenuation and Tip Speed Derating for Wind Turbine Leading Edge Erosion
S. Jerzembeck

Atomization: Applications II *Chair: Michael Beeston*
Atomization Characteristics of a Convergent-Orifice Fan Nozzle in a Crossflow
H. Wang, Y. Liu, G. Liu, Q. Zhang, J. Lu, H. Dong

Coupling Mechanism of Penetration and Atomization of an External-Mixing Nozzle for Airblast Jet in Crossflow
Q. Zhang, Y. Liu, G. Liu

Spray Dynamics II *Chair: Lydia Achelis*
Impinging jets interaction at the collision point under moderate flash-boiling conditions
Ł.J. Kapusta, H. Amaya, B. Kaźmierski

Impact of Nozzle Material and Surface Morphology on the Flash-Boiling Atomization of Liquid Ammonia
P. Augustin, E.C. Okafor, H. Axt, M. Kratz, C. Hinke, F. Fröde, H. Choi, H. Pitsch, H. Schmitz, L. Eich, W. Rohlfs, M. Reddemann

11:10 – 12:25

Atomization Fundamentals III *Chair: François-Xavier Demoulin*
An Experimental Study on the Impact of Liquid Film Thickness on the Spray Characteristics with Various Nozzle Axial Positions in a Hybrid Airblast Atomizer
W. Choi, Y. Seo, S. Kim, J. Lee

TUESDAY

Experimental and Numerical Investigation of Primary Atomisation Mechanisms in Effervescent Nozzles
P. Novosad, C. Pairetti, M. Maly, J. Hajek, J. Jedelsky, S. Zaleski

A Comprehensive Study of the Governing Factors and Mechanisms in Continuous Flow in Ultrasonic Atomization
W. Jin, J.J.C. Remmers, N.G. Deen, G. Finotello

Numerical methods for jets *Chair: Konstantina Vogiatzaki*
High-fidelity multi-scale simulations of liquid jet-in-crossflow in realistic test-rig geometry using adaptive tetrahedral grids
O. Jegou, M. Cailler, J. Leparoux, R. Mercier, V. Moureau, R. Janodet, D. Noel

Numerical characterization of a confined evaporating liquid jet in crossflow
M. Sior, D. Zuzio, J.-M. Senoner, V. Bodoc, O. Rouzaud, L. Richard, J.-L. Estivalezes

A Mixture Density Network Model for Droplet Velocity Statistics in Jet-in-Crossflow Atomisation
G. Tretola, K. Vogiatzaki

Spray Dynamics III *Chair: Marco Piras*
Experimental Study of High-Pressure Hydrogen Direct-Injection Jets in Engine-Like Conditions
A. Montanaro, D. Piazzullo, G. Meccariello, L. Allocca

Experimental Spray Analysis for Multi-Hole Atomizers relevant for Large Two-Stroke Marine Engines Combustion Systems
G. González Saiz, M. Bohnenblust, P. Süess, S. Wüthrich, B. von Rotz

Experimental Investigation of Transient Spray Penetration Characteristics of OME_1–Diesel 1 Blends Under Different Injection Pressures in a Constant-Volume Chamber
A. Qadri, S.K. Vankeswaram, D. Rrustemi, T. Megaritis, L. Ganippa

TUESDAY

14:00 – 15:15

Droplet-wall interaction II *Chair: Guillaume Castanet*

The development of a continuum-level volume-of-fluid approach to model frost deposition on a cryogenic temperature flat plate
H. Lunn, K. Vogiatzaki

Droplet impact and evaporation on thin, porous coatings: delayed imbibition and enhanced heat transfer
T. Potaufeux, O. Caballina, J. Palumbo, M. Hosseinzadeh, S. Chandra, G. Castanet

Droplets bouncing on liquid surfaces: effect of surface tension
Y. Stern, I. Frog, V. Multanen, M. Hananovitz, A. Vinod, R. Tadmor

Experimental methods for droplets & sprays III *Chair: Ophélie Caballina*

Advanced Differential Infrared Thermography for Transcritical Spray Analysis Using a Reconstruction Algorithm
M. Son, L. Zigan, T. Sander

Thermometry in Transcritical Model Fuels Using Two-Color Laser-Induced Fluorescence
A. Schaumäker, L. Zigan

Error Investigation of a Monte-Carlo Simulation based Scattering Correction for Diffuse Back Illumination Extinction Imaging (DBI-EI)
F. Kohlmann, B. Lehnert, E. Berrocal, M. Wensing

Interfacial and multiphase physics II *Chair: Tali Bar-Kohany*

A Novel Non-Intrusive Method for Detecting CHF during Rapid Transients: Application to Spray and Immersion Cooling
T. Bar-Kohany, D. Biton, A. Koyfman, M. Vaknin, G. Ziskind

TUESDAY

A Theoretical and Numerical Framework to Predict the Onset of Nucleate Boiling during Rapid Depressurization
O. Magen, T. Bar-Kohany, N. Banerjee, M. Marengo, Y. Kozak

Sub-model and fluids sensitivity of two-fluid boiling predictions
G. Tretola, K. Vogiatzaki

Revealing Dispersion Dynamics to Advance CBRN Defense

Arjan van Wuijckhuijse*[1]

[1]CBRN Protection, TNO Defence Safety and Security, TNO, The Netherlands
*Corresponding author: arjan.vanwuijckhuijse@tno.nl

Abstract
From the ongoing conflict in Ukraine to the 2018 Salisbury Novichok attack and the 1995 Tokyo subway sarin incident, these events highlight that threats are often invisible, counterintuitive, and unconstrained by traditional battlefields. Whether involving chemical, biological, radiological or nuclear (CBRN) agents or other emerging technologies and environmental hazards, the core challenge remains: how do we protect people in environments where risks are complex, evolving, and not always immediately seen?
For many CBRN hazards, agent behavior in real-world settings is far from straightforward. Often dispersed as liquids or fine droplets, the fate of the agent, how it travels, how it transforms, and interacts with airflow, surfaces, and human activity must be understood to develop effective defense capabilities. To structure this complexity, CBRN exposure can be divided into three interconnected elements: source, pathway, and exposure. Exposure outcomes can vary significantly depending on the adversary's capabilities and intended method of dissemination. To support such variability, standardized experimental and theoretical test and evaluation procedures for physical protection and detection systems have been developed and are currently applied. Accurate exposure assessments require detailed characterization of the source term, such as droplet/particle size distributions, as well as the fate and transformation of these particles after dispersion. The experimental measurements and theoretical approaches used to describe both the source and transport pathway will be discussed to address actual exposure conditions.
Keywords: CBRN defence; aerosol dispersion; droplet size distribution; exposure assessment

Reduced Order Assessment of Rain Droplet Impact, Wet Film Attenuation and Tip Speed Derating for Wind Turbine Leading Edge Erosion

S. Jerzembeck*

Faculty Sustainable Engineering, Hamburg University of Applied Sciences, Germany
*Corresponding author: sven.jerzembeck@haw-hamburg.de

Abstract

Leading edge erosion of wind turbine blades is driven by repeated high velocity impacts of rain droplets, but the severity of a rain event is not determined by impact velocity alone [1,2]. Droplet inertia, aerodynamic filtering close to the stagnation region, droplet deformation or breakup, the presence of a liquid surface film and operational tip speed all affect the effective impact load. This paper presents a compact reduced order workflow intended for early stage screening of erosion relevant operating states. The model combines a Stokes Number based collision efficiency estimate, a Taylor Analogy Breakup indicator for large droplets [3], a wet film attenuation factor for water hammer pressure [4], and a Miner type cumulative damage measure [5] using a Springer style fatigue law [6]. The calculation is applied to a representative outer blade section at nominal tip speeds between 70 and 90 m/s. The results indicate that medium sized droplets around 1.5–2.5 mm dominate the screened impact risk, while very small droplets are aerodynamically deflected and very large droplets may enter a breakup limited regime before impact. The wet film calculation further suggests that even sub millimetre liquid films can reduce peak pressure substantially. Finally, tip speed derating during intense precipitation reduces the relative damage rate disproportionately compared with the instantaneous power reduction. The contribution is therefore not a replacement for validated CFD/FE erosion prediction, but a transparent engineering screening method that links rain conditions, droplet impact physics and erosion safe operation in one computationally inexpensive framework.

Keywords: rain erosion; droplet impact, leading edge erosion, reduced order model

Atomization Characteristics of a Convergent-Orifice Fan Nozzle in a Crossflow

Hang Wang[1], Yuying LIU*[1], Guanghai LIU[1], Quan Zhang[1], Jiaxiang LU[1], Hangyu Dong[1]

[1]School of Energy and Power Engineering, Beihang University, China
*Corresponding author: yyliu@buaa.edu.cn

Abstract

Fan nozzles hold significant potential for application in aero-engine combustors owing to their wide spray coverage and fine droplet sizes. To investigate the control mechanisms of internal nozzle geometry on the spray characteristics of a fan liquid sheet, a convergent-orifice fan nozzle was designed in this study. Using purified water as the working fluid, experiments were conducted at ambient temperature and pressure with a crossflow Mach number of 0.15 and initial jet-to-crossflow momentum flux ratio ranging from 5 to 30. High-speed backlight shadowgraphy and Particle Image Velocimetry (PIV) were employed to systematically compare the atomization characteristics of the convergent-orifice fan nozzle with those of a typical fan nozzle. The results indicate that, compared to the typical fan nozzle, the convergent-orifice fan nozzle forms a more intact liquid sheet at the exit, and its atomization and breakup process is rapid and efficient. The spray penetration depth is reduced, whereas the spanwise boundary is significantly broadened. A trajectory inflection point appears, and a localized low-velocity recirculation zone forms on the inner downstream side. Therefore, the converging flow passage design can effectively regulate the initial momentum distribution of the fan liquid sheet, thereby modulating the primary and secondary atomization processes.

Keywords: Afterburner; Atomization; Fan nozzle; Liquid film; PIV.

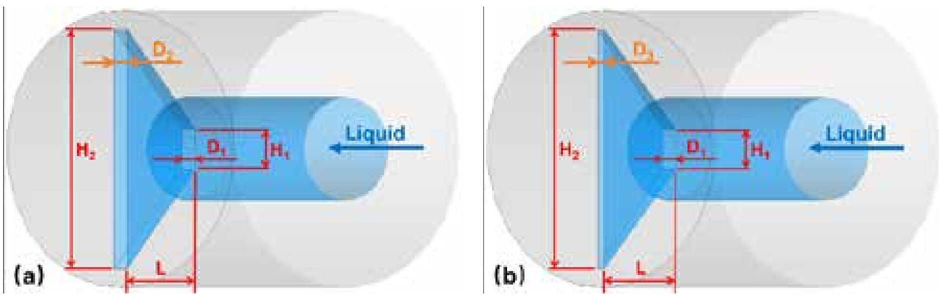


Figure 1. Schematic of nozzle configuration and key parameters.
(a) Typical fan nozzle. (b) Convergent-orifice fan nozzle.

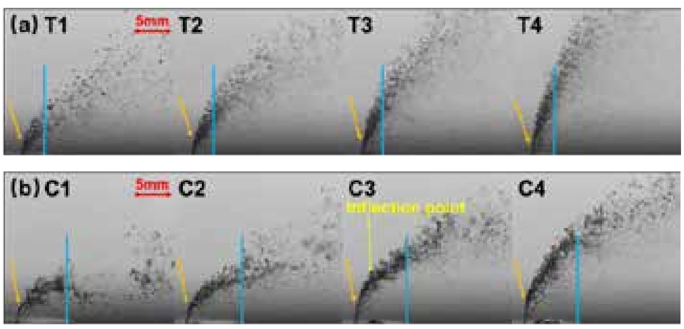


Figure 2. Shadowgraph image of the spray in the streamwise plane. (a) TFN. (b) COFN.

Coupling Mechanism of Penetration and Atomization of an External-Mixing Nozzle for Airblast Jet in Crossflow

ZHANG QUAN[1], LIU YUYING*[1], LIU GUANGHAI[1]
[1]BEIHANG University, School of Energy and Power Engineering BEIJING
*Corresponding author: yyliu@buaa.edu.cn

Abstract

Airblast jet in crossflow (AJIC) fuel injection offers exceptional atomization and penetration for aero-engine combustors. The coupling effects of the auxiliary airflow and the crossflow during jet atomization of an external-mixing coaxial airblast nozzle are investigated experimentally using high-speed photography analysis under conditions including crossflow Mach numbers of 0.1–0.2, 325 K, liquid-to-air momentum flux ratios q of 10–30, and air-to-liquid mass flow rate ratios (ALR) of 0–1.25. Results show that the dominant breakup mechanism is determined by the relative magnitude between the crossflow Weber number ($\mathrm{We_g}$) and the auxiliary airflow Weber number ($\mathrm{We_{ag}}$). When $\mathrm{We_g} > \mathrm{We_{ag}}$, the atomization is dominated by crossflow, characterized by surface stripping and column breakup, with low-frequency large-scale wake vortex shedding in the jet. At $\mathrm{We_g} \approx \mathrm{We_{ag}}$, the initial atomization of the liquid jet shifts to be dominated by the auxiliary airflow, and the liquid column oscillation transforms into helical waves induced by Rayleigh instability. At $\mathrm{We_g} < \mathrm{We_{ag}}$, auxiliary airflow fully controls breakup: helical oscillations with multi-frequency fine vortex chains occur at 15 <$\mathrm{We_{ag}}$< 60, while fibrous rapid near-nozzle breakup appears at $\mathrm{We_{ag}}$ > 60. The penetration depth is mainly controlled by ALR. When ALR > 0.5, the aerodynamic shielding effect is significantly enhanced, which greatly increases the penetration depth by maintaining high axial momentum

Keywords: Airblast jet in crossflow, External-Mixing Nozzle, Atomization

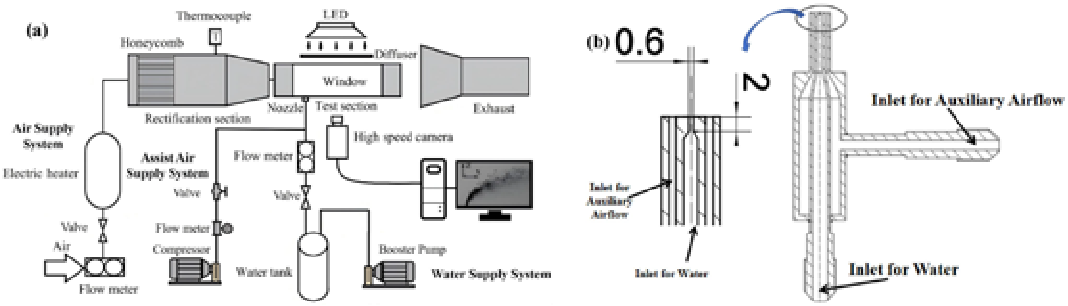
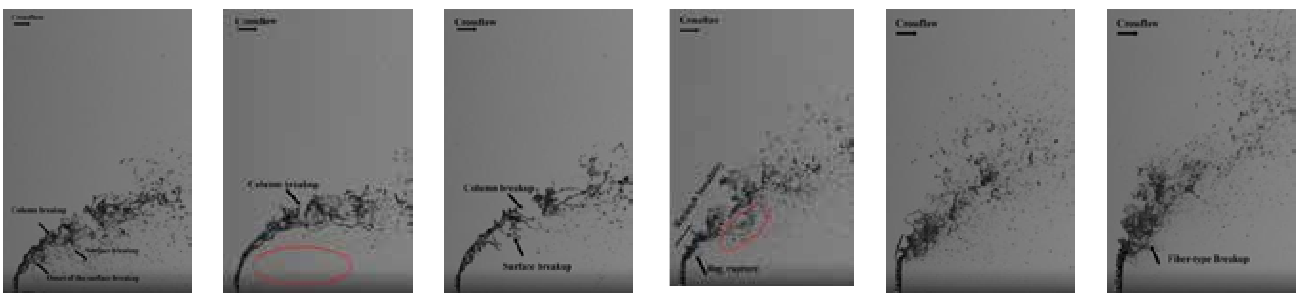


Figure 1 *Schematic diagram of the test section and schematic diagram of the coaxial airblast nozzle*



(a)ALR=0	(b)ALR=0.25	(c)ALR=0.5 q=10	(d)ALR=0.75	(e)ALR=1	(f)ALR=1.25
$\mathrm{We_g}$=37.67	$\mathrm{We_g}$=37.67	$\mathrm{We_g}$=37.67	$\mathrm{We_g}$=37.67	$\mathrm{We_g}$=37.67	$\mathrm{We_g}$=37.67
$\mathrm{We_{ag}}$=0	$\mathrm{We_{ag}}$=2.49	$\mathrm{We_{ag}}$=13.79	$\mathrm{We_{ag}}$=34.26	$\mathrm{We_{ag}}$=63.79	$\mathrm{We_{ag}}$=102.5

Figure 2 Comparison of transient images under different ALR at 0.15ma

Impinging jets interaction at the collision point under moderate flash-boiling conditions

Łukasz Jan Kapusta*[1], Hernan Amaya[1], Bartosz Kaźmierski[1]

[1]Faculty of Power and Aeronautical Engineering, Warsaw University of Technology, Poland
*Corresponding author: lukasz.kapusta@pw.edu.pl

Abstract
Impinging-jet atomisation has been demonstrated to be an effective method for enhancing liquid-jet break-up and promoting smaller droplet formation. However, such effects are expected only when the jets travel along the nozzle axes and collide precisely at the impingement point. Only then is the momentum exchange between the jets efficient. When the liquid is injected into an environment with ambient pressure below its vapour pressure, flash boiling may occur, substantially altering the jets' behaviour. Under such conditions, the liquid streams may begin to oscillate or even break up into ligaments and droplets. Instead of the collision of the intact streams travelling along the nozzles' axes, an interaction of clouds of ligaments and droplets distributed over a larger volume will be observed. Previous studies have shown that post-impingement spray structures are considerably altered even at moderate superheating. However, it remains unclear how moderate flash boiling influences the collision zone. The present study aims to address this gap by examining the liquid structures produced by two colliding liquid jets in the collision zone under moderate flash-boiling conditions. The results indicated that the flash boiling influences the interaction already in the collision zone, but does not change it entirely. At low injection velocities, relatively smooth, stable jets tend to corrugate under flash-boiling conditions, thereby accelerating the liquid detachment from the liquid sheet. It should be noted that the observed changes may be partially associated with changes in surface tension and viscosity, as the flash boiling was achieved by increasing the injection temperature.
Keywords: impinging jets, flashing jets, flash boiling, rocket injector, propellant injection

Impact of Nozzle Material and Surface Morphology on the Flash-Boiling Atomization of Liquid Ammonia

PeterAugustin*[1],EkenechukwuC.Okafor[2],HenryAxt[3],MartinKratz[3],ChristianHinke[3], ManuelReddemann[1]
FabianFröde[4],HongsukChoi[4],HeinzPitsch[4],HenrikSchmitz[1],LukasEich[1],WilkoRohlfs[1],

[1]InstituteofHeatandMassTransfer,RWTHAachenUniversity,Germany
[2]DepartmentofMechanicalEngineering,KyushuUniversity,Japan
[3]ChairforLaserTechnology,RWTHAachenUniversity,Germany
[4]InstituteforCombustionTechnology,RWTHAachenUniversity,Germany
*Correspondingauthor:augustin@wsa.rwth-aachen.de

Abstract

Liquid ammonia (LNH3) is a candidate for carbon-free energy systems, yet its high saturation pressure may induce complex flash-boiling dynamics upon injection that remain challenging to predict. This study characterizes the transition between aerodynamically and thermodynamically dominated atomization behavior of ultra-short orifices (L/D = 2.5) with a diameter of 100 µm. We investigate the impact of surface morphology and thermal properties on the onset of phase change with two sets of nozzles: Helical laser-drilled stainless-steel nozzles and fused silica nozzles fabricated via selective laser-induced etching (SLE), which yield optically accessible and precise geometries. Injection pressures were varied from 7.6 to 14 bar to evaluate the near-Rayleigh and first wind-induced breakup regimes while maintaining superheat ratios (Rp) between 3 and 8. The results reveal a distinct stochastic bifurcation in spray morphology for the L/D = 2.5 geometry, wherein identical hydraulic and thermodynamic boundary conditions yield fundamentally different atomization states: a stable Rayleigh-type jet case and a fully flashing plume case. This switch in spray morphology is accompanied by a discontinuous 20% reduction in mass flow from the Rayleigh-type case to the flashing plume case, indicating the onset of internal two-phase choked flow. We observe that the persistence of the flashing state during cooling is governed by boiling hysteresis, which could be attributed to the activation of pre-existing gaseous nuclei entrapped within microscopic surface irregularities. These findings highlight the critical role of thermal history in predicting the performance of flexible LNH3 injection systems, providing a deterministic baseline for stabilizing ammonia injection systems.

Keywords: Liquid Ammonia, Flash-Boiling, SLE

An Experimental Study on the Impact of Liquid Film Thickness on the Spray Characteristics with Various Nozzle Axial Positions in a Hybrid Airblast Atomizer

Won Choi[1], Yechan Seo[1], Sigyu Kim[1], Jeekeun Lee*[2]

[1]Graduate School of Mechanical Aerospace Electrical Convergence Engineering., Jeonbuk National University, Korea
[2]Dept. of Mechanical Aerospace Electrical Convergence Eng., Jeonbuk National University, Korea
*Corresponding author: leejk@jbnu.ac.kr

Abstract
The hybrid airblast nozzles (HAN), an assembly nozzle-swirler, is widely applied in modern aerospace gas turbine combustor. This study experimentally investigated the effects of liquid film thickness on the spray characteristics by varying the nozzle axial position. Due to the small and curved geometry of the swirler, direct analysis of liquid film thickness was challenged. Instead, the liquid film thickness was qualitatively measured using an acrylic flat plate without swirling airflow. Subsequently, droplets size and velocity were measured simultaneously utilizing a Phase Doppler Particle Analyzer (PDPA). When the spray impinged on the swirler, most droplets were splashed or accumulated, forming a liquid rim at the Venturi tip. The liquid film thickness obtained via image analysis was compared with prediction result based on the liquid jet. As the non-dimensional distance (h*) increased, friction between the swirling airflow passing through inner swirler and liquid film was increased because of the larger wetted surface area. This phenomenon reduced the angular momentum of the swirling air flow, leading to decreased shear stress in the shear layer. In contrast, a widened spray spatial distribution and a large recirculation were formed because of the thickened liquid rim and the enhanced effect of the swirling air flow through the outer swirler. Consequently, as the liquid film thickness increased, the small droplets within the shear layer increased, while the atomization performance in the outer spray region significantly decreased.
Keywords: Gas turbine combustor, Hybrid airblast nozzle, Liquid film thickness, Spray atomization

Experimental and Numerical Investigation of Primary Atomisation Mechanisms in Effervescent Nozzles

PavelNovosad*[1],CesarPairetti[2],MilanMaly[1],JiriHajek[1],JanJedelsky[1],StephaneZaleski[2]

[1]EnergyInstitute,BrnoUniversityofTechnology,CzechRepublic
[2]L'InstitutJeanLeRondd'Alembert,SorbonneUniversity,France
*Correspondingauthor:pavel.novosad@vutbr.cz

Abstract

Primary breakup in effervescent atomisers is driven by the passage of gas–liquid structures through the discharge orifice, which causes a rapid generation of interfacial area. This study examines how bubble diameter and bubble–wall contact influence this process through a combined experimental and numerical investigation conducted on a see-through polymethyl methacrylate atomiser. Experimentally, high-speed back-lit imaging at 120 000 fps is employed in conjunction with a refractiveindex-matched ammonium thiocyanate solution to visualise bubble transit and jet response. Venacontracta separation was observed in all tested cases, preventing optical access to the interior of the orifice and limiting direct observation to events in which a bubble is not large enough to cause liquid attachment to the orifice wall. This attachment constitutes a qualitative bifurcation in spray behaviour: it produces pronounced jet disturbance, sometimes causing the liquid to attach to the discharge orifice walls, which can persist for up to 5 ms after bubble passage, whereas transit events without wall contact generate comparatively limited breakup. Numerically, three-dimensional Volume-of-Fluid simulations with Large Eddy Simulation are performed in ANSYS Fluent. The gas phase is modelled as a compressible ideal gas to capture the rapid expansion at the orifice exit; the liquid is treated as incompressible. Predicted instability frequencies show close agreement with experimental measurements across all tested discharge pressures. Kelvin–Helmholtz instability is identified as the dominant mechanism driving gas-jet pinch-off during bubble transit. The pinch-off frequency scales inversely with bubble diameter, consistent with the longer time required for the instability to bridge a wider gas column. For larger bubbles, the slower pinch-off allows the intervening liquid sheet to thin further, increasing its susceptibility to secondary atomisation. A proper description of this mechanism could allow for increased atomisation efficiency and a more stable spray.

Keywords: Atomisation, Kelvin-Helmholtz instability, primary breakup, Volume of fluid

A Comprehensive Study of the Governing Factors and Mechanisms in Continuous Flow in Ultrasonic Atomization

WenJin*[1],JorisJ.C.Remmers[2,4],NielsG.Deen[1,3],GiuliaFinotello[1,3]

[1]Power&FlowGroup,EindhovenUniversityofTechnology,Eindhoven,TheNetherlands
[2]MechanicsofMaterialsGroup,EindhovenUniversityofTechnology,Eindhoven,The Netherlands
[3]EindhovenInstituteforRenewableEnergySystems(EIRES),EindhovenUniversityof Technology,TheNetherlands
[4]EindhovenArtificialIntelligenceSystemsInstitute(EAISI),EindhovenUniversityof Technology,TheNetherlands
*Correspondingauthor:w.jin@tue.nl

Abstract

Ultrasonic atomization is a promising energy-efficient method for producing high-quality metal powders for additive manufacturing. Unlike gas or water atomization, droplets are generated from a thin liquid film on an ultrasonically vibrating surface and rapidly solidify into powders. In this study, an ultrasonic transducer coupled with a controlled feeding system is used to atomize liquids under varying liquid film thicknesses, flow rates, and vibration amplitudes. High-speed imaging is employed to analyze spray characteristics. Results reveal that increasing liquid film thickness leads to larger droplets with a broader size distribution due to enhanced capillary wave amplitude and stronger cavitation effects. In contrast, varying the flow rate at a fixed film thickness has little effect on droplet size, confirming that liquid film thickness is the primary parameter governing droplet formation. These results highlight the potential of ultrasonic atomization as a sustainable route for metal powder production.

Keywords: Ultrasonic atomization, Droplet formation, Droplet size distribution, Liquid film thickness

High-fidelity multi-scale simulations of liquid jet-in-crossflow in realistic test-rig geometry using adaptive tetrahedral grids

OanigJegou[1],MelodyCailler[2],JulienLeparoux[2],RenaudMercier[2],VincentMoureau[3],
RomainJanodet*[1],DamienNoel[1]

[1]SafranAircraftEngines,Combustiondesignoffice,Villaroche,France
[2]SafranTech,DigitalSciencesandTechnologiesDepartment,SafranTech,78117,
Chateaufort,France
[3]CORIA,CNRS,NormandieUniv,UNIROUEN,INSARouen,76000Rouen,France
*Correspondingauthor:romain.janodet@safrangroup.com

Abstract
In the past and because of the harsh conditions, the design of afterburner systems was carried out performing tests. Over the years and with the progress of high-performance computing, numerical simulation has become another major tool to rely on for designing such systems, provided the numerical methodologies are thoroughly validated. A recent experimental campaign was performed at ambient temperature and pressure to study state-ofthe-art fuel injection technologies for afterburners, in particular spray bars. These technologies mostly inject fuel jet into a high-velocity air crossflow. In this case, fuel jet penetration and spray drop size distributions have a direct impact on flame stabilization and combustion performances. Better understanding the mechanisms that control atomization and acquiring quantitative data are therefore essential for the development of innovative and optimized technologies. The goal of this work is to validate a high-fidelity numerical methodology to simulate jet-in-crossflow fuel injection within the realistic test-rig geometry. To this aim, the massively parallel spray solver of the YALES2 library is employed. This solver features the accurate conservative level-set method for interface capturing, the ghost-fluid method for surface-tension modelling, dynamic mesh adaptation around the liquid-gas interface, and a dynamic Euler-to-Lagrange conversion. The numerically obtained spray characteristics are compared to the experimental data and existing correlations of the literature for two operating points, characterized by the momentum flux ratio and the gaseous Weber number. This comparison is done in particular for (i) the jet penetration, (ii) transverse fuel fluxes, and (iii) drop distributions. (i) Satisfactory quantitative results are obtained for the jet penetration using both Eulerian and Lagrangian phases; (ii) The transverse Lagrangian fuel distribution displays that the counter-rotatingvortex-pair structure of the spray is retrieved; (iii) The droplet size distributions show that for the first operating point the grid resolution is not sufficient for drop size predictions without a secondary atomization model, while the resolution seems sufficient for the second operating point but a concentration effect in drop size is observed.
Keywords: Atomization; CFD; Jet-in-crossflow; Conservative Level Set; AMR; Euler-Lagrange coupling

Numerical characterization of a confined evaporating liquid jet in crossflow

MaxenceSior*[1],DavideZuzio[1],Jean-MathieuSenoner[1],VirgineIBodoc[1],OlivierRouzaud[1], LilouRichard[1],Jean-LucEstivalezes[1]

[1]DepartmentofMulti-PhysicsforEnergetic,ONERAToulouse,France
*Correspondingauthor:maxence.sior@onera.fr

Abstract

Aeronautical injectors produce fuel sprays by an assisted atomization process. The fuel enters the chamber as a liquid jet or sheet and breaks up into a cloud of fine droplets under the shearing action of the airflow. The evaporation of the liquid droplets then produces the gaseous mixture of oxidant and fuel required to feed the flame. As industrial large eddy simulations cannot afford to fully resolve the atomization process, appropriate modelling strategies are required. In particular, distinct numerical approaches must be employed to represent the dense flow near the injection and the dispersed cloud of droplets respectively. The ONERA CEDRE CFD software uses such a multi-model approach for the numerical simulation of fuel injection. This approach has been tested in previous work on a Liquid Jet of water In gaseous Cross-Flow (LJICF) in a confined channel, a configuration representative of multi-point liquid fuel injection systems. The jet was studied both in steady state and under imposed acoustic forcing. The numerical results obtained with this approach were very encouraging. In this paper, the approach is applied to a similar liquid jet in gaseous cross flow. The present study will extend the methodology to evaporating conditions in order to assess the ability of the approach to describe the liquid vapor distribution downstream the liquid column. Preliminary results showed that the dense phase approach can capture the main liquid body of the jet, in terms of both its trajectory and its effect on the gas stream. When coupled with the dispersed phase approach, this makes it possible to efficiently simulate the production of the droplet cloud by the primary atomisation of the jet, and later its evaporation. This approach is well adapted to study the interaction of the evaporating spray with an acoustic forcing, simulating a flow instability.

Keywords: liquid jet in crossflow, atomization, mutli-scale, evaporation

A Mixture Density Network Model for Droplet Velocity Statistics in Jet-in-Crossflow Atomisation

GiovanniTretola*[1],KonstantinaVogiatzaki[1]

[1]DepartmentofEngineeringScience,UniversityofOxford,UnitedKingdom
*Correspondingauthor:giovanni.tretola@eng.ox.ac.uk

Abstract

Predicting droplet velocity statistics in liquid jets in crossflow is important for modelling spray penetration, residence time, dispersion and downstream phase interaction. However, high-fidelity simulations remain too expensive for rapid parametric exploration, while scalar descriptors cannot fully represent the variability of the droplet population. In this work, we develop a Mixture Density Network (MDN) surrogate to predict the full probability density function of liquid-droplet velocity magnitude from non-dimensional operating parameters. The training dataset is generated from Large Eddy Simulations of jet-in-crossflow atomisation using a stochastic-fields formulation. The database spans 27 operating conditions defined by Weber number We, momentum-flux ratio q and density ratio r_ρ , with 16 stochastic realisations for each case. Droplet velocity magnitudes are sampled in the dilute spray region and converted into binned probability density functions for training. A model-capacity study compares MDNs with $K = 1$, $K = 3$ and $K = 5$ Gaussian components. All models converge robustly with the stabilised loss formulation, while $K = 3$ provides the best balance between predictive capability, stability and parsimony. The selected MDN captures the dominant peak, global distribution width, tail decay and systematic shifts of the LES-derived velocity PDFs across the parameter space, demonstrating a compact uncertainty-aware surrogate for rapid prediction of spray velocity statistics.

Keywords: mixture density network, droplet velocity distribution, spray statistics, jet in crossflow

Experimental Study of High-Pressure Hydrogen Direct-Injection Jets in Engine-Like Conditions

A. Montanaro*, G. Meccariello, D. Piazzullo

STEMS, CNR Institute (Italy)
Napoli, Italy, 80125
*Corresponding author: alessandro.montanaro@stems.cnr.it

Abstract
The growing challenges of global energy shortages and greenhouse gas emissions highlight the need for sustainable energy alternatives. Hydrogen is increasingly recognized as a promising zero-carbon fuel for road transportation, especially when produced through renewable pathways such as solar-powered water electrolysis. Due to its distinctive thermo-physical properties, including low density, high diffusivity, and wide flammability limits, hydrogen significantly affects jet dynamics, turbulent mixing, and in-cylinder mixture formation. Understanding these processes is crucial for optimizing fuel injection strategies and injector design, thereby enhancing combustion efficiency, stability, and emissions performance.
Within this framework, the present study aims to deliver a comprehensive and systematic investigation of the behavior of a high-pressure hydrogen jet generated by a pintle-type outward-opening injector, operating under a wide range of injection pressures (up to 100 bar), ambient back-pressures, and temperatures representative of realistic engine operating conditions. The high-pressure hydrogen injection system was characterized both in terms of its mass-flow dynamics, using a measurement apparatus specifically engineered for gaseous fuels, and in terms of its jet-morphology evolution, assessed through cycle-resolved schlieren imaging conducted in a constant-volume vessel filled with nitrogen. The resulting schlieren datasets were subsequently subjected to offline post-processing using a dedicated image-analysis algorithm designed to enhance jet-boundary discrimination, suppress background noise, and extract key macroscopic descriptors of jet evolution, including penetration length, spreading angle, shock-cell structure, and overall plume topology.
Figure 1 depicts the influence of the injection pressure on jet morphology for the three ambient pressures at 1.48 ms after the SOI. The main consequence of the increase in injection pressure, keeping constant the injection duration and ambient pressure, is the increase in the total quantity injected per cycle. This led to a greater spread of the injected gas mainly in the radial direction rather than in the axial one as the jet images in figure show for each ambient pressure condition. Moreover, for each injection pressure, H_2 jets at higher ambient pressures experience a greater decrease in momentum due to higher aerodynamic drag forces. The net effect is less hydrogen dispersion into the environment, both axially and radially, resulting in an increase in jet overall density due to the compressibility of the gas.

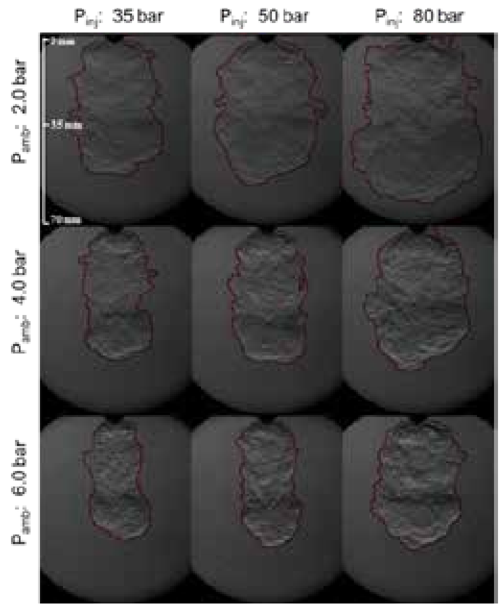


Figure 1. Injection and ambient pressure effects on H_2 on jet morphology.

Experimental Spray Analysis for Multi-Hole Atomizers relevant for Large Two-Stroke Marine Engines Combustion Systems

Gabriel González Saiz*, Martin Bohnenblust, Pascal Süess, Silas Wüthrich, Beat von Rotz

Applied Research and Measurements, Research & Development, WinGD Ltd., Switzerland
*Corresponding author: gabriel.gonzalezsaiz@wingd.com

Abstract

Spray formation in large low-speed two-stroke marine engine combustion systems is governed by peripherically located multi-hole atomizers that generate interacting liquid fuel plumes during the fuel admission process. These complex spray structures challenge conventional analysis methods, which are largely derived from simplified single-hole nozzle or centrally placed injection systems with symmetrically distributed orifice configurations. This work presents a post-processing methodology for the characterization of multi-hole fuel sprays representative of large marine combustion systems based on optical diagnostics. Experiments were conducted in an optically accessible, large-scale spray combustion chamber operated at engine-relevant pressure, temperature, and swirl conditions. High-speed Mie-scattering imaging was used to investigate spray development for both conventional marine diesel oil as well as methanol, covering inert and reactive experiments. A threshold-based contour detection approach is combined with a low-order spray regularization technique, enabling robust extraction of global spray parameters, including liquid penetration length and projected spray area, despite strong plume interaction and asymmetric spray morphologies. The methodology allows consistent tracking of spray evolution and provides physically interpretable metrics for complex multi-plume sprays. Results show the well-functioning and applicability to such spray types. Furthermore, they highlight distinct differences in spray coalescence, penetration behaviour, and evaporation characteristics between diesel and methanol, driven by their contrasting thermophysical properties. Methanol sprays exhibit reduced plume coalescence and enhanced evaporation, emphasizing the need to adapt classical spray analysis definitions for alternative future fuel types. The proposed methodology allows for improved spray characterisation, thereby supporting the development and validation of combustion systems for next-generation marine engines.

Keywords: Mie Scattering, image processing, spray, multi-hole, diesel, methanol, marine 2-stroke engine

Experimental Investigation of Transient Spray Penetration Characteristics of OME_1–Diesel 1 Blends Under Different Injection Pressures in a Constant-Volume Chamber

Ahmed Qadri*, S.K.Vankeswaram, Dardan Rrustemi, Thanos Megaritis, Lionel Ganippa

Department of Mechanical and Aerospace Engineering, Brunel University London, Uxbridge, UB8 3PH, UK

*Corresponding author: Ahmed.Qadri@brunel.ac.uk

Abstract

Near-nozzle transient spray development of neat diesel (D100) and a 25 vol% OME_1 blend (D75–O25) was investigated experimentally in an optically accessible constant-volume combustion chamber (CVCC) under non-reactive nitrogen conditions. High-speed shadowgraph imaging at 20,000 fps was used to resolve spray tip penetration, cone angle, and projected spray area over 0–500 µs after start of injection, for injection pressures of 70, 90, and 110 MPa and ambient pressures of 3.0 and 4.5 MPa the D75–O25 blend exhibits enhanced penetration, wider cone angles, and larger projected spray area relative to D100 during the early transient phase. These effects are most pronounced at 70 MPa and become less pronounced toward 110 MPa, as injection momentum increasingly dominates spray development the results demonstrate that the influence of OME_1 blending on spray behaviour is concentrated within the early injection transient and is strongly dependent on the injection pressure regime, highlighting the critical role of fuel properties in near-nozzle dispersion under low-to-moderate momentum conditions.

Keywords: Near-nozzle spray, OME_1-Diesel blends, Fuel density, Injection pressure

The development of a continuum-level volume-of-fluid approach to model frost deposition on a cryogenic temperature flat plate

HilaryLunn[1],KonstantinaVogiatzaki*[1]

[1]ThermalPropulsionSystemsResearchGroup,CryogenicFluidDynamicsLab,Department ofEngineeringScience,UniversityofOxford,UK
*Correspondingauthor:konstantina.vogiatzaki@eng.ox.ac.uk

Abstract

Solidification is a fundamental phase-change process encountered in a wide range of engineering applications, including cryogenic storage, aerospace systems, thermal management and icing phenomena. Depending on the application, solidification may occur under conventional low-temperature conditions or under ultra-low and cryogenic-temperature conditions, where the relevant heat and mass transfer processes can differ substantially. We are developing a numerical solver capable of modeling solidification across both regimes within a unified interfacial-flow framework. The present paper reports the first stages of development and assessment of this solver, focusing on the incorporation of solidification into a volume-of-fluid (VOF)-based OpenFOAM® framework. The model solves a single set of governing equations throughout the computational domain and introduces solidification through source terms in the VOF formulation. A Darcy-type momentum source term is used to represent the solidifying region as a porous solid. The solver is developed by adapting an existing single-component evaporation– condensation model into a solidification model. A benchmark configuration based on Byun et al. [1], which will support future cryogenic frosting simulations, is described, and preliminary simulations using the new solver are presented for a similar setup. The effects of inlet velocity, inlet temperature, and wall temperature are analyzed, and future developments required for fully coupled cryogenic solidification and frosting simulations are discussed.

Keywords: Solidification, cryogenics, multiphase, desublimation, frosting, phase-change, OpenFOAM

Impact and Evaporation of Water Droplets on a Wicking Surface

Thomas Potaufeux[1], Ophélie Caballina[1], Josh Palumbo[2], Mohammad Hosseinzadeh[2], Sanjeev Chandra[2], Guillaume Castanet[1]

[1]Université de Lorraine, Nancy, France
[2]Centre for Advanced Coating Technologies, University of Toronto, Toronto, Canada

Abstract

Evaporative cooling on rough or porous surfaces is a promising approach for enhancing heat transfer under high heat flux conditions, due to the combined effects of thin-film evaporation and capillary wicking. In this context, understanding droplet impact and evaporation dynamics on porous coatings is essential for the design of advanced thermal management systems.

In this study, the impact and evaporation of millimetric water droplets on heated porous aluminum coatings are experimentally investigated and compared to smooth surfaces. Porous aluminum layers are deposited by wire-arc spraying onto a sapphire substrate (Figure 1a). A black paint layer is applied prior to coating to ensure high surface emissivity. Infrared thermography is used to measure the transient surface temperature, and an inverse heat conduction method is employed to quantify the local heat transfer between the droplet and the substrate (Figure 1b). High-speed shadowgraphy complements the thermal measurements by providing detailed visualization of droplet spreading and imbibition. Experiments are conducted over a surface temperature range of 80-150 °C and for various Weber numbers.

The results show that the porous coating strongly modifies the droplet dynamics. Capillary wicking promotes liquid penetration into the porous structure, significantly reducing the evaporation time compared to the non-porous surface. A competition between wicking-driven spreading and evaporation is observed, depending on surface temperature (Figure 1c). A model describing the temporal evolution of the wetted radius is proposed and captures the main experimental trends. While the maximum wetted area is larger on the porous surface, the total exchanged energy remains comparable for both configurations.

The Weber number has a limited influence on the overall behavior, as the impact phase occurs on a millisecond time scale, whereas the dominant wicking-evaporation processes develop over several seconds. Despite the enlarged wetted area, accelerated evaporation ensures significantly higher mean heat fluxes on porous surfaces, increasing with temperature. These results demonstrate the strong potential of porous aluminum coatings for enhanced evaporative cooling applications.

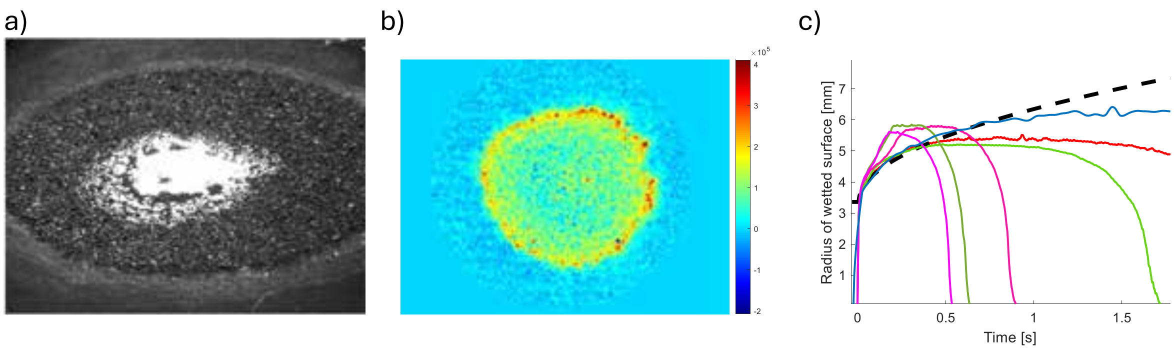


Figure 1: Illustration of droplet impact on a porous surface. (a) Shadowgraph images highlighting liquid imbibition into the porous coating. (b) Reconstructed local heat flux distribution. (c) Time evolution of the wetted radius at different surface temperatures.

Droplets bouncing on liquid surfaces: effect of surface tension

Yotam Stern, Ivgeni Frog, Victor Multanen, Mor Hananovitz, Appu Vinod, Rafael Tadmor*

Dept. of Mechanical Engineering, Ben Gurion University of the Negev, Beer Sheva, Israel

Abstract

While drops impinging on surfaces, a biologically prevalent phenomenon, has been extensively researched, the resultant bouncing probability is yet to be quantified. We derive a model capturing the probability of a drop to bounce on a liquid surface, obtaining a dependence on three ratios: inertia to capillarity (We number), inertia-capillarity to viscosity (La number), and drop radius to liquid layer depth. Our theory is backed by laboratory experiments, with maximal bouncing at We ≈ 1, radius-depth ratios of ≈ 1/5, and $La^{1/4} < 100$. As La increases, the probability decreases in a sigmoidal fashion.

Advanced Differential Infrared Thermography for Transcritical Spray Analysis Using a Reconstruction Algorithm

Min Son*[1], Lars Zigan[1], Tobias Sander[1]

[1] Institute of Thermodynamics, Universität der Bundeswehr München, Germany
*Corresponding author: Min.Son@unibw.de

Abstract

Investigation of spray behavior under transcritical conditions is crucial for understanding atomization, evaporation, and mixing phenomena in high-pressure combustion. Modern high-pressure combustion systems frequently operate under conditions in which the ambient fluid is supercritical while the injected fuel initially remains subcritical, resulting in complex transcritical behavior. A central challenge in analyzing such sprays is identifying the local thermodynamic state and phase-transition behavior. Conventional spray imaging techniques, such as shadowgraphy, can effectively resolve qualitative density gradients, but they cannot directly provide quantitative temperature fields. In contrast, infrared (IR) imaging provides a non-intrusive means of measuring thermal radiation from the spray, which can be related to temperature through radiometric calibration.

In this study, a reconstruction algorithm based on the Radiative Transfer Equation (RTE), combined with Differential Infrared Thermography (DIT), is proposed to extract spatially resolved temperature profiles of sprays in a transcritical environment. DIT uses spray images acquired at two background temperatures to determine the line-of-sight integrated emissivity. Under the assumptions of axisymmetry, negligible scattering, and local thermodynamic equilibrium, the attenuation coefficient and local radiative source term are subsequently reconstructed. The reconstructed source term is then converted into the effective black-body spray radiance. Since both reconstruction steps are inherently ill-posed inverse problems, a multi-parameter regularization strategy is employed to obtain stable solutions.

Radiometric calibration is performed to relate the corrected camera signal to effective black-body radiance and subsequently to temperature, while accounting for the camera dark signal, chamber-related contributions, and transmission losses through the sapphire windows. The proposed algorithm is applied to transcritical spray measurements conducted on the Multi-diagnostic Spray Investigation Chamber (MuSIC) test bench at the Institute of Thermodynamics, Universität der Bundeswehr München. Cyclopentane (C_5H_{10}) is injected into heated nitrogen under chamber conditions above the critical point of cyclopentane.

Compared with conventional DIT, the reconstructed spray temperature profile obtained in this work shows improved agreement with numerical simulations. Where valid reconstruction is possible, the reconstructed centerline temperature is lower than the conventional DIT estimate and follows the numerical trend more closely, while conventional DIT produces higher centerline temperatures than both the reconstructed result and the numerical prediction. However, near-nozzle results remain uncertain because elastic scattering is neglected and may be incorrectly interpreted as absorptive attenuation. A pronounced downstream temperature fluctuation is observed near the phase-transition location predicted by the previous numerical study, although its physical origin requires further investigation. Additional investigations are required to fully characterize the phase-change behavior of transcritical sprays. Overall, the proposed approach provides a robust framework and an improved quantitative methodology for investigating evaporating sprays in a hot chamber, with direct relevance to high-performance combustion engines.

Keywords: Differential Infrared Thermography, Transcritical Sprays, Radiative Transfer Equation, Spray Temperature

Acknowledgements

This research is funded by dtec.bw – Digitalization and Technology Research Center of the Bundeswehr under the project MaST - Macro/Micro-simulation of Phase Decomposition in the Transcritical Regime. dtec.bw is funded by the European Union – NextGenerationEU.

Thermometry in Transcritical Model Fuels Using Two-Color Laser-Induced Fluorescence

Anton Schaumäker*[1], Lars Zigan[1]

[1]Institute of Thermodynamics, Universität der Bundeswehr München, Neubiberg, Germany
*Corresponding author: anton.schaumaeker@unibw.de

Abstract
Understanding transcritical spray processes is important for various technical systems operating under high-pressure and high-temperature conditions. Temperature measurements are crucial for analyzing phase-change behavior. In this work, temperature measurements in transcritical isooctane are conducted using two-color laser-induced fluorescence (2c-LIF). This measurement technique employs 1-methylnaphthalene (1-MN) as a tracer. To achieve different phases of the fluid, a cuvette is placed in a flow cell filled with nitrogen. The measurements are performed under a nitrogen atmosphere at a pressure of 3 MPa and at fluid temperatures of up to 779 K. For the supercritical regime, a temperaturedependent 2c-LIF calibration curve can be formulated, with the signal ratio increasing by approximately 8 % per 100 K. No such dependence is observed in the liquid phase or the transition region. The temperature sensitivity of the signal ratio is lower than that observed in previous measurements with leaner mixtures of vaporized isooctane in nitrogen, for which an increase of approximately 20 % per 100 K was determined. Further investigations will focus on measurements in droplets and sprays, different model fuels, and techniques to discriminate between the different phases.
Keywords: two-color LIF, thermometry, fluorescence spectroscopy, transcritical, 1-methylnaphthalene

Error Investigation of a Monte-Carlo Simulation based Scattering Correction for Diffuse Back Illumination Extinction Imaging (DBI-EI)

FlorianKohlmann*[1,2],BastianLehnert[1,2],EdouardBerrocal[2,3],MichaelWensing[1,2]

[1]ProfessorshipforFluidSystemsTechnology,Friedrich-Alexander-Universität Erlangen-Nürnberg,Germany
[2]SchoolinAdvancedOpticalTechnologies,Friedrich-Alexander-Universität Erlangen-Nürnberg,Germany
[3]DivisionofCombustionPhysics,DepartmentofPhysics,LundUniversity,Sweden
*Correspondingauthor:florian.kohlmann@fau.de

Abstract
In spray diagnostics, the liquid volume fraction (LVF) is a key parameter for characterizing transient sprays. Diffuse Back Illumination Extinction Imaging (DBI-EI) is an optical technique that, beyond providing two-dimensional spray parameters, enables quantitative tomographic measurements of the extinction coefficient. When droplet diameter information is available, the liquid volume fraction can be derived from these measurements. A major advantage of DBI-EI is both the diffuse light source character that significantly reduces beamsteering effects under harsh, engine-like conditions and the detection of scattered photons that enables quantitative measurements in dense spray regions. However, this scattered light introduces a systematic measurement bias that must be corrected to obtain reliable quantitative results. Recently, a correction approach based on Monte-Carlo photon scattering simulations has been developed. This approach applies a correction function to the measured data to compensate for scattering effects. The method uses a simplified scattering scenario compared to the complex structures of a real, transient spray and therefore relies on several assumptions regarding droplet size (distribution), refractive index and geometry of the scattering medium. In this study, the influence of different underlying assumptions on the resulting correction function is systematically investigated. The objective is to identify the dominant parameters that affect the correction and to provide a realistic assessment of the uncertainties introduced by the method.
Keywords: Spray Diagnostics, Extinction Imaging, DBI-EI, Multi Scattering, Extinction Coefficient

A Novel Non-Intrusive Method for Detecting CHF during Rapid Transients: Application to Spray and Immersion Cooling

Tali Bar-Kohany*[1,2], Doron Biton[2,3], Alex Koyfman[2], Matan Vaknin[2], Gennady Zisking[3]

[1]School of Mechanical Engineering, Tel Aviv University, Tel Aviv, Israel
[2]Department of Mechanical Engineering, NRCN, Israel
[3]Department of Mechanical Engineering, Ben-Gurion University of the Negev, Israel

*Corresponding author: talibk@tauex.tau.ac.il

Abstract

The escalating heat fluxes in compact, high-power electronic devices, with local hotspots projected to reach 5 kW/cm^2, necessitate advanced phase-change strategies, such as high-flux spray and immersion cooling, to prevent catastrophic device failure. Under these regimes, temperature escalation rates can reach the order of 10^6 K/s. While traditional research focuses on the Critical Heat Flux (CHF) value, there is a significant lack of data regarding the corresponding wall temperature (T_{CHF}) during these extreme transients.

This study investigates the dependence of the Onset of Nucleate Boiling (ONB) and CHF on high heating rates using a platinum micro-heater submerged in HFE-7100. Although conducted in a pool environment, the extreme heating rates utilized simulate the rapid local temperature fluctuations experienced during the droplet impingement cycle in spray cooling, often leading to film boiling. A novel mathematical criterion for identifying CHF is proposed, based on the maximum temperature-rise rate. This non-intrusive method allows for robust CHF identification via electrical measurements alone, which was successfully validated against high-speed optical visualization at 200,000 fps.

The results reveal a clear transition from Hydrodynamic Instability (HI) to thermally dominated Heterogeneous Spontaneous Nucleation (HSN) mechanisms. Within the HSN regime, the CHF temperature initially rises with the heating rate but drops sharply as it approaches the fluid's critical temperature. Furthermore, the time interval between ONB and CHF events decreases rapidly as heating rates increase.

Notably, rapid heating rates significantly enhance the boiling limit, yielding CHF values over 100 times higher than Zuber's classic steady-state predictions. These findings provide a physically sound framework for understanding the boiling crisis under extreme transient conditions, offering critical guidance for optimization of high-performance thermal management systems.

Keywords: Transient boiling, Rapid Heating, Nucleation, CHF

A Theoretical and Numerical Framework to Predict the Onset of Nucleate Boiling during Rapid Depressurization

Omry Magen*[1], Yoram Kozak*[1], Nilojendu Banerjee[2], Marco Marengo[2],Tali Bar-Kohany[1,3]

[1] School of Mechanical Engineering, Tel Aviv University, Tel Aviv, Israel
[2] Department of Civil Engineering and Architecture, University of Pavia, Pavia, Italy
[3] Department of Mechanical Engineering, NRCN, Israel
*Corresponding author: omrymagen1@tauex.tau.ac.il

Abstract

A thermodynamic model recently proposed to predict the onset of nucleate boiling (ONB) during rapid depressurization is here implemented in a fully compressible Godunov-type flow solver. The formulation augments the Euler equations with transport equations for bubble number density and partial vapor mass, enabling metastable-liquid evolution and subsequent recovery toward equilibrium to be captured within a unified framework. The present study is intentionally restricted to a 1D inviscid configuration, with no-slip and local mechanical and thermal equilibrium assumptions, a monodisperse bubble population, and inertially controlled bubble growth. The aim is not to provide a complete flashing-flow model, but to assess whether the proposed nucleation-based formulation can reproduce the main pressure-transient features of rapid depressurization experiments without case-specific tuning. The model is assessed against two classical water expansion-tube experiments spanning different initial pressures and depressurization rates. Despite the simplified formulation adopted here, the computed pressure transients reproduce the measured ONB pressure and subsequent recovery with encouraging accuracy, without case-specific tuning of phase-change parameters. These results support the promise of the approach as a physically grounded basis for future simulations of rapid-depressurization and flashing-flow problems.

Keywords: Flash boiling, Rapid depressurization, Compressible multiphase flow, CFD.

Sub-model and fluids sensitivity of two-fluid boiling predictions

GiovanniTretola[1],KonstantinaVogiatzaki*[1]

[1]DepartmentofEngineeringScience,UniversityofOxford,UnitedKingdom
*Correspondingauthor:konstantina.vogiatzaki@eng.ox.ac.uk

Abstract
Vertical flow boiling in confined channels is increasingly modelled with Euler–Euler two-fluid approaches, yet the predictive behaviour of wall-boiling closures remains strongly dependent on both the chosen submodels and the working fluid. This limits the transferability of calibrated model sets across operating conditions and applications. In this work, a critical analysis is performed for two-fluid boiling simulations in the DEBORA benchmark configuration, consisting of upward flow in a vertical 19.2 mm inner-diameter pipe. The study considers R12 and ethanol, with the latter selected through approximate matching of liquid-to-vapour density ratio, Weber number, Boiling number, and inlet equilibrium quality. The simulations employ an Euler–Euler two-fluid framework with wall heat partitioning. The sensitivity analysis focuses on three coupled modelling ingredients: nucleation-site density, bubble departure frequency, and bubble-diameter treatment, represented either by a constant diameter or by a population-balance model. Their effects are assessed through the axial evolution of region-averaged vapour volume fraction, wall–bulk segregation measures, and signed axial slip velocity. A limited comparison with experimental data shows satisfactory agreement for radial profiles of vapour volume fraction, liquid temperature, and Sauter mean bubble diameter. For the R12 cases, the nucleation-site model is the dominant source of sensitivity, while the influence of departure frequency depends strongly on both the nucleation model and the bubble-diameter treatment. Comparison between R12 and ethanol further shows that fluid choice remains a major source of sensitivity even when selected non-dimensional groups are matched. Overall, the results highlight strong coupled interactions between wall-boiling closures and fluid properties.
Keywords: two-fluid model, Euler–Euler, flow boiling, wall-boiling model

WEDNESDAY

09:50 – 10:40

Droplet Dynamics I *Chair: Marco Marengo*

Event Sequencing in Drop–Pool Impacts at Comparable Reynolds Numbers
D.R. Elbaz, E. Fisher, D. Bykhovsky, M. Arogeti

Surface Curvature Analysis of Atomizing Sprays using a Hybrid Level-set/Lagrangian Method on Unstructured Meshes
E. Bourrel, L. Voivenel, J. Carmona, N. Gasnier, T. Laroche, V. Moureau, F.-X. Demoulin

Interfacial and multiphase physics II *Chair: Günter Brenn*

Off-centre Drop-Particle Collisions: Drop Extension and full Deposition Limit
F. Scope, I. Roa, T. Reisner, C. Planchette

Millimetric drop impacts onto printed meso-structured surfaces
V. Ausfelder, M. Nabernik, I. Roa, C. Planchette

Numerical methods for droplets *Chair: Maria Rosaria Vetrano*

A Sharp-Interface Numerical Framework for Simulating Droplet Impact and Coalescence Phenomena
O. Magen, O. Peles, Y. Kozak

Numerical Investigation Of Droplet Impact On Spherical Particles
M.S. Usmani, S. Bakshi, M. Marengo

11:10 – 12:25

Droplet Dynamics II *Chair: Volfango Bertola*

Size-Dependent Particle Manipulation under Multi-Wave Forcing: An Experimental and Numerical Study
K. Kutsher, N. Haikin, D. Katoshevski

WEDNESDAY

Reactive Phase Change of a Single Droplet in Turbulent Flow with Reactive Vapor–Liquid Equilibrium Using a Two-Scalar Diffusive-Interface Method
S. Jafari, S. Lasala, R. Yao, C. Duwig

Numerical Modeling of Droplet Adhesion and Motion in PEMFC Flow Channels
C. Antetomaso, S.S. Merola, A. Irimescu

Experimental methods for droplets & sprays IV *Chair: Cyril Crua*

Influence of Dual PDA Acquisition Parameters on Measurement Results in a Water Spray
L. Araneo, M. Zimmermann, R. Haidl, B. Weigand

Recent Advances In Phase-Doppler Interferometry For Efficient Spray Characterization
M.D. Leandro, W.D. Bachalo, K.M. Ibrahim, G.A. Payne, P.Y. Chuang

Aerosol Characterization of Small Volume Atomization Using Phase Doppler Anemometry
S. Ferson, E. Ferraris, M.R. Vetrano

Droplet-wall interaction III *Chair: Manuel Reddemann*

Stability Behavior of Faraday Waves for Ultrasonic Atomization
K. Morse, F. Fröde, W. Rohlfs, H. Pitsch, G. Brenn

Analysing spray–wall interaction via near-wall PDA measurements
J. Hajek, P. Novosad, O. Cejpek, M. Maly, J. Jedelsky

1D Modelling of Sprays and Soot Reduction for Mixing-Controlled-Compression-Ignition Engines with Ducted Fuel Injection
R.P. Fitzgerald, Y. Qi, K.I. Svensson, J.M. Garcia-Oliver, J.M. Pastor

WEDNESDAY

14:00 – 15:15

Spray Dynamics IV *Chair: Giulia Finotello*

Formation of streaks in Close-End Pressure Swirl Atomizers with Helical Inlets
O.E. Nural, Ö. Ertunç

Spray characteristics of an electrostatic assisted bi-fluid nozzle
A. Dokmak, R. Bernard, A. Maudhuit, R.J. Schick

Experimental Investigation of Electrostatic Pressure Swirl Atomizer Performance Compared to Conventional Pressure Swirl Atomizer
E.A. Baraya, I.A. Ibrahim, H.M. Gad

Atomization Fundamentals IV *Chair: Simona Tonini*

Review on SCA (Surface Curvature Analysis) an approach to characterize atomization between injection point the final spray
F.-X. Demoulin, A. Sekar, B. Duret, J. Réveillon

Influence of Gas-Liquid Impingement Angle on Internal and External Flow Characteristics in an Internal-Mixing Twin-Fluid Atomizer
S.S. Dharan, M. Raju, Vinil Kumar R.R., S.L.N. Desikan, A. Vaidyanathan

Fractal Dimension Analysis of a 2D Liquid-Jet Interface in DNS
G. Ji, Y. Kulkarni, S. Zaleski

Atomization: Applications III *Chair: Giancarlo Sorrentino*

Spray characteristics of a liquid jet in crossflow: Insights for a gas turbine and high-speed air-breathing engines
R. Kukkarasi, S. Deivandren, R.N. Govardhan

Assessment of Batch Ultrasonic Atomisation for Submicron Seawater Aerosol Generation in Marine Cloud Brightening
M. Sedighi, S. MacLennan, L.P. Harrison, C. Holloway, D.P. Harrison

WEDNESDAY

15:45 – 17:00

Spray Dynamics V *Chair: Ana Moita*

Spray and Evaporation Characteristics of Gasoline-Methanol Blends from a Multi-hole GDI Injector under Engine-like Conditions

D. Ryu, G. Bae, S. Moon

Spray Structure and Atomisation Mechanisms of Flash-Boiling Methanol under Reduced Ambient Pressure and Split Injection

V. Sharma, G. de Sercey, A. Panesar, S. Begg

Interfacial and multiphase physics IV *Chair: Miguel Panão*

Quantitative Visualization of Forced Vapor Advection using an Accessible BOS Approach

B. Ben David Holtzer, L. Tartakovsky

Experimental Investigation of Spray Atomization and Droplet Dynamics of Hydrotreated Vegetable Oil and Jet A-1 Using an Air-Assisted Atomizer

P. di Gloria, I.A. Ferrão, G. Marseglia, A.S. Moita, M.G. De Giorgi

Numerical methods for sprays IV *Chair: Łukasz Jan Kapusta*

Drag and Shape Oscillations in Post-Coalescence Droplet Dynamics

R. Bar-On, J.B. Greenberg, M. Ibach, A. Arad, M. Kumar, V. Vaikuntanathan, D. Katoshevski, B. Weigand

Role of Surface Energy Flux in the Formation of Circular Hydraulic Jumps

H. Askarizadeh, M. Ghanem, Q. Shu, W. Rohlfs

The Role of Droplet Vaporization in the Response of a Laminar Spray Flame to Acoustic Forcing

N.A. Leder, H.S. Gopalakrishnan, M.R. Bothien

Event Sequencing in Drop–Pool Impacts at Comparable Reynolds Numbers

DoronRinaElbaz[1],EtanFisher[1],DimaBykhovsky[2],MeravArogeti*[1]

[1]DepartmentofMechanicalEngineering,ShamoonCollegeofEngineering,Israel
[2]DepartmentofElectricalandElectrinicEngineering,ShamoonCollegeofEngineering,Beer Sheva,Israel
*Correspondingauthor:meravar@sce.ac.il

Abstract
The present study examines the role of Reynolds numbers in mapping the response of a drop impacting a water pool. The central assumption is that changes in Reynolds number are reflected in post-impact events, such as the jetting sequence, secondary-droplet formation, and the strength, persistence, and time–frequency structure of the emitted underwater sound. To examine this relationship, water drops were released into a quiescent deep-water pool over the range 10,000 ≤ Re ≤ 20,000. Each Reynolds number was realized using two to three combinations of drop diameter and impact velocity, allowing impacts at comparable Reynolds numbers to be examined under different geometric and kinematic conditions. The events after impact were documented and measured using high-speed imaging and underwater sound recording. Visual records were used to identify the post-impact sequence, jet morphology, and secondary-droplet production, while the acoustic response was characterized using RMS levels, representative waveforms, and spectrograms. The results show a clear Reynolds-number organization of the impact response, consistent with two known entrainment regimes: Mesler entrainment and irregular entrainment. In the Mesler-side cases, a three-stage sequence is preserved, but the number of secondary droplets decreases as the Reynolds number increases. At higher Reynolds numbers, the response changes more substantially: the impact enters the irregular entrainment regime, a secondary upward jet appears, and the acoustic emission becomes markedly stronger and more persistent. These findings support the use of the Reynolds number as an organizing parameter for constructing a visual–acoustic regime map of the drop–pool impact response.
Keywords: drop impact acoustics; bubble entrainment; Reynolds-number regimes

Surface Curvature Analysis of Atomizing Sprays using a Hybrid Level-set/Lagrangian Method on Unstructured Meshes

Elisa Bourrel*[1], Léa Voivenel[1], Julien Carmona[1], Nicolas Gasnier[2], Thomas Laroche[3], Vincent Moureau[1], François Xavier Demoulin [1]

[1] CNRS, INSA Rouen Normandie, Univ Rouen Normandie, Normandie Univ, CORIA UMR 6614, F-76000 Rouen, France
[2] SAFRAN Tech, 78772 Magny-les-Hameaux, France
[3] SAFRAN Helicopter Engines, 64510 Bordes, France
*Corresponding author: elisa.bourrel@coria.fr

Abstract

The transition to Sustainable Aviation Fuels (SAF) and the wider operating range of aircraft engines demand a better understanding of liquid injection systems. Variations in fuel properties, particularly viscosity due to composition or temperature changes, strongly affect spray formation and injector performance. Changes in viscosity influence hydrodynamic instabilities, liquid film thickness, jet breakup, and atomization, which in turn impact momentum and energy exchanges between liquid and gas, modifying spray angle and droplet size. Accurately capturing these effects is essential for designing injection systems that perform reliably across diverse fuels and conditions.

In this work, a strategy for characterizing aeronautical injection systems based on high-fidelity numerical simulations is proposed. The liquid phase dynamics are explicitly resolved using the Accurate Conservative Level Set (ACLS) method combined with unstructured mesh adaptation [4]. ACLS belongs to the broader class of interface-capturing techniques, alongside Volume-of-Fluid (VoF) [3] and coupled CLSVoF formulations [6], which have been widely applied to atomization studies. However, employing these approaches for configurations relevant to aero-engines necessitates validation using representative experimental data.

To this end, specific test cases derived from experiments are examined, with a particular focus on the CRSB configuration, which has been thoroughly reported in the literature (see Ferrando [2] and other related experimental works). Simulation results for this configuration have previously been reported by Salvador [5] and Carmona [1] with respect to conventional validation metrics, including spray angle and droplet size distributions. Beyond these standard quantities, high-fidelity simulations provide access to a much richer set of information, since the entire liquid phase is resolved.

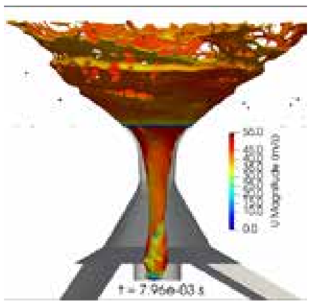


Figure 1. *Snapshots of the liquid sheet interface. [1].*

The originality of the present work lies in exploiting this detailed information to propose advanced characterization metrics for liquid sprays. In particular, the complex morphology of the liquid surface is analyzed through Surface Curvature Analysis (SCA) [2], while velocity fields, energy balance, and momentum exchanges between the liquid and gas phases are examined. These quantities are shown to be highly sensitive to liquid properties, especially viscosity, and therefore provide valuable insights into the influence of fuel characteristics and operating conditions. While similar analyses have been conducted in previous studies, the present work extends them by systematically exploring the effect of viscosity variations induced by fuel temperature changes. The proposed methodology is intended to provide numerical results that can be directly used to design and optimize aeronautical injection systems, particularly in the context of fuel diversification and the ongoing energy transition.

References

[1] J. Carmona et al. In: *5th ICNMMF*, Reykjavik, Iceland, 2024.

[2] D. Ferrando et al. *Atomization and Sprays* 33.3 (2023), pp. 1–28.

[3] C.W Hirt et al. *Journal of Computational Physics* 39.1 (1981), pp. 201–225.

[4] Romain Janodet. *Journal of Computational Physics* 458 (2022), p. 111075.

[5] F.J. Salvador et al. *International Journal of Multiphase Flow* 183 (2025), p. 105066.

[6] M. Sussman et al. *Journal of Computational Physics* 162.2 (2000), pp. 301–337.

Off-centre Drop-Particle Collisions: Drop Extension and full Deposition Limit

Felix Scope*[1], Ignacio Roa[1], Teresa Reisner, Carole Planchette*[1]

[1] Institute of Fluid Dynamics and Heat Transfer, Graz University of Technology, Austria
*Corresponding authors: scope@tugraz.at, carole.planchette@tugraz.at

Abstract

Drop-Particle interactions are relevant to both natural atmospheric phenomenon and industrial processes aiming to collect, bridge or encapsulate the dispersed particles with the liquid. The experimental and numerial difficulties encountered for these studies have limited the availability of reliable data, resulting in poor understanding and modelling of these processes and their outcomes. This study exploits on the one side the experimental method developed by Le Gac and Planchette to systematically explore the full deposition limit for Ohnesorge numbers of 0.014 and 0.032 with Weber numbers (We) between 25 and 200 over full eccentricity range, i.e. $0.0 < X < 1.0$. On the other side, these experimental data are used to validate a numerical method, presented here and designed for further exploring the rich parameter space of drop-particle collisions. Very good agreement between experimental and numerical results is obtained opening the route for predicting the limit of full deposition beyond the domain that can be experimentally explored. In the parameter space probed to date, we show that a critical drop extension can be used as a separation criterion. We note, that the evolution of this extension follows different trends below and above an eccentricity of X 0.5. We interpret this observation as a consequence of a strong flow field modification related to the redirection of the drop momentum around or aside the particle.

Keywords: drop, particle, collision, deposition, separation, numerical simulations, experimental method

Millimetric drop impacts onto printed meso-structured surfaces

VeronikaAusfelder*[1],MarioNabernik[1],IgnacioRoa[1],CarolePlanchette*[1]

[1]InstituteofFluidDynamicsandHeatTransfer,GrazUniversityofTechnology,Austria
*Correspondingauthor:carole.planchette@tugraz.at

Abstract
Microor meso-structures are known to affect extension and recoil of drops impacting onto these surfaces. Thus, it has been proposed to make use of these effects to tune printing resolution, especially by modulating line thickness, or to adjust deposited film thickness by controlling the surface on which the drops may spread. This study aims to explore these possibilities and compares the dynamics and final state of millimetric water drops impacting onto various substrates consisting of the same material but whose structure is changed. The studied topologies include smooth surface as well as triangular ridges with fixed period and heights found between roughly 50 and 200µm. Weber number ranging from unity to 150 are considered and both numerical and experimental approaches are combined. On smooth surfaces, the simulations carried out using ANSYS with a dynamic contact angle based on Kistler-Hoffman law show a good agreement with the experiments. Yet, these simulations also indicate some limitation, especially in their ability to reproduce inertia driven wetting and appear at this stage insufficient to explore effects produced by slightly differently structured surfaces. To clarify the role of the microstructure and its irregularities, experiments were conducted for similar W e on structured substrates produced via inkjet printing by translating the print-head either perpendicularly or parallel to the ridge axis. Small but systematic differences are observed between substrates produced using different printing directions. However, in the absence of a complete topographical characterization, it is not yet possible to distinguish the influence of secondary microscopic surface features from that of possible variations in the actual ridge geometry. Further topographical characterization is therefore required.
Keywords: Drop impact, spreading, deposition, surface structure, roughness

A Sharp-Interface Numerical Framework for Simulating Droplet Impact and Coalescence Phenomena

OmryMagen[1],OrenPeles[1],YoramKozak*[1]

[1]CombustionANDEnergyLab(CANDEL),SchoolofMechanicalEngineering,Facultyof Engineering,TelAvivUniversity,Israel
*Correspondingauthor:kozaky@tauex.tau.ac.il

Abstract
The accurate simulation of spray atomization and droplet dynamics requires a numerical framework capable of resolving multiscale topological changes across a wide range of flow regimes. In sharpinterface formulations, however, the interaction between liquid structures and the surrounding gaseous phase often leads to numerical difficulties, which become particularly pronounced in three dimensions. This work presents a robust computational framework implemented within the adaptive mesh refinement code Athena++. We solve the fully compressible conservation equations for multiple phases utilizing the Level Set Method (LSM) for precise interface tracking. The phases are coupled via the Ghost Fluid Method (GFM), where local Riemann problems are solved across the liquid-gas interface to determine the necessary boundary conditions for each phase. To address topological transitions, a specialized scheme for coalescence and breakup is introduced. Prior to the hydrodynamic update, a sub-grid detection mechanism flags configurations in which the interfacial Riemann problem becomes ill-posed, and the affected cells are populated with consistent primitive variables through a controlled numerical procedure.
Keywords: Level-set method, Ghost Fluid Method, Multiphase, Surface Tension, Coalescence

Numerical Investigation Of Droplet Impact On Spherical Particles

Mohd Saif Usmani[1], Shamit Bakshi*[1], Marco Marengo[2]

[1]Indian Institute of Technology Madras, Chennai, India
[2]University of Pavia, Pavia, Italy
*Corresponding author: shamit@iitm.ac.in

Abstract

Droplet impact on solid surfaces is a widespread phenomenon encountered in natural and industrial processes, including raindrop impingement, inkjet printing, surface coating, and pharmaceutical applications. Over the past decades, extensive research has focused on droplet impact dynamics, primarily considering flat and stationary surfaces. However, in many practical situations droplets interact with curved and moving geometries rather than flat substrates. This highlights the need to investigate droplet behaviour on more complex configurations, such as spherical particles. Although previous studies have examined droplet impact on spherical surfaces, most have assumed the particle to be stationary while only the droplet was in motion. To address this limitation, the present study numerically investigates the impact of a droplet onto a moving spherical particle. The simulations are performed using the commercial finite-element-based solver COMSOL Multiphysics under an axisymmetric framework. To accurately capture the interfacial dynamics, the influence of slip length on droplet spreading and rebound is incorporated. In addition, the dynamic contact angle model is implemented and validated against available experimental data from the literature. Various impact outcomes, were observed at different parameters. The findings provide deeper insight into droplet–particle collision dynamics and represent a foundational step toward understanding droplet impact on more complex moving geometries.

Keywords: Droplet–particle collision; dynamic contact angle; finite element method; moving spherical particle

Size-Dependent Particle Manipulation under Multi-Wave Forcing: An Experimental and Numerical Study

Kobi Kutsher*[1,2], Nitsa Haikin[3], and David Katoshevski[1]

[1] Civil and Environmental Engineering, Ben-Gurion University of the Negev, Israel
[2] Soreq-NRC, Yavne, Israel
[3] Department of Physics, NRCN, Beer-Sheva, Israel
*Corresponding author: kutsher@post.bgu.ac.il

Abstract

The manipulation of particle and droplet size distributions utilizing grouping and coalescence phenomena provides a highly promising approach for optimizing multiphase systems. Historically, particle grouping has been achieved using either oscillatory flow fields or acoustic wave manipulation independently. This study introduces a novel multi-wave approach that superimposes frequencies of flow oscillations, acoustics, and geometric modulation to significantly enhance the grouping and coalescence efficiency of fine-dispersed phases in a controlled manner. This extended abstract presents the theoretical foundation, a comprehensive 1D mathematical model incorporating Brownian motion and multi-wave sources, the design of an advanced experimental grouping system, 2D computational fluid dynamics (CFD) simulations, and empirical results. The experimental results demonstrate that geometric modulation (wavy ducts) has a profoundly dominant impact on size-shifting. Furthermore, numerical analysis reveals that Brownian motion significantly accelerates grouping for particles smaller than $\mathbf{0.3\ \mu m}$. Additionally, 1D modeling and 2D CFD confirm that inducing an acoustical source simultaneously with flow oscillation lowers the required grouping time by up to 33%, while highfrequency oscillations generate localized recirculation zones. Beyond its implications for particulate filtration, this synergistic multi-wave manipulation offers profound potential for advancing spray dynamics, controlling droplet coalescence and evaporation rates, and driving innovations in space engineering and nanotechnology.

Keywords: Agglomeration, Filtration efficiency, Segregation and sorting, Spray Dynamics

Reactive Phase Change of a Single Droplet in Turbulent Flow with Reactive Vapor–Liquid Equilibrium Using a Two-Scalar Diffusive-Interface Method

SajadJafari*[1],SilviaLasala[2],RanYao[1],ChristopheDuwig[1]

[1]DepartmentofChemicalEngineering,RoyalInstituteofTechnology(KTH),Sweden
[2]UniversitédeLorraine,CNRS,LRGP,F-54000Nancy,France
*Correspondingauthor:sajadja@kth.se

Abstract
Reactive phase change fluids have been shown to enhance thermodynamic performance by integrating chemical reactions within thermodynamic cycles. This study investigates the turbulent evaporation of a single droplet using Direct Numerical Simulation (DNS) to model the behavior of reactive phase change fluids. Applying Reactive vapor-liquid equilibrium (RVLE) thermodynamics, the study simultaneously captures the dynamics of reactive phase transitions, mass transfer, and heat transfer during the evaporation of dinitrogen tetroxide (N_2O_4). A second-order phase field framework is used to derive two scalar transport equations, accurately describing species behavior at equilibrium. The equilibrium is maintained by balancing concentration exchange rates across both the liquid-to-vapor and vapor-to-liquid phases. Results indicate that chemical reactivity significantly influences phase transition dynamics, altering the phase dome structure by modifying critical points and the thermodynamic saturation lines. Endothermic and exothermic behaviors are observed at the interface during the fast reversible evaporation process of N_2O_4 (v,l) ⇌ $2NO_2$ (v,l). Turbulence simulations further show enhanced mixing efficiency and improved gas-liquid separation, driven by chaotic flow patterns. This study elucidates the role of reactive phase change fluids in contrast to conventional non-reactive phase flows, emphasizing the influence of chemical reactions on phase transition dynamics and overall system performance.
Keywords: DNS, Reactive Vapor-Liquid Equilibrium, Reactive Evaporation, Diffuse Interface, Two scalar

Numerical Modeling of Droplet Adhesion and Motion in PEMFC Flow Channels

Christian Antetomaso*[1], Giovanni Cecere[1], Simona Silvia Merola[1], Adrian Irimescu[1]

[1]CNR STEMS, Naples, Italy
*Corresponding author: christian.antetomaso@stems.cnr.it

Abstract
Efficient water management in Proton Exchange Membrane Fuel Cells (PEMFCs) is essential to enhance electrochemical performance and enable large-scale commercialization. A key issue is the reliable numerical prediction of liquid water behavior within gas flow channels, where droplet accumulation can obstruct reactant transport and degrade overall efficiency.

This study presents a novel implementation of a dynamic contact angle boundary condition in OpenFOAM through an extension of the interFoam solver and the dynamicAlphaContactAngle boundary. The methodology integrates a flood-fill algorithm to identify and isolate individual droplets within the computational domain. The liquid–gas interface is reconstructed to evaluate interfacial forces, enabling the calculation of drag and adhesion contributions acting on each droplet. Based on these quantities, the solver determines whether a droplet remains pinned to the surface or transitions to motion.

The proposed framework enhances the physical consistency of contact line modeling and provides a more realistic description of droplet–wall interactions under dynamic flow conditions. Numerical simulations demonstrate the model's capability to capture dynamic wetting effects, shielding phenomena between adjacent droplets, and complex deformation patterns. The results confirm the robustness of the approach and its potential for improving predictive simulations of water transport in PEMFC channels.

Influence of Dual PDA Acquisition Parameters on Measurement Results in a Water Spray

*1222
Lucio Araneo, Manuel Zimmermann, Richard Haidl, Bernhard Weigand

1
Politecnico di Milano, Italy
2
Institute of Aerospace Thermodynamics (ITLR), University of Stuttgart, Germany
*Corresponding author: lucio.araneo@polimi.it

Abstract

In the present work, a Dual PDA system by Dantec Dynamics is used to non-invasively and simultaneously measure the droplet diameter and velocity distribution of a water spray. While the Dual PDA system is a powerful measuring system, various possibilities to change the combination of acquisition parameters on the hardware and software side make it difficult to find an optimal setup, especially for different operating conditions of the spray. The focus of this work lays on understanding the effects caused by changes of the acquisition parameters of the measuring system and the optimisation of these parameters to achieve high-quality measurements. The set-up was firstly optimised for a wide set of operation conditions, by choosing appropriate optical parameters to allow velocity and diameter measurement ranges to cover the expected outcomes, considering three different nozzles operated at different water pressure levels. Then, for a single spray operating condition, measurements were performed at a fixed position in the spray which was considered sufficiently difficult for measurements due to a high droplet density and high velocity. The acquisition parameters of the PDA were varied to understand and quantify their effects on the measurement results. The studied parameters include: centring of the receiving optic slit aperture in the measurement volume, slit width, laser power, signal gain, signal SNR validation, photo multiplier amplification voltage, and spherical validation. Results are summarized by the effect of the studied parameters on different statistical values like the acquired sample rates, the average velocity and diameters (D10, D20, D30, D32), and the spherical validation. More in depth data analysis will estimate the measurement volume real dimension as a function of the droplet diameter, thus allowing to calculate more reliable statistical values, and finally help to better understand how and why the PDA results should be interpreted.

Keywords: PDA, Dual PDA, LDA/LDV, Spray, Droplets, Hollow cone nozzle

Recent Advances In Phase-Doppler Interferometry For Efficient Spray Characterization

Mason D. Leandro*[1,2], William D. Bachalo[1], Khalid M. Ibrahim[1], Gregory A. Payne[1], Patrick Y. Chuang[2]

[1]Artium Technologies, Inc., Sunnyvale, California, USA
[2]Earth and Planetary Sciences, University of California Santa Cruz, USA
*Corresponding author: mleandro@artium.com

Abstract
Phase-Doppler interferometry (PDI, aka PDA, PDPA) is a worldwide standard for spray characterization, providing simultaneous measurements of droplet size and up to three components of velocity from lightscattering interferometry signals referenced to the laser wavelength. It enables defensible measurements of droplet size distributions and derived quantities such as number concentration, liquid water content, and volume flux in laboratory and applied spray environments. The method has the proven capability of providing nonintrusive in situ measurements in challenging environments including in reacting spray flows. This work highlights recent advances that improve measurement efficiency and robustness, focusing on separation and sizing during coincident events, an extended dynamic drop size range, and probe volume characterization. We first present a time-resolved signal parsing strategy that separates and sizes individual droplets during coincident events. Coupled with a dynamically adjusted probe volume informed by built-in Poisson analysis, this approach improves data quality and throughput across a wide range of spray densities. Next, we introduce a next-generation signal processing means that supports a dynamic size range of 100:1 or greater at a given setting (i.e., 1-100 μm), enabling wider coverage for broad polydisperse sprays. Finally, we describe a probe volume mapping approach using a traversing monodisperse droplet stream to quantify each dimension of the effective sample area. Systematic scans reveal spatial limits of detection, which are then compared to theoretical estimates and values inferred from in situ measurements. Together, these developments extend the PDI capability into more challenging spray regimes, including higher concentrations and broader size distributions, while improving confidence in derived spray metrics.
Keywords: phase Doppler, spray characterization, droplet sizing, interferometry

Aerosol Characterization of Small Volume Atomization Using Phase Doppler Anemometry

SimonFerson*[1],EleonoraFerraris[2],MariaRosariaVetrano[1]

[1]DivisionofAppliedMechanicsandEnergyConversion(TME),DepartmentofMechanical Engineering,KULeuven,Belgium
[2]DivisionofManufacturingProcessesandSystems(MaPS),DepartmentofMechanical Engineering,KULeuven,Belgium
*Correspondingauthor:simon.ferson@kuleuven.be

Abstract
Small Volume Nebulizers (SVNs) are widely employed in drug delivery systems and micro manufacturing processes for producing tailored aerosols thanks to their ability to atomize small liquid quantities with minimal residual volume. The characteristics of the resulting aerosol, particularly its droplet size distribution and concentration, are critical for downstream applications as they govern subsequent physical phenomena such as droplet-gas coupling and evaporation. Additionally, the stability of these properties is equally essential. SVNs typically share a similar design: an ultrasonic transducer is mounted on the bottom of a water bath, which is used to transfer ultrasonic energy into milliliter-scale liquid volumes in a vial or on a membrane [1]. The applied transducer voltage and liquid temperature play a central role in governing atomization behavior, as these influence the onset and stability of the ultrasonic fountain and affects subsequent droplet breakup dynamics. In this study, we examine how increasing the driving voltage and liquid temperature alter both the droplet size distribution and the total aerosol yield. Phase Doppler anemometry (PDA) is employed to characterize relative changes in the droplet population by varying the transducer voltage and bath temperature. As droplet sizes approach the lower detection limit of conventional PDA systems, a preliminary sensitivity analysis is conducted to assess PDA's capability to reliably measure submicron droplets. This sensitivity study is inherently challenging, as the droplet population cannot be assumed stationary in time, owing to the intrinsically transient and intermittent nature of ultrasonic atomization.
Keywords: Small Volume Nebulizer, Phase Doppler Anemometry, Ultrasonic Atomization

Stability Behavior of Faraday Waves for Ultrasonic Atomization

KristenMorse*[1],FabianFröde[2],WilkoRohlfs[3],HeinzPitsch[2],GünterBrenn[1]

[1]GrazUniversityofTechnology,InstituteofFluidMechanicsandHeatTransfer(ISW),Graz,AT
[2]RWTHAachen,InstituteforCombustionTechnology(ITV),Aachen,DE
[3]RWTHAachen,InstituteofHeatandMassTransfer(WSA),Aachen,DE
*Correspondingauthor:kristen.morse@tugraz.at

Abstract

A liquid film perturbed by vertical oscillations produces Faraday waves that initially exhibit linear growth. The linear growth behavior of these waves is investigated numerically via three-dimensional interfaceresolved simulations and compared with the linear theory for predicting surface wave growth rates established in Morse et al. [1], exhibiting excellent agreement. A study of wavenumber selection in a large domain initialized with white noise is then used to investigate the physically realized wavelengths. Modal analysis shows that the selected wavenumbers coincide with local growth-rate maxima predicted by Morse et al. [1], with the largest-growth mode dominating the film response at a given forcing amplitude. We identify the wavenumbers present in the linear regime persist through the onset of the nonlinear regime, with the additional emergence of the second harmonic. Visualizations show that the wave pattern established in the linear regime remains intact in the nonlinear regime. In the nonlinear regime, the surface waves transition to having sharpened, inertia-dominated crests with pronounced ridges. The nonlinear structures near the point of droplet ejection further provide insight into the broader applicability of Lang's equation [2] for predicting characteristic droplet diameters.

Keywords: Faraday waves, Ultrasonic atomization, Interface-resolved simulations, Stability analysis

Analysing spray–wall interaction via near-wall PDA measurements

Jirí Hajek*, Pavel Novosad, Ondrej Cejpek, Milan Maly, Jan Jedelsky

Laboratory of Multiphase-flow, Faculty of Mechanical Engineering, Brno University of Technology, Czech Republic
*Corresponding author: Jiri.Hajek2@vutbr.cz

Abstract
Droplet–wall interaction criteria are commonly used to predict droplet behaviour during spray-wall impingement. However, the spray is directly influenced by the wall, shifting from an axial to a radial direction near the wall. This flow deflection changes a drag force on the droplets, the effect of which is determined by the droplet Stokes number (St). To investigate these dynamics, an angled configuration of Phase Doppler Anemometry (PDA) system was employed to measure droplet sizes and velocities as close as 0.1 mm above the target surface. The study evaluated different scenarios by varying the airbrush (AB) operating pressure from 50 to 250 kPa and the target diameter from 10 to 30 mm. The results indicate that the radial position (X), the distance above the target (Z), and the droplet St are the primary parameters influencing the droplet velocity deflection from the spray axis θ. No significant influence of target size was observed. The wall's influence on the flow was first detected 10 mm above the target, where droplets with small St began to redirect. In contrast, droplets with St > 5 maintained a stable trajectory down to 4 mm above the target. At a position Z = 0.2 mm and X = 2 mm from the spray axis, even droplets with St > 5 reached θ values exceed 10°. Based on the measured data, an empirical correlation for θ was developed, counting the influence of radial and axial position and St, achieving an R2 = 0.8. These findings are crucial for improving the accuracy of spray-wall impingement models when applying single-droplet interaction criteria.
Keywords: Spray–wall interaction, Phase doppler anemometry, Near-wall measurement, Droplet velocity angle

1D Modelling of Sprays and Soot Reduction for Mixing-Controlled-Compression-Ignition Engines with Ducted Fuel Injection

Russell P. Fitzgerald*[1], Yongli Qi[1], Kenth I. Svensson[1], Jose Maria Garcia-Oliver[2], Jose Manuel Pastor[2]

[1]Combustion and Air Systems Research, Caterpillar Inc., USA
[2]Clean Mobility & Thermofluids, Univérsitat Politécnica de Valencia, Spain
*Corresponding author: Fitzgerald_Russell_P@cat.com

Abstract
Mixing-controlled-compression ignition (MCCI) of traditional fuels (like diesel) offer distinct performance advantages over other combustion modes but is often challenged with high in-cylinder emissions like soot. Ducted fuel injection (DFI), a promising strategy for reducing engine-out soot has been investigated over the last decade with some progress; however, some challenges remain. Retrofit DFI appears to be challenged by soot oxidation and poor/modest performance over an extended engine load-range, while questions still linger about the fundamental physics of soot formation and reduction for such wall-constrained sprays. The latter of these are explored in this work through development and validation of a 1-D spray model (DICOM) extended to include effects of DFI. The extended DICOM model simulates previously observed features of DFI by limiting radial airentrainment when the expanding spray impinges on the duct inner diameter, thus capturing higher mean jet velocities with faster jet penetration, locally richer/cooler sprays with longer ignition delays and extended lift-off lengths. These features are necessary to achieve lower fuel mixture fraction in the early flame zone. Simulation results and model predictions are compared with previous experimental data (including spray penetration, dispersion, mixture fraction, lift-off length, and natural luminosity from soot) to assess predictive capability and assumption validity. Optimization of the duct diameter is achieved by minimizing the local fuel mixture fraction at the predicted flame lift off length without forcing the spray interaction too close to the duct entrance.
Keywords: fuel sprays, reciprocating engines, ducted fuel injection, wall-interaction, mixture formation, simulations

Formation of streaks in Close-End Pressure Swirl Atomizers with Helical Inlets

Ozan Ekin Nural[1], Özgür Ertunç*[2]

[1] TUSAS Engine Industries, Eskişehir, Türkiye
[2] Fluid Dynamics and Spray Laboratory, Mechanical Engineering, Özyeğin University, Istanbul, Türkiye
*Corresponding author: ozgur.ertunc@ozyegin.edu.tr

Abstract
While certain parameters, such as spray angle and droplet distribution, have been widely studied and modeled for pressure swirl atomizers (PSAs), the circumferential uniformity of the atomizer flow and its relation to streak formation remain poorly understood. In the present work, a novel pressure swirl atomizer design with helical inlets [1] was used to study uniformity and streak formation. 3D numerical simulations using the volume-of-fluid method were performed over a range of PSAs, and three atomizers were fabricated via selective laser etching to ensure maximum manufacturing precision. Of the fabricated atomizers, one exhibited uniform distribution, while the other two had deficiencies in both internal and external flow fields. LIF/Mie spray imaging on a radial plane and high-speed images on the central axial plane of the manufactured atomizers were conducted. The results indicated that non-uniformity in the internal flow greatly impacts downstream streak formation and droplet distribution. One important root cause of non-uniformity was determined to be the interaction of inlet flows as they enter the swirl chamber. It is concluded that ensuring a uniform mass flux distribution in the atomizer during the design phase by reducing this interaction is essential for a more uniform spray distribution.

Keywords: Pressure swirl atomizer, helical, streak, uniformity

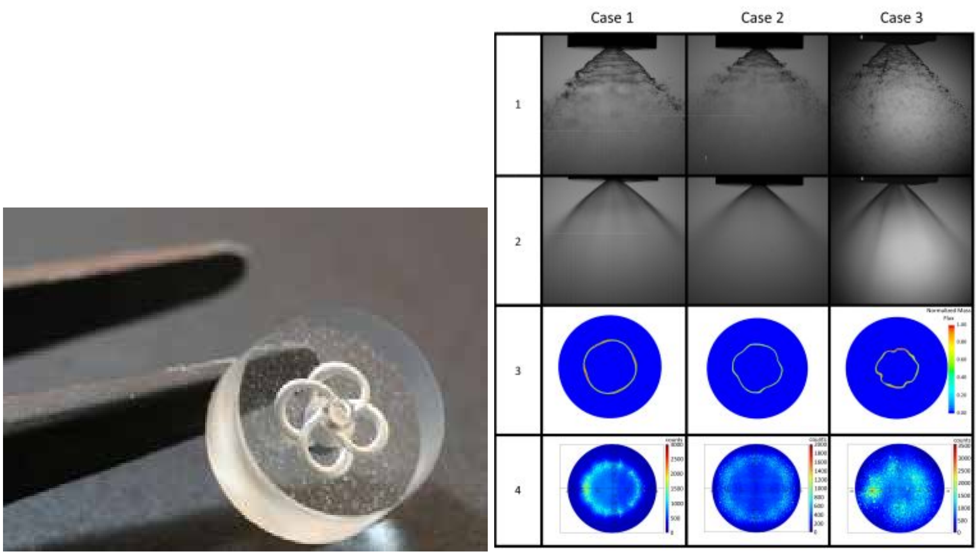


Figure 1. *PSA with helical inlet (left). Findings for three manufactured atomizers operated at 3.34 g/s (right). 1st and 2nd row: instantaneous and average spray images, respectively. 3rd row: Computed normalized mass flux at a plane 0.7 mm downstream of the orifice exit. 4th row: the intensity of LIF signal taken 32 mm downstream of the orifice exit.*

References
[1] T. Hacıbekir, I. H. Savcı, O. E. Nural, Ö. Bebek, and Ö. Ertunç, "Fluid atomizer with helical inlet channel," US12220717B2 (2025).

Spray characteristics of an electrostatic assisted bi-fluid nozzle

Akaber Dokmak*[1], Ronan Bernard*[1], Audrey Maudhuit[2], R. J. Schick[1]

[1]Spray Analysis and Research Services, Spraying Systems Co.
[2]Fluid Air, Division of Spraying Systems Co., France

*Corresponding authors: akaber.dokmak@spray.com, ronan.bernard@spray.com

Abstract
Aerodynamic and electrostatic atomization techniques are commonly employed separately to provide spray atomization, while their combined use in electrostatic-assisted bi-fluid nozzles has been only sparsely documented, although used in several applications like spray drying. This study investigates the atomization behavior of a hybrid electrostatic-assisted bi-fluid nozzle, in which pneumatic shear and electric field act simultaneously. An experimental study was conducted using deionized water, where liquid flow rate, air-atomizing pressure, and applied voltage were varied. Droplet size distributions were measured alongside spray pattern characterization using laser sheet imaging combined with in-house image processing techniques, including dispersion and circularity analysis. The results show that droplet size is primarily governed by aerodynamic shear, decreasing with increasing air pressure, while electrostatic effects become significant only at low air pressures, where they promote droplet fragmentation. Spray dispersion exhibits a strong voltage dependence under weak aerodynamic conditions, reflecting enhanced radial spreading due to electrostatic forces, whereas the dense core region remains largely unaffected, indicating inertia-dominated behavior in the spray core. Circularity analysis further reveals that electrostatic assistance broadens spray but reduces its symmetry. Overall, the study demonstrates a regime-dependent interplay between aerodynamic, inertial, and electrostatic forces, highlighting that electrostatic effects on spray behavior occur mostly under low air atomizing pressure conditions.

Keywords: Electrostatic assisted atomization; bi-fluid nozzle; droplet size distribution; spray pattern; electric effects.

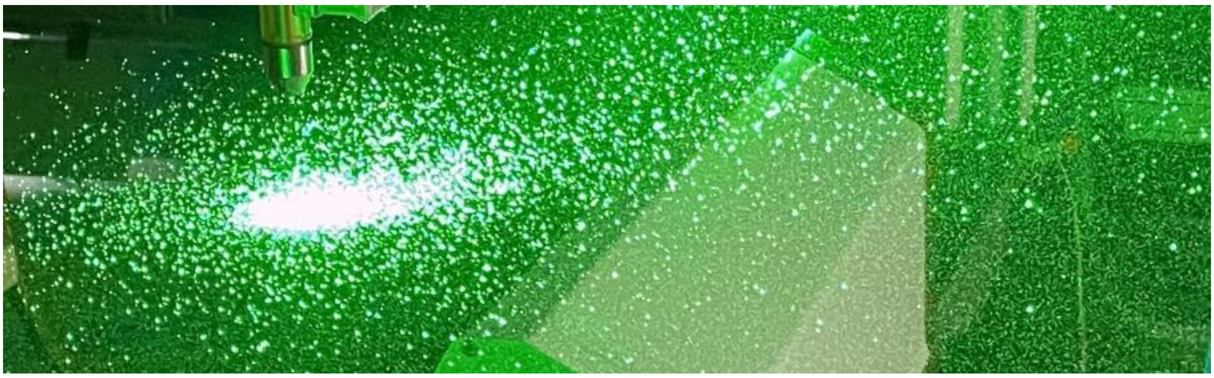


Figure 1. Laser Sheet Imaging of an electrostatic-assisted air-atomizing spray nozzle

Experimental Investigation of Electrostatic Pressure Swirl Atomizer Performance Compared to Conventional Pressure Swirl Atomizer

Baraya E.A.*[1], Ibrahim I.A.[2], Gad H.M.[2]
[1]Suez Canal Authority, Dredging Department, Ismailia, Egypt
[2]Mechanical Power Engineering Department, Faculty of Engineering, Port Said University, Egypt
*eslam.ahmad@eng.psu.edu.eg

Abstract

Pressure swirl atomizers (PSAs) are extensively used in combustion and industrial spray systems because of their simple design and reliable operation. However, improving atomization generally requires higher injection pressures or nozzle redesign, increasing energy consumption and system complexity. This study presents an electrically insulated Electrostatic Pressure Swirl Atomizer (ESPSA) that combines conventional hydraulic swirl atomization with electrohydrodynamic charging while preserving the original hydraulic geometry. Both atomizers were manufactured with identical dimensions and tested under identical operating conditions to isolate the effect of electrostatic charging. Injection pressure was varied from 1 to 9 bar, while the applied voltage ranged from 10 to 40 kV. Spray characteristics including spray cone angle (SCA), breakup length (L_b), Sauter Mean Diameter (SMD), spray concentration distribution (SCD), and spray momentum distribution were experimentally investigated using high-speed imaging, validated ImageJ image processing, inline patternation, and force-based spray momentum measurements.

The application of the electric field significantly enhanced atomization performance. Increasing the applied voltage from 10 to 40 kV increased the spray cone angle from 52° to 74°, reduced the breakup length from 9 mm to approximately 0.5 mm, and decreased the SMD from 63 µm to 30 µm. Under identical operating conditions, the ESPSA achieved a reduction in SMD of up to 70–75% compared with the conventional PSA. Electrostatic charging also transformed the spray from a hollow-cone pattern to a partially filled cone, producing a more uniform liquid concentration and spray momentum distribution across the spray cross-section. The experimental results demonstrate that electrohydrodynamic assistance provides an effective supplementary atomization mechanism without modifying the hydraulic nozzle design or increasing injection pressure. The proposed ESPSA offers a practical approach for improving atomization quality in combustion systems and other industrial spray applications requiring enhanced mixing, higher efficiency, and reduced emissions.

Keywords: Electrostatic pressure swirl atomizer; Electrohydrodynamic atomization; Spray cone angle; Sauter Mean Diameter; Breakup length; Industrial combustion.

Atomizer Design and Experimental Configuration

1	Atomizer body
2	Locking part to swirl fuel passage
3	Swirling part of fuel
4	Atomizer cap that contains spin chamber
5	Dc charge injection needle

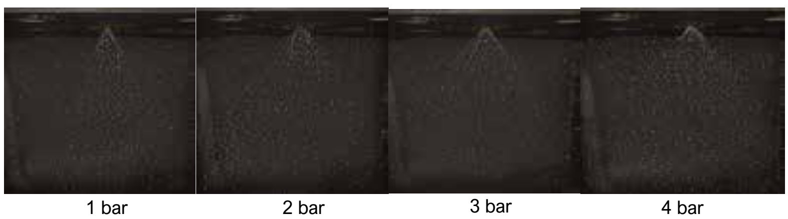


Fig.1.Effect of Injection Pressure P_{inj} on the ESPSA at 30 Kv with fixed geometry

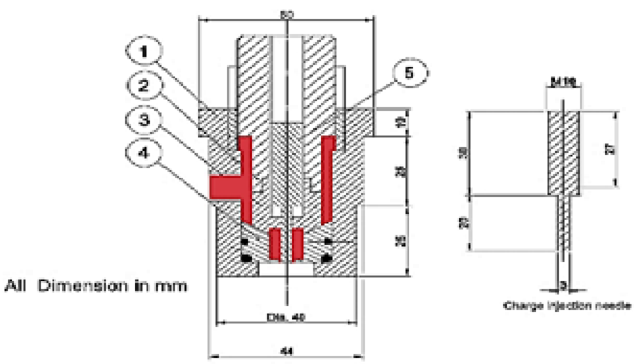


Fig.2. Detailed dimensions of the ESPSA

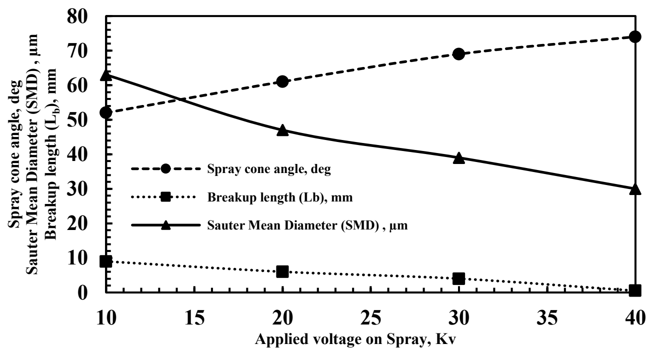


Fig. 3. Effect of Applied Voltage on the SCA, SMD and L_b of the ESPSA

Review on SCA (Surface Curvature Analysis) an approach to characterize atomization between injection point the final spray

François-Xavier Demoulin*[1], Anbarasan Sekar[1], Benjamin Duret[1], Julien Réveillon[1]

[1]University of Rouen Normandy, CRNS CORIA, France
*Corresponding author: demoulin@coria.fr

Abstract
Surface Curvature Analysis (SCA) [1] is an approach used to characterize atomization from the injection point to the final spray. This work aims to present the current status of SCA, a method designed to characterize liquid-gas surfaces with complex interface morphologies. The atomization process can be defined as the transformation of simple liquid structures (liquid sheets, jets, etc.) into a final spray, which is expected to be dilute and composed of spherical droplets. Between these stages, instabilities, breakup events, collisions, and coalescence occur, leading to liquid elements with highly complex interface morphologies. Despite great achievements in linear instability analysis and atomization models based on liquid blobs and droplets, there has been a lack of descriptive tools for characterizing atomization when the surface morphology is complex. Nowadays, it is possible to access these zones either through advanced numerical simulations, where such complex surfaces are captured, or by experimental means using high-resolution imagery. To describe complex liquid structures, different tools exist, including scale distribution and Minkowski functionals; however, a simpler approach consists of analysing the distribution of curvature at the liquid-gas interface: the so-called Surface Curvature Analysis (SCA). The benefit of this approach is its accessibility when based on principal curvatures in numerical simulations, as this information is already available for computing surface tension forces. From an experimental perspective, the simplest approach involves determining the liquid-gas interface as the contour of liquid elements. While many methods exist for contour detection and for measuring surface area and curvature, it is important to note that most images are 2D, meaning one dimension is lost. Accordingly, liquid volume is reduced to a projected surface, and surface area to contour length. Consequently, quantities like $D32$ (volume-to-surface ratio) are restricted to $D21$ (surface-to-contour length ratio). This 2D experimental limitation does not diminish the importance of experiments for SCA, as they characterize real flows independent of numerical resolution constraints, whereas simulations provide 3D features but may be affected by resolution limits that impact flow dynamics. Current understanding of SCA demonstrates the complementarity of experiments and numerical simulations in establishing the following: 1. SCA can be used in both experiments and simulations to characterize atomization where the surface morphology is complex. 2. SCA is quantitative and sensitive enough to describe minute changes in atomization, such as those caused by variations in transport properties due to temperature fluctuations. 3. It is possible to transform the surface curvature distribution into a droplet/bubble size distribution (dDSD) based on number, with mathematical conservation of information. Furthermore, one can isolate surface segments corresponding to potential droplets to determine an equivalent drop size distribution (eDSD). It has been observed in all cases studied to date that the eDSD determined in step 3 collapses very early during the atomization process (while the surface is still complex) into a stable eDSD. For these cases, the eDSD determined at the early stage compares well with standard approaches used to characterize the final drop size distribution, such as PDA. The reason why the initial stage of atomization carries the final DSD is not yet understood, nor are its possible limitations. This question remains under investigation.
Keywords: Atomization, Curvature, Liquid-gas surface, complex morphology

Influence of Gas-Liquid Impingement Angle on Internal and External Flow Characteristics in an Internal-Mixing Twin-Fluid Atomizer

Saran S. Dharan*[1], Martin Raju[1], Vinil Kumar R.R.[1], S.L.N. Desikan[2], Aravind Vaidyanathan[1]

[1]Indian Institute of Space Science and Technology, Thiruvananthapuram, Kerala, India
[2]Vikram Sarabhai Space Centre, Thiruvananthapuram, Kerala, India
*Corresponding author: saran96.d@gmail.com

Abstract
This study examines the effect of gas–liquid impingement angle on internal two-phase structure and external atomization in an internal-mixing twin-fluid (effervescent) atomizer. Three optically accessible quartz injectors with impingement angles of 65°, 90°, and 108° are investigated using high-speed shadowgraph. The transparent configuration enables simultaneous visualization of internal flow topology and downstream spray development. Results demonstrate that impingement angle significantly modifies internal mixing dynamics, alters regime transitions, and directly governs the primary breakup behaviour.

Introduction
Internal-mixing twin-fluid (effervescent) atomizers are widely employed in propulsion, combustion, and industrial spraying due to their ability to generate fine sprays at relatively low liquid injection pressures. Atomization quality is fundamentally governed by the two-phase flow regime established within the mixing chamber. However, conventional metallic injectors do not permit direct correlation between internal flow topology and external spray characteristics. The present study addresses this limitation using an optically accessible quartz injector to isolate the influence of gas–liquid impingement angle on internal mixing dynamics and atomization performance.

Experimental Methodology
Experiments are conducted using a fused silica (quartz) internal-mixing atomizer (Fig. 1) comprising a central liquid port (d_l = 1.0 mm) and an impinging gas channel (d_g = 0.5 mm). Three configurations with gas-liquid impingement angles (θ) of 65° (co-flow), 90° (transverse flow), and 108° (counter-flow) are examined. Water is injected at a constant mass flow rate of 1.08 g/s (Re_l = 1618), while nitrogen flow rate is varied to achieve gas-to-liquid ratios by mass (GLR) of 1.0, 2.5, 5.0, 7.5, and 10.0 %, spanning multiple internal flow regimes. High-speed shadowgraph imaging (10,000 fps, 1 μs exposure) is used to capture the simultaneous internal and external flow evolution.

Results and Discussion
High-speed imaging reveals distinct internal mixing and atomization regimes governed by both GLR and impingement angle. In the co-flow configuration (65°) at low GLR, large gas bubbles form and gradually disintegrate near the exit. Whereas the transverse (90°) and counter-flow (108°) configurations exhibit more intense bubbly structures with faster breakup and stronger upstream interaction (Fig. 2). Enhanced gas penetration into the liquid port induces oscillatory motion within the mixing section and accelerates regime transition. The counter-flow geometry shows the earliest transition to vigorous slug flow, resulting in enhanced atomization at identical liquid mass flow rate. This configuration also requires higher injection pressure, producing a larger gas-side pressure drop and thereby intensifying primary breakup.

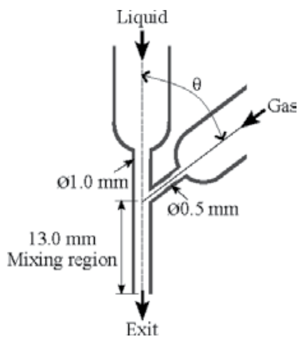


Figure 1. *Atomizer geometry*

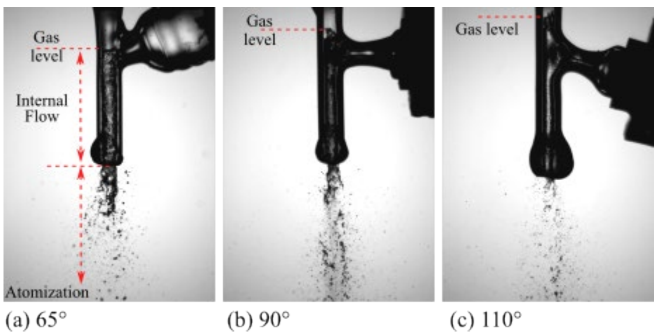


Figure 2. *Instantaneous images of the geometries at GLR = 7.5%.*

Fractal Dimension Analysis of a 2D Liquid-Jet Interface in DNS

GuangnianJi*[1,2],YashKulkarni[2],StéphaneZaleski[2]

[1]ÉcolePolytechnique,InstitutPolytechniquedeParis,RoutedeSaclay,91128Palaiseau Cedex,France
[2]SorbonneUniversitéandCNRS,UMR7190,InstitutJeanLeRond∂'Alembert,75005Paris, France
*Correspondingauthor:guangnian.ji@polytechnique.edu;kulkarniyash2398@gmail.com;

Abstract
Liquid-jet atomization involves a multiscale rearrangement of vortical motion and liquid–gas topology, in which a connected jet is stretched, corrugated, and fragmented into ligaments and droplets. We analyse this process using two-dimensional volume-of-fluid direct numerical simulations of a pulsed liquid jet in Basilisk, with adaptive mesh refinement. Box-counting is applied both to the reconstructed liquid–gas interface and to vorticity-threshold contours. For the reference case Rel = 5800, W eg = 200, and t = 3.0, the full interface has an effective dimension Df ≃ 1.33. This value becomes nearly stationary after the initial breakup stage, but it is not a universal exponent: it depends on the finite observation range and on the AMR-resolved interface geometry. Resolution tests show that the droplet-size distribution, especially at small diameters, is controlled by the finest resolved scale and by the refinement criterion. To relate this interfacial geometry to the flow field, we define a vorticity interface Γω (α) = ∂{|ωz |/ωrms > α}. The vorticity spectrum identifies an intermediate active range, over which the box-counting dimension of Γω is close to 5/3 near α = 1. Higher thresholds isolate smoother vortex-core boundaries and drive the dimension toward unity. Overall, the results show that fractal dimension is best used as a finiteresolution, scale-dependent diagnostic linking AMR-resolved breakup morphology, droplet statistics, and cascade-like vortical structures.

Keywords: Liquid jet; DNS; VOF; AMR; fractal dimension; vorticity interface

Spray characteristics of a liquid jet in crossflow: Insights for a gas turbine and high-speed air-breathing engines

RamanaKukkarasi*[1,2],SivakumarDeivandren[3],RaghuramanNGovardhan[1]

[1]DepartmentofMechanicalEngineering,IndianInstituteofScience,Bangalore,India
[2]CORIALaboratory,Saint-Étienne-du-Rouvray,Rouen,France
[3]DepartmentofAerospaceEngineering,IndianInstituteofScience,Bangalore,India
*Correspondingauthor:kukkarasir@alum.iisc.ac.in

Abstract
The spray characterisation is essential for preliminary design and modelling of the combustion of highspeed aero engines, such as gas turbine, turbojet and ramjet/scramjet engines. Moreover, a wet compression concept by inter-stage spray injection into an axial compressor of a gas turbine engine has been proven to be an effective strategy to enhance compressor performance. The present study experimentally investigates the spray characteristics of a transverse liquid jet injection into sub-and supersonic crossflow, relatively similar to the fuel injection configuration of a high-speed aero engine's propulsion system. A wall-mounted water jet with an orifice diameter, D = 1 mm is injected into a crossflow of air with different velocities (U_∞ = 160, 215 & 580 m/s) in a wind tunnel. The spray droplet images near the vicinity of a spray core location are acquired using the Particle/Droplet Image Analysis (PDIA) technique at a streamwise distance of x = 90D from the jet injection location to measure the droplet characteristics for a momentum flux ratio, J = 9.7 condition. The quantitative spray droplet number, volume & velocity distributions are estimated for each crossflow U_∞ case. Further, the Sauter mean diameter (SMD), the mean diameter and the mean velocity of the droplets are estimated to present variations in the mean spray characteristics with U_∞ . The mean diameter and the SMD are observed to decrease with increasing crossflow velocity, U_∞ , due to enhanced atomization. The spray distribution outcomes of the present study are discussed in light of how they would affect the combustion of high-speed aero engines in terms of droplet evaporation and residence time scales.
Keywords: Jet in crossflow, droplet size distribution & evaporation, high-speed aero engines

Assessment of Batch Ultrasonic Atomisation for Submicron Seawater Aerosol Generation in Marine Cloud Brightening

Mohammadreza Sedighi*[1], Stuart MacLennan[1], Luke P. Harrison[1], Ceylena Holloway[1], Daniel P. Harrison[1]

[1] Reefs and Oceans Research Cluster, National Marine Science Centre, Southern Cross University, Coffs Harbour, 2450, Australia
*Corresponding author: mohammadreza.sedighi@scu.edu.au

Abstract
Marine Cloud Brightening (MCB) requires the high-rate production of submicron salt particles. This study investigates batch ultrasonic atomisation of saline water (3.5% NaCl) as a potential method. A 1.6 MHz commercial atomiser was employed, and dry particle size distributions were directly measured 9 m downstream using electrical mobility and aerodynamic particle sizing instruments to resolve submicron particle distributions. The results show a multimodal, polydisperse distribution ranging from ~160 nm to ~6 µm with a geometric standard deviation (GSD) of 1.84. The measured numberand volume-median diameters were ~640 nm and ~1160 nm, respectively. The total production rate was on the order of 108 particles s-1 with ∼80% of aerosols within submicron range. Comparing these results to classical scaling laws reveals that established models significantly overestimate particle size for saline solutions under far-field dry sampling conditions. Correction factors of 0.65 for the Lang (1962) number-median model and 0.42 for the Yasuda (2005) volume-median model were identified. While scaling suggests that frequencies ≥10 MHz could theoretically achieve the MCB target size of 200 nm, the estimated production rate (i.e., ~1010 particles s-1) still presents a major bottleneck. Reaching the production rate targets required for MCB (≥1015 particles s-1) would still require several orders of magnitude more individual units, presenting a significant challenge for large-scale deployment.
Keywords: Marine Cloud Brightening, Ultrasonic Atomisation, Sub-micron Saline Aerosols, Scaling Laws

Spray and Evaporation Characteristics of Gasoline-Methanol Blends from a Multi-hole GDI Injector under Engine-like Conditions

Dongwoo Ryu[1], Gyuhan Bae[1], Seoksu Moon[*1]

[1]Department of Mechanical Engineering, Inha University, Republic of Korea
*Corresponding author: ss.moon@inha.ac.kr

Abstract
Methanol, the simplest synthetic form of e-fuels, has been regarded as a promising drop-in fuel for conventional gasoline engines by blending it with conventional gasoline. Understanding the influence of fuel thermophysical properties on engine operation is considered a prerequisite in adopting such alternative fuels. As these effects fundamentally manifest through governing the underlying injection and mixture formation processes, spray-level investigation is essential for a mechanistic understanding, yet remains limited. In this study, macroscopic spray and evaporation characteristics of gasoline-methanol blended fuels injected from a multi-hole GDI injector are investigated under conditions simulating earlyand late-injections in engines. Spray visualization was conducted via shadowgraph imaging, with spray penetration and spray angle quantified for both liquid and vapour phases, alongside the liquid-to-vapour penetration ratio. Under the early injection condition with the ambient conditions of 298 K and 0.1 MPa, variations in methanol content yielded no significant changes in both macroscopic spray geometry and evaporation behaviour, despite differences in physical properties. Under the late injection condition with the ambient conditions of 423 K and 1.0 MPa, the vapour-phase spray structure remained largely insensitive to methanol content. The liquid-phase penetration and spray angle, however, increased progressively with methanol blending ratio, suggesting retarded evaporation. The mechanisms related to these observations were discussed in detail with a focus on the nozzle internal flow characteristics of multi-hole GDI injectors and physical properties of fuels such as viscosity, distillation characteristics, and latent heat of vaporization.
Keywords: Gasoline-Methanol blends, Multi-hole GDI injector, Spray characteristics, Evaporation

Spray Structure and Atomisation Mechanisms of Flash-Boiling Methanol under Reduced Ambient Pressure and Split Injection

Vikas Sharma, Guillaume de Sercey, Angad Panesar and Steven Begg*

Advanced Engineering Centre, School of Architecture, Technology and Engineering,
University of Brighton, United Kingdom
*Corresponding author: S.M.Begg@brighton.ac.uk

Abstract
Methanol is a promising low-carbon fuel for future propulsion systems. However, its high latent heat of vaporisation and poor auto-ignition characteristics increase the dependence on efficient spray atomisation and mixture preparation. In this study, the spray structure and atomisation mechanisms of flash-boiling methanol were experimentally investigated using an eight-hole ECN Spray M injector under reduced ambient pressure conditions. Experiments were conducted at injection pressures of 50–200 bar, fuel temperatures between 70 and 90 °C, and atmospheric absolute gas pressures of 1.0, 0.5 and 0.3 bar, corresponding to a saturation pressure ratio (Rp = $Psat$ /$Pamb$) in the range of 0.8 to 3.2. Highspeed shadowgraph imaging was employed to investigate spray regime transition, plume interaction, spray collapse and injector-tip wetting across the reported range and for the cases of pulsed, split injection scenarios commonly encountered in direct injection combustion strategies. Results showed that an increase in pressure ratio promoted the transition from non-flashing to strong flash-boiling regimes. intensifying vapour generation and inward plume entrainment. Under strong flash-boiling conditions, neighbouring plumes merged, forming a central spray structure preceded by an axially protruding tip, extending by up to 6.5 mm beneath the main spray core of approximately 20 mm in length for a fuel pressure of 100 bar and Rp = 3.2. Significant plume widening was observed from 0.04 ms after the start of injection due to rapid radial vapour expansion and enhanced plume interaction. Micro-scale imaging further revealed vapour bubble formation, growth and bursting that resulted in poorly atomised large droplets and ligament structures near to the injector tip, following the needle closing phase. Sequential pulsed injection experiments revealed that temperature induced effects within the liquid fuel captive within the nozzle cavity, leads to internal vapour formation. accelerated external flashing and jet expansion that influenced the spray penetration during repeated injections. Consistently, the third injection exhibited approximately 4% lower penetration than the first injection under continuous flashing conditions. The results provide insight into flash-boiling-assisted methanol injection strategies for improved mixture preparation in future low-carbon and dual-fuel engine applications.
Keywords: Flash boiling, methanol, atomisation, spray structure, multiple injection

Quantitative Visualization of Forced Vapor Advection using an Accessible BOS Approach

Ben Ben David Holtzer*[1], Leonid Tartakovsky[1]

[1]Technion Internal Combustion Engines Laboratory, Technion – Israel Institute of Technology, Israel
*Corresponding author: bb@campus.technion.ac.il

Abstract
Recent studies highlight the critical role of gaseous jet-lubricant interactions in generating particulate matter within hydrogen internal combustion engines. While gas-into-gas discharges have been extensively studied using costly instrumentation, the multiphase mechanisms of jet-film interaction and vapor advection remain under-investigated. This work proposes an innovative, cost-efficient framework for analyzing forced advection employing Background-oriented Schlieren (BOS) and open-source software. The developed framework allows for precise quantitative concentration analysis and is applicable to a broad spectrum of transport phenomena, including evaporation and mixing studies.

Keywords: Jet-film interaction; Forced advection; Evaporation; Background-oriented Schlieren

Experimental Investigation of Spray Atomization and Droplet Dynamics of Hydrotreated Vegetable Oil and Jet A-1 Using an Air-Assisted Atomizer

Pasquale di Gloria[1], Ines Alexandra Ferrão[2 3], Guido Marseglia[14]*, Ana Sofia Moita[3 5], Maria Grazia De Giorgi[1]

[1] Department of Engineering for Innovation,
University of Salento, Via per Monteroni, 73100 Lecce, Italy
[2] Aeronautics and Astronautics Research Center (AEROG) – LAETA, Universidade da Beira Interior, Covilhã, Portugal
[3] IN+ Center for Innovation, Technology and Policy Research, Instituto Superior Técnico, Universidade de Lisboa, 1049-001 Lisboa, Portugal
[4] Universidad de Sevilla, Departamento de Matemática Aplicada I, Escuela Técnica Superior de Ingeniería Informática, Universidad de Sevilla,Avenida Reina Mercedes, s/n, 41012 Sevilla,
España
[5] CINAMIL – Centro de Investigação Desenvolvimento e Inovação da Academia Militar, Academia Militar, Instituto Universitário Militar, Lisboa, Portugal
*Corresponding author: anamoita@tecnico.ulisboa.pt

Abstract

Hydrotreated Vegetable Oil (HVO) is a sustainable alternative to conventional aviation fuels due to its renewable origin, low aromatic content, and favourable combustion properties. However, its higher viscosity compared to Jet A-1 may impair atomization and spray development, which are critical for efficient gas turbine combustion. The present study is part of a wider investigation to infer on the potential benefits of using biofuels added with nanoparticles for aviation. Building upon our previous research, , the present paper particularly focuses on the atomization characteristics of a biofuel (HVO) and on the effect of the addition of aluminum nanoparticles on droplet size and velocity distributions, mostly obtained from Phase-Doppler measurements and high-speed visualization. The effect of the nanoparticles was inferred adding concentrations of aluminum nanoparticles (40nm of average size) of 0.5 wt. %. The sprays are obtained using an air-assisted atomizer. Results show the predominance of smaller droplets and higher axial velocities in the central spray, while larger droplets concentrate near the periphery for the sprays resulting from the atomization of both Jet A-1 and HVO. Adding 0.5 wt.% aluminum nanoparticles increases the diameter of the atomized droplets but not in a significant way, while HVO's primary physical properties remain unchanged. These findings highlight the dominant role of fuel viscosity and AFR in spray formation and suggest some atomization benefits from low-concentration nanoparticle additives in air-assisted atomizers.

Keywords: Sustainable aviation fuels; HVO; Phase Doppler measurements, nanoparticles; spray atomization

Drag and Shape Oscillations in Post-Coalescence Droplet Dynamics

RoiBar-On[1],J.BarryGreenberg[1],MatthiasIbach[2],AlumahArad[3],ManishKumar[2],Visakh Vaikuntanathan[4],DavidKatoshevski[3],BernhardWeigand[2]

[1]FacultyofAerospaceEngineering,Technion–IsraelInstituteofTechnology,Haifa,Israel
[2]InstituteofAerospaceThermodynamics(ITLR),UniversityofStuttgart,Stuttgart,Germany
[3]DepartmentofCivilandEnvironmentalEngineering,Ben-GurionUniversityoftheNegev, Beer-Sheva,Israel
[4]DepartmentofMechanicalEngineering,ShivNadarInstitutionofEminence(Deemedtobe University),UttarPradesh,India
*Correspondingauthor:roi1812@technion.ac.il

Abstract
The post-coalescence dynamics of liquid droplets play an important role in spray and atomization processes, particularly under moderate-to-high Reynolds number conditions where transient shape deformations significantly influence drag and settling behavior. In this study, we investigate the temporal evolution of a deforming droplet following coalescence, focusing on initial droplet velocities that correspond to initial Reynolds numbers in the range Re = 200 − 500. A reduced-order framework is developed in which the droplet is modeled as an evolving ellipsoid with time-dependent semi-axes obtained from direct numerical simulation (DNS) data. The aspect ratio dynamics are approximated using a damped oscillatory model, whose parameters are interpolated across several initial conditions (droplet initial velocity) corresponding to different Reynolds numbers, to construct a continuous representation of droplet deformation. This enables the reconstruction of timeresolved geometric properties, including volume-equivalent diameter and projected area. The aerodynamic response is modeled using the empirical Ganser drag correlation as a quasi-steady approximation for the instantaneous ellipsoidal droplet shape. Since the original Ganser correlation was developed for rigid non-spherical particles, its use here is treated as an approximate closure for incorporating time-dependent geometric effects into the far-field droplet dynamics. The droplet velocity evolution is computed using a piece-wise analytical solution, based on a local quasisteady approximation, and a numerical solution. A detailed comparison reveals that, when geometric effects are fully time-resolved, the local piecewise analytical solution closely matches the numerical solution. The results provide a computationally efficient framework for predicting droplet dynamics in regimes where deformation effects cannot be neglected.
Keywords: droplet dynamics, deformation, drag, modeling, free-fall, far-field viewpoint, oblate particle, prolate particle

Role of Surface Energy Flux in the Formation of Circular Hydraulic Jumps

HosseinAskarizadeh*[1],MohammadGhanem[1],QiyaShu[1],WilkoRohlfs[1]

[1]InstituteofHeatandMassTransfer,RWTHAachenUniversity,Germany
*Correspondingauthor:askarizadeh@wsa.rwth-aachen.de

Abstract

A vertical liquid jet impinging on a flat plate forms a radially spreading thin film followed by a circular hydraulic jump. Traditionally, gravity has been considered the dominant mechanism governing the formation and position of the jump. More recently, however, it has been proposed that surface tension and capillary effects, rather than gravity, control the formation of circular hydraulic jumps. This claim was based on observations showing an approximately constant jump radius during the development of the flow on plates with different orientations. To support this interpretation, an additional term representing the energy flux associated with surface tension was introduced into the mechanical energy balance for an annular control volume upstream of the jump. This proposal has stimulated considerable discussion within the fluid mechanics community. One line of criticism argues that surface tension should instead be represented through the Laplace pressure acting normal to the interface. Subsequent work suggested that such a formulation neglects tensile forces acting along the control-volume boundary arising from velocity differences between the flowing liquid and the contour of the interface. In response, it has been argued that the liquid velocity used in the surface tension work should be replaced by the velocity of the contour itself, since the interface is typically assumed to be infinitely thin, massless, and free of inertia. Consequently, no consensus has yet been reached regarding the correct formulation of the surface-tension-related energy flux. In the present study, fully resolved numerical simulations of a vertically impinging liquid jet are performed to evaluate all terms of the proposed mechanical energy balance. The simulations allow a detailed assessment of the individual energy flux contributions and enable a direct comparison between the competing formulations. The results clarify the role of surface tension in the overall energy budget and provide insight into which formulation yields a physically consistent description of circular hydraulic jump formation.

Keywords: Thin film flow, Circular hydraulic jump, Surface tension, Mechanical energy balance

The Role of Droplet Vaporization in the Response of a Laminar Spray Flame to Acoustic Forcing

Nino A. Leder*[1], Harish S. Gopalakrishnan[1], Mirko R. Bothien[1]

[1] TASC lab, Institute of Energy Systems and Fluid-Engineering, ZHAW Zurich University of Applied Sciences, Winterthur, Switzerland
*Corresponding author: ledr@zhaw.ch

Abstract

Gas turbine engines used for aircraft propulsion typically involve liquid fuels owing to their high energy density. In these systems, the liquid fuel is atomized, mixed with the air, and then reacts to result in a flame. Strict NO_x regulations have driven gas-turbine manufacturers towards lean-premixed combustion systems. In these systems, fuel is premixed with more air than required for stoichiometric combustion, thereby reducing flame temperatures and hot spots resulting in lower NO_x. These lean-premixed combustors are, however, susceptible to thermo-acoustic instabilities, a destructive feedback loop involving interactions between the flame and the acoustic modes, which can significantly degrade performance and compromise the structural integrity of combustors. Compared with gaseous fuel flames, spray flames introduce additional response mechanisms to acoustic waves due to unsteady droplet transport, number-density-wave formation, unsteady vaporization, and the spatial distribution of liquid and gaseous fuel. The relative importance and phase relation of these mechanisms remain insufficiently understood. This work isolates the role of droplet vaporization in the acoustic response of a laminar spray flame using one-dimensional direct numerical simulations. Atomization, fuel polydispersity, and multi-component evaporation are deliberately neglected in order to focus on the interaction between acoustically forced droplet transport, evaporation, and heat release. The simulations are performed with the well-established solver AVBP using a two-step single-component kerosene-surrogate mechanism. The liquid phase is described by a Lagrangian formulation, and droplet evaporation is modeled with the Abramzon–Sirignano model using modified evaporating Schmidt and Prandtl numbers obtained from zero-dimensional Cantera calculations.
All cases use a fixed droplet temperature of $T_{ptcl} = 300$ K, droplet diameter of $d_{ptcl} = 20\ \mu\mathrm{m}$, gas temperature of $T_{gas} = 400$ K, and total equivalence ratio of $\phi_{tot} = 0.9$. Three liquid-fuel fractions, $\phi_{liq}/\phi_{tot} = 1/3$, $2/3$, and 1, are investigated to vary the strength of number-density-wave formation. For each fuel split, the mean inlet velocity is varied, thereby shifting the flame position relative to the evaporation progress. This changes the effective equivalence ratio at the flame front, the amount of evaporation occurring inside the reaction zone, and the resulting flame thickness and heat-release distribution. The flame is then forced by harmonic inlet-velocity perturbations with an amplitude of 5% of the mean velocity.
The computed flame transfer functions show that the global heat-release response does not follow the monotonic low-pass behavior commonly expected for gaseous premixed flames or for number-density-wave formation alone. Instead, several operating points exhibit a non-monotonic frequency response. To explain this behavior, the heat-release response is decomposed into two physical pathways: an upstream gaseous-fuel pathway, caused by evaporation of convected droplet number-density waves upstream of the flame, and an in-flame evaporation pathway, caused by liquid fuel vaporizing inside the reaction zone. While the individual pathway responses show clearer trends, their complex superposition produces the irregular global flame transfer function.
Phase-averaged fields show that the global response depends on the relative phase between gaseous-fuel fluctuations and liquid-fuel fluctuations near the flame. Constructive interference occurs when both pathways increase the local fuel supply in phase, leading to an amplified heat-release response. Conversely, phase-shifted interactions reduce the global gain through partial cancellation. The phase relation is governed primarily by the convective residence time from injection to the flame, the droplet relaxation and evaporation time scales, and the acoustic forcing frequency. These results demonstrate that in-flame vaporization can play a dominant role in the acoustic response of spray flames and that reduced-order models must account not only for number-density-wave generation, but also for evaporation timing and pathway interference.

Keywords: Spray flames, Droplet vaporization, Thermoacoustics, Flame response, DNS

POSTERS

Regular Posters

Effect Of Energisation Regimes On Spray Characteristics In Electrostatic Atomisation
E.D. Andrews, I. Timoshkin, G. Burt

Modeling Breakup And Solidification Dynamics In Gas Atomization Of Recycled Steel
S. Heidarpour, N.G. Deen, J.J. Remmers, G. Finotello

On Particle Transport Mechanisms In Cylinder Wakes Using A High-Fidelity Eulerian-Lagrangian Framework
R. Boukharfane

Experimental Study Of Heat Transfer In Multiple Droplet Impacts
A.F. Campos-Araya, T. Potaufeux, O. Caballina, M. Gradeck, A.V.S. Oliveira, G. Castanet

Experimental And Numerical Investigation Of Motive Nozzle Positioning In A Two-Phase Water Ejector For High-Temperature Heat Pumps
O.A. Khass, M. Cristofaro, S. Klöppel, P. Stathopoulos

Experimental Investigation On The Influence Of Surface Roughness On Thin Shear-Driven Water Films
J.H. Ruesch, M. Schatz

Atomization In Airblast Atomizers: A Comparison Of Two Numerical Frameworks
R. Forte, C. Hasse

Optical Rapid Compression Machine Test Facility For Low And Zero-Carbon Fuels
S. Begg, G. De Sercey, V. Sharma, A. Panesar

POSTERS

Numerical Investigation Of Non-Premixed Combustion Of Hydrogen- Enriched Methane And Ammonia In A Reverse-Flow Gas Turbine Combustor
R. Renane, R. Allouche, H. Remmal

Critical Transition Of Ethanol Droplets In Vapor-Rich Environments: A Numerical Study Using The REFPROP F8 Helmholtz EOS
R. Coelho, J. Vedovotto, P. Mota, R. Serfaty, G. Sabino

Interferometric Laser Imaging for Droplet Sizing of Flash-Boiling Ammonia Sprays
U. Sonawane, V. Venkatesan, C. Hespel, C. Rousselle

Work-in-Progress Posters

Analytical Modelling Of Finite Amplitude Oscillation Of Viscous Drops In Single Mode
G.E. Cossali, S. Tonini

Optical Investigation Of Cavitation And Flash Boiling In Transcritical Cryogenic Sprays
F. Murphy-Hunt, C. Marlow-Eastwood, M. White, C. Crua

Dynamic Analysis Of A Solenoid-Driven Diesel Injector Using A One-Dimensional Model For DME Fuel
J. Lee

Flow Dynamics And Stability Limits Of Electrodynamic Sprays Through Permeable Electrodes
K. Mehrabi, F.J. Higuera

On The Impact Of Nanoparticle Shape On Droplet Retraction Over Smooth Surfaces
M.R. Vetrano, Q. Dong, Y. Shang, D. Fontanarosa, M. Van Alphen

POSTERS

Three-Layer Gas-Phase Modelling For Numerical Simulation Of Sodium Spray Combustion In A Sodium-Cooled Fast Reactor Compartment
J. Kim, B.J. Kim

Light Field Imaging For Gas-Liquid Interfaces In Multiphase Flows: Bubble/Droplet Sizes
R. Kukkarasi, A. Marszalek, J.-B. Blaisot, M. Brunel

On The Relevance Of Nozzle-Internal Surface Roughness On Flash-Boiling Phenomena Of Liquid Ammonia
P.W. Augustin

Dispersion Gas Effects On Flash-Boiling Atomization And Stability Of Liquid Ammonia Spray Flames
A. Arpert, P. Augustin, L. Heine, A. Maßmeyer, W. Rohlfs, M. Reddemann

Visualization, Measurement, And Analysis Of Spray And Flame Propagation In An Ammonia Direct-Injection Engine
Y. Kim, C. Park, C. Park, Y. Choi

CFD-Guided Geometric Design For Swirling Airflow Control In Ultrasonic Spray Systems
X. Hou, P. Vesely, R. Schick

Sustainable Ultrasonic Atomization With Advanced Spray Analysis For Resource-Efficient Semiconductor Wet Processing
X. Hou, P. Vesely, R. Schick

Flash-Boiling Characteristics Of Methanol Sprays: From Bubble Dynamics To Plume Collapse
V. Sharma

POSTERS

Validation Of Spray Characteristics Of Hydraulic Axial Full Cone Nozzles Using The Smoothed Particle Hydrodynamics (SPH) Method
S. Markus, P. Hellard

Development Of Experimental Test Platforms For Aerospace Propulsion Research: Example Results On 3D-Printed Pintle Injector Sprays And Shock–Droplet Interaction
M. Son, M. Wenderoth, M. Drost, T. Sander, L. Zigan

Numerical Assessment Of Multi-Jet Injection In Turbofan Exhausts
H. Huck, C. Kieffer-Roth, H. Scheffold, B. Enderle, G. Eckel, A. Huber

Stabilization Of Ethanol-In-Oil Emulsion Produced With An Injector Nozzle In-Line With A Microchannel Mixer
A. Oliveira, M. Bailly, J. Bellettre

Spatial Evolution Of Liquid Sheets Formed By Impinging Jets Under Subcooled And Flash-Boiling Conditions
H.A. Gonzalez, B. Kazmierski, L.J. Kapusta

Reduced Gas Consumption And Enhanced Powder Fineness For Additive Manufacturing By High-Temperature Gas Atomization
A.K. Geppert, A. Weicht, S. Evers, L. Achelis, V. Uhlenwinkel, U. Fritsching

Industry Session Posters

Advanced Optical Instrumentation For Spray, Droplet, And Particle Characterization
M.D. Leandro, W.D. Bachalo, K.M. Ibrahim, G.A. Payne

Impact Of Nozzle Shape And Dimensions On Liquid Jet-In-CrossFlow Spray Characteristics For The Design Of A Multi-Staged Fuel Injection System
A. Tomba

POSTERS

A Retrofit Electrostatic Pressure Swirl Atomizer For Improving Industrial Spray Performance Without Nozzle Redesign

E.A. Baraya, I.A. Ibrahim, H.M. Gad

Effect of energisation regimes on spray characteristics in electrostatic atomisation

Ewan Andrews, Igor Timoshkin, Graeme Burt

Institute for Energy and Environment
Department of Electrical and Electronic Engineering
University of Strathclyde
204 George St, Glasgow, UK, G1 1XW

Abstract

Electrostatic atomisation has been used for decades across various industries, including agriculture, painting, and medicine. The application of a high voltage enables production of fine droplets, through the formation of different dripping and jet modes [1]. The various modes are classified by the fragmentation geometry of the liquid at the capillary outlet. Although widely studied, the relationships between injected charge, operating conditions and the characteristics of the jet modes and droplets are not yet fully resolved.

In conjunction with jet and droplet dynamics, the behaviour of electric charge within the liquid plays a critical role in electrostatic atomisation. Induced charge at the liquid surface controls jet behaviour and droplet characteristics [2]. Understanding how injected charge influences jet formation and droplet behaviour is crucial in controlling the atomisation process and predicting droplet response.

This study will investigate parameters and behaviour of the jets and droplets formed at different energisation regimes. Different electrode topologies for charging the liquid will be investigated. Charge delivered into the liquid and droplet parameters following atomisation will be analysed. This experimental campaign will assess liquid charge retention under different energisation and hydrodynamic conditions, providing broader insight into how charge governs the behaviour and characteristics of the formed droplets. Further understanding will thereby be provided into the electrostatic atomisation process, which will aid further development of practical applications of the method to droplet formation.

References

[1] J. Rosell-Llompart, J. Grifoll, and I. G. Loscertales, "Electrosprays in the cone-jet mode: From Taylor cone formation to spray development," *Journal of Aerosol Science*, 125, pp. 2-31 (2018).

[2] Y. Wang, M. K. Tan, D. B. Go, and H.-C. Chang, "Electrospray cone-jet breakup and droplet production for electrolyte solutions," *Europhysics Letters* 99, 6 (2012).

Modeling Breakup and Solidification Dynamics in Gas Atomization of Recycled Steel

Sotoudeh Heidarpour*[1,2], Niels G. Deen[1,2], Joris J.C. Remmers[3,4], Giulia Finotello[1,2]

[1]Power & Flow, Department of mechanical Engineering, Eindhoven University of Technology
[2]Eindhoven Institute for Renewable Energy Systems (EIRES), Eindhoven University of Technology
[3]Mechanics of Materials, Department of Mechanical Engineering, Eindhoven University of Technology
[4]Eindhoven Artificial Intelligence Systems Institute (EAISI), Eindhoven University of Technology
*Corresponding author: s.heidarpour@tue.nl

Abstract

The direct reuse of scrap steel for producing metal powders offers a potential route toward circular and low-carbon steel manufacturing. Gas atomization is a key technology for generating powders suitable for additive manufacturing; however, its application to recycled and compositionally variable feedstock remains challenging. In particular, the spray dynamics governing molten jet disintegration, secondary droplet breakup, and rapid cooling are not yet fully understood for scrap-based systems, leading to wide particle size distributions and inefficient process design. Addressing these challenges requires predictive tools that link atomization physics to powder quality.

This work presents the development of a Computational Fluid Dynamics (CFD) framework for gas atomization of recycled steel, focusing on secondary breakup and droplet solidification. An Eulerian–Lagrangian approach will be employed, where the gas phase is resolved as a continuum and molten metal droplets are tracked as discrete particles. Different breakup models are evaluated for their applicability to high-temperature metal sprays, and a thermal model is incorporated to capture droplet cooling and phase change. The OpenFOAM enables systematic parametric studies of key operating conditions such as gas pressure and gas-to-melt flow ratio. By linking operating parameters to breakup behavior and solidification outcomes, the model identifies the conditions that minimize satellite formation and narrow the particle size distribution, thereby supporting physics-based optimization of sustainable powder production.

Keywords: Recycled steel, sustainable powder production, gas atomization, CFD, droplet breakup

On particle transport mechanisms in cylinder wakes using a high-fidelity Eulerian–Lagrangian framework

Radouan Boukharfane*[1]

[1] College of Computing, Mohammed VI Polytechnic University (UM6P), Benguerir, Morocco
*Corresponding author: radouan.boukharfane@um6p.ma

Abstract

The transport of suspended particles in the wake of a circular cylinder depends sensitively on particle density, size, and the topology of the carrier flow. Classical studies have shown that neutrally buoyant, monodisperse particles released inside a steady recirculating wake migrate outward toward a stable limit-cycle trajectory near the wake boundary [1]. In denser suspensions, particle–particle interactions generate velocity fluctuations that push particles across the separatrix, ultimately producing the particle-depleted wake observed experimentally. At much smaller scales, ultrafine nanoparticles disperse primarily through Brownian motion and turbulence, which first trap them at the periphery of von Kármán vortices and later mix them into their cores [2]. The present work brings these mechanisms together by examining how density ratio and size distribution influence particle transport in cylinder wakes. We vary the particle density from $\rho_p = 100$ to $800\ \mathrm{kg/m^3}$ and introduce controlled polydispersity around $100\ \mu\mathrm{m}$. All simulations are performed at a Reynolds number below the onset of vortex shedding, ensuring a steady recirculating wake. A fixed particle mass-flow rate allows the study to isolate density and size effects from changes in particle loading. The computations rely on a high-fidelity Eulerian–Lagrangian solver capable of resolving strong gas–particle coupling, finite-size effects, and accurate particle–boundary interactions [3]. Increasing particle density raises the Stokes number and accelerates outward migration

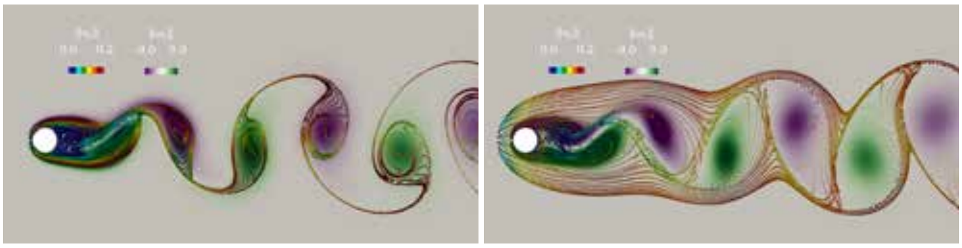


Figure 1. *Instantaneous visualization of particle-laden flow past a circular cylinder for two particle densities, $\rho_p = 100$ (left) and $\rho_p = 800$ kg/m^3 (right). Colors denote fluid vorticity magnitude and streamfunction, while dots represent Lagrangian particle positions.*

from the vortex cores. As shown in Fig. 1, heavy particles reach the separatrix more rapidly, follow more inertial and smoother trajectories, and respond weakly to small-scale fluctuations. This leads to faster depletion of the inner recirculation region for $\rho_p = 800$ compared with the more tightly entrained paths at $\rho_p = 100$. Under polydispersity, the same mechanism applies: larger or heavier particles escape the wake sooner, while smaller or lighter ones remain trapped longer, generating size-dependent stratification and enhancing wake–free-stream exchange. These trends highlight how density ratio and size jointly govern particle organization in vortical flows and pave the way for studying unsteady wakes and reacting or evaporating particles.

References

[1] H. Haddadi, S. Shojaei-Zadeh, K. Connington, and J. F. Morris, "Suspension flow past a cylinder: particle interactions with recirculating wakes," Journal of Fluid Mechanics **760**, R2 (2014).

[2] N. S. Keita, A. Mehel, F. Murzyn, A. Tanière, B. Arcen, and B. Diourte, "Numerical study of ultrafine particles dispersion in the wake of a cylinder," Atmospheric Pollution Research **10**, 294–302 (2019).

[3] F. E.-Z. El Hamra and R. Boukharfane, "Development and assessment of algorithms for DEM-LES simulations of fluidized bed," Particuology **83**, 241–257 (2023).

Experimental Study Of Heat Transfer In Multiple Droplet Impacts

Alavaro F. Campos-Araya[1], Thomas Potaufeux[2], Ophélie Caballina[2], Michel Gradeck[2], Arthur V. S. Oliveira[1], Guillaume Castanet[2]

[1]University of São Paulo, São Carlos, Brazil
[2]Université de Lorraine, Nancy, France

Abstract

Spray cooling offers high heat-removal capacity but remains difficult to model due to complex droplet-surface and droplet-droplet interactions. In particular, collective effects arising from closely spaced or temporally staggered droplet impacts cannot be inferred from isolated droplet behavior. This work experimentally investigates the hydrodynamics and heat-transfer characteristics of single droplets and closely spaced droplet pairs impacting a heated, initially dry surface. Synchronized high-speed shadow imaging and infrared thermography are used to simultaneously resolve impact dynamics and transient substrate temperature fields. Local and global heat fluxes are reconstructed using an inverse heat conduction method, enabling quantitative comparison between interacting and non-interacting droplets over wall temperatures ranging from non-boiling to intense nucleate boiling conditions. For simultaneous impacts, droplet collision during spreading generates a vertical jet and reduces the initial contact area, leading to a lower heat flux than that of two isolated droplets (Figure 1). At later times, droplet merging produces elongated wetted footprints that enhance heat transfer during receding and boiling. Wall temperature strongly modulates these effects through its influence on boiling-induced contact-line mobility. Introducing a time delay between impacts further alters the interaction mechanisms, promoting asymmetric spreading, secondary droplet formation, and prolonged heat-transfer enhancement. These results identify distinct heat-transfer regimes governed by spreading, receding, and boiling, and provide new insight into multi-droplet effects relevant to spray cooling applications.

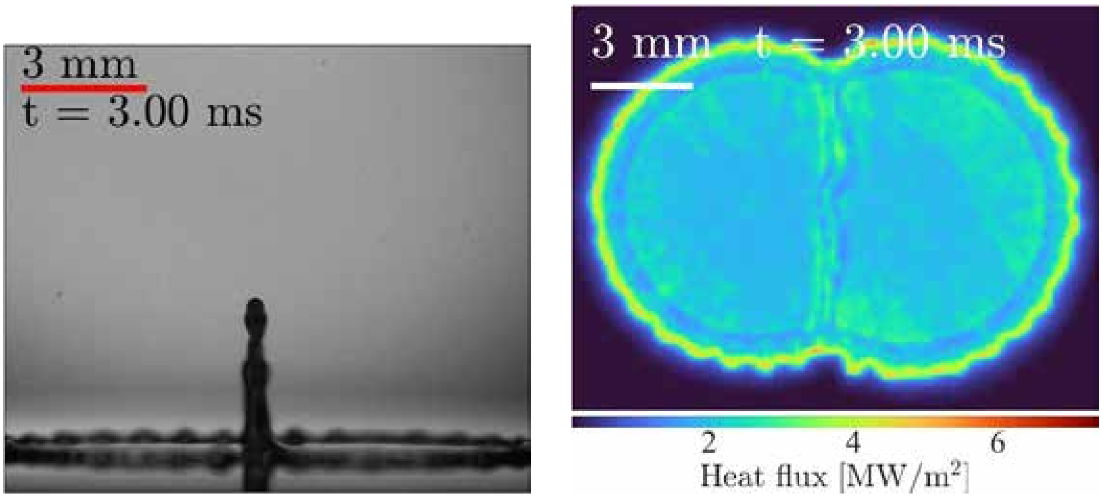


Figure 1. Simultaneous droplet pair impact (We = 130, T_w = 120°C): side-view shadowgraphy, substrate wall temperature, and wall heat flux.

Experimental and Numerical Investigation of Motive Nozzle Positioning in a Two-Phase Water Ejector for High-Temperature Heat Pumps

Omar Abu Khass*[1], Marco Cristofaro[2], Steffen Klöppel[1], Panagiotis Stathopoulos[1]

[1]Institute of Low-Carbon Industrial Processes, German Aerospace Centre (DLR), Germany
[2] Institute of Software Methods for Product Virtualization, German Aerospace Centre (DLR), Germany
*Corresponding author: Omar.AbuKhass@dlr.de

Abstract

Two-phase water ejectors are promising components for high-temperature heat pump applications, where high-pressure liquid water is injected through a motive nozzle to entrain and compress superheated steam. The resulting liquid jet breakup, mixing, and phase interaction inside the ejector are closely linked to interphase momentum transfer, making ejector operation highly sensitive to nozzle geometry. However, the influence of motive nozzle positioning on entrainment behaviour, mixing quality, and overall ejector performance remains insufficiently understood, particularly from a combined experimental and numerical perspective. This study focuses on how variations in motive nozzle length, and the associated nozzle exit position inside the ejector body, influence the effective suction nozzle throat diameter, steam entrainment, and outlet steam quality. Two motive nozzle configurations with different lengths and exit positions were investigated experimentally and numerically. Both configurations inject high-pressure water into the ejector, leading to distinct geometric interactions with the suction nozzle. Computational fluid dynamics simulations were performed using a compressible two-phase mixture approach to analyse entrainment behaviour, mixing intensity, and phase distribution. In parallel, experimental tests were conducted on an ejector test stand to validate the numerical trends and assess real operating performance. Preliminary results indicate that positioning the motive nozzle further inside the ejector reduces the effective suction nozzle throat diameter, enhances steam entrainment, intensifies mixing, and improves outlet steam quality. The combined experimental–numerical findings provide valuable insights into how motive nozzle positioning governs entrainment and mixing in two-phase water ejectors for high-temperature heat pump applications.

Keywords: Two-phase ejector; CFD; High-temperature heat pump; Experimental investigation.

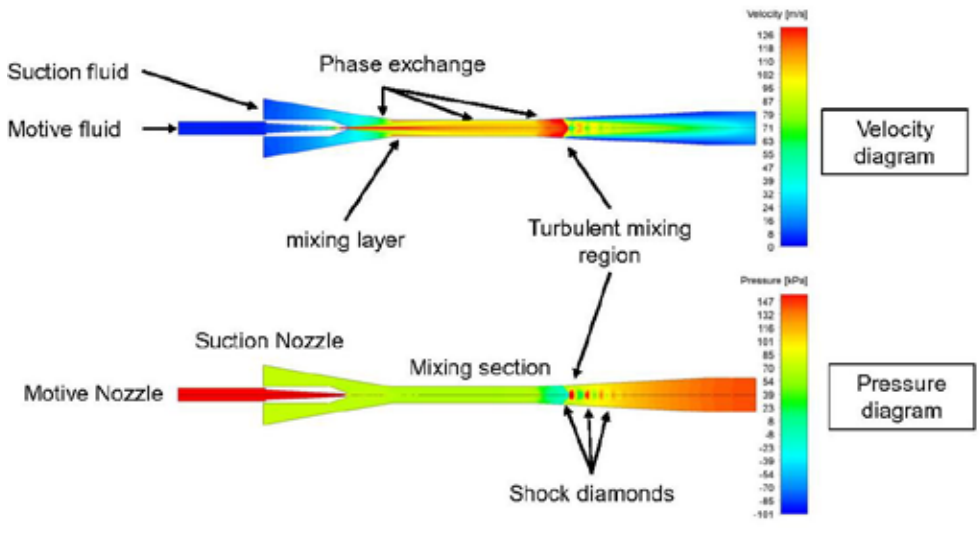


Figure 1. *CFD simulations of two-phase water ejector, from Abu Khass et al. ASME Turbo Expo* (2025)

Experimental investigation on the influence of surface roughness on thin shear-driven water films

Jonas H. Ruesch*[1], Markus Schatz[1]

[1]Chair of Fluid Machinery for Energy Technology, Helmut-Schmidt-University, University of the Federal Armed Forces Hamburg, Germany
*Corresponding author: rueschj@hsu-hh.de

Abstract

Thin shear stress-driven water films can be found in a wide variety of applications, such film atomizers, low-pressure stages of steam turbines or in the first stages of gas turbines in the case of high fogging. The challenge in researching these films is the complex relationship between the turbulent boundary layer and the three-dimensional wavy film, as they influence each other. However, available literature mostly features time resolved but localized pointwise measurements of film behavior. Moreover, experiments are often done in low-speed gas flows, which are not characteristic for various applications. A new experimental test rig, which allows temporal and spatial resolution of films under turbomachinery-like conditions was built at the Helmut-Schmidt-University in Hamburg, Germany. The test rig can be operated at air velocities exceeding 100m/s at Film Reynolds numbers up to 200. The shear-driven water films are investigated using a novel measurement technique based on the light absorption of a water-ink mixture, which is illuminated by a white light source. The light intensity attenuation, which is proportional to the film thickness and ink concentration, is recorded by a high-speed camera, this permits films to be recorded both temporally and spatially resolved, allowing to identify and investigate small and large waves as shown in FIG. 1. This contribution extends previous experimental research [1] by investigating the influence of surface roughness on the water film behavior. The effect of roughness on the film is critical in applications such as additive manufacturing of turbine blades or surface erosion caused by droplet impact. To enable the use of the light absorption method, the experiments were conducted using glass panes with a predefined surface roughness. The surface geometry consists of structured cuboids, which have been etched into the glass. This enables the examination of films with an average surface roughness of less than 77µm. The results were compared to the case of a smooth wall.

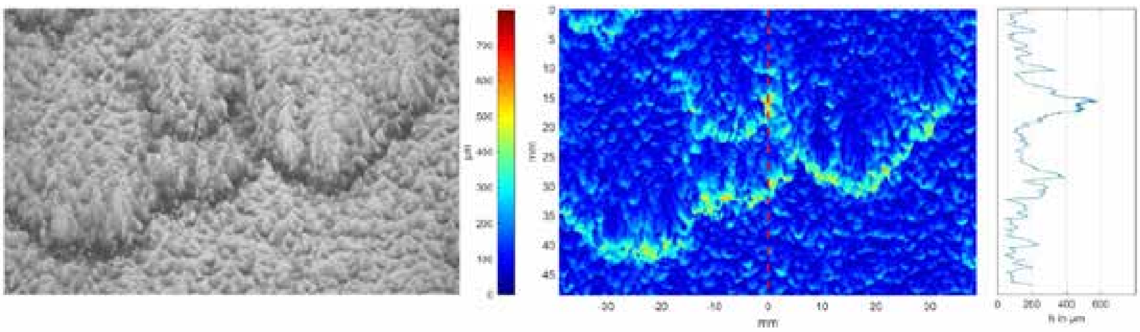


FIG 1: (Left) image of a water ink-mixture forming a wavy film due to air flow from topside at a velocity of 75 m/s and film Reynolds number of 120. (Middle) Surface plot of the evaluated image showing the film thickness in µm. (Right) Line plot from the red dashed line in the surface plot, showing a cross section of the wavy film.

References

[1] Ruesch, J.H., Behn, P.J., Nied, E., Schatz, M., Weigand, B. Temporal and spatial experimental investigation on thin shear-driven fluid films under turbomachinery conditions. *CEAS Aeronaut J* (2025). https://doi.org/10.1007/s13272-025-00904-4.

First Characterisation of a typical Aero-engine pre-filmer atomizer through Surface Curvature Analysis

Anbarasan Sekar*[1], Saïd Idlahcen[1], Stefan R Harth[2], Julien Reveillon[1], Benjamin Duret[1], Jean-Bernard Blaisot[1], François-Xavier Demoulin[1]

[1] Univ Rouen Normandie, INSA Rouen Normandie, CNRS CORIA, F-76000, Rouen, France
[2] Engler-Bunte-Institute, Karlsruhe Institute of Technology, 76131 Karlsruhe, Germany
*Corresponding author: anbumsr@gmail.com

Abstract
Atomization processes of a typical aero-engine pre-filmer atomizer are investigated through surface curvature analysis. Surface Curvature Analysis (SCA) is a robust method for describing the atomization process. Atomization transforms a simple liquid flow (such as a liquid sheet or a cylindrical jet) into final spherical droplets. Throughout this transition, the surface morphology becomes highly complex; however, it can be quantitatively characterized using SCA. This approach makes it possible to monitor the atomization process, maintain sensitivity to minor variations within the flow, and ultimately predict the resulting equivalent droplet size distribution. It is important to note that the standard Sauter Mean Diameter (D_{32}), defined as the ratio of liquid volume to surface area, cannot be directly determined from 2D measurements without specific symmetry hypotheses. Such assumptions, however, are often invalid in the dense region of atomization where morphologies are highly irregular. Consequently, the D_{21} parameter—defined as the ratio of the surface area covered by the liquid to its contour length—is utilized as a more suitable metric for this analysis. The recent progress in the surface curvature analysis [1, 2] made possible for the application of the method to two-dimensional or line-of-sight imaging. The experimental data are acquired using the diffused back-illumination imaging technique with a microscopic arrangement. The effects of external co-flow in the pre-filmer atomizer are explored while keeping the liquid flow rate constant. Consistent with previous studies, the equivalent droplet size distribution determined at the early stages of the atomization process will be compared against standard diagnostic techniques used for fully formed sprays, such as high-speed imaging and Phase Doppler Anemometry (PDA). Furthermore, the influence of liquid properties on the atomization regime will be investigated by varying the injection temperature. We anticipate that these results will be finalized and incorporated into the upcoming publication and presented at the upcoming conference.

Keywords: Surface Curvature Analysis, Pre-filmer atomizer, Aero-engine

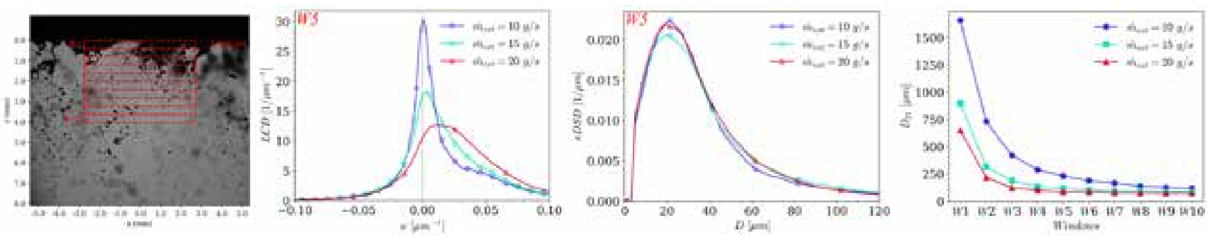


Figure represents (from left to right) the high-resolution image obtained from DBI imaging with windows used in surface curvature analysis, length-curvature distribution, equivalent droplet size distribution, and the variation of surface mean diameter for different co-flow conditions

References
[1] L. Huang, B. Duret, and F.-X. Demoulin, “Curvature analysis of the projections of the 3d liquid structures,” International Journal of Multiphase Flow **197**, 105607 (2026).

[2] C. S. Vegad, D. Ferrando, M. Carreres, S. Idlahcen, B. Duret, J. Reveillon, and F.-X. Demoulin, “Curvature-based analysis of complex morphologies at early stages of atomising spray,” Journal of Fluid Mechanics **1027**, A43 (2026).

Optical Rapid Compression Machine Test Facility for Low and Zero-Carbon Fuels

Steven Begg*, Guillaume de Sercey, Vikas Sharma, and Angad Panesar

Advanced Engineering Centre, School of Architecture, Technology and Engineering, University of Brighton, United Kingdom
*Corresponding author: S.M.Begg@brighton.ac.uk

Abstract
Rapid Compression Machines (RCMs) are widely used to investigate fuel spray dynamics and combustion processes under well-controlled and highly repeatable thermodynamic conditions. In combination with optical diagnostics, they enable detailed investigation of spray formation, atomisation, fuel–air mixing, ignition, flame development and pollutant formation. The Advanced Engineering Centre (AEC) at the University of Brighton has updated a highly configurable optical RCM based on the single cylinder, Ricardo Proteus, medium duty platform (*Fig. 1*). The system comprises an intake heated and boosted, reciprocating, single-cylinder, two-stroke, Schnürle loop-scavenged engine, operating between 100 and 1000 rpm with variable back pressure. The modular design provides adaptable combustion chamber geometry, optical access and in-cylinder flow conditions. Interchangeable chambers and cylinders are available, including quiescent and turbulent configurations, that can incorporate top-hat optical and Brewsters angle (PDA) heads, enabling both planar and volumetric optical diagnostics. A pre-chamber head design, incorporating dual fuel injection, flame ports and adjustable, tipped, internal tuning rods can be utilised to significantly vary turbulence intensity, flow structures and promote jet-like ignition conditions, allowing systematic investigation of spray–flow–combustion interactions. The facility has supported a wide range of experimental campaigns, spanning non-reacting spray studies, ignition and combustion of conventional and low-carbon fuels, and validation of numerical models. Ongoing research includes methanol, ammonia and hydrogen sprays and combustion, increasingly focused on dual-fuel and staged combustion concepts. The facility enables studies relevant to split-cycle and advanced combustor concepts, where the investigation of decoupled compression and combustion processes are of high interest.

Keywords: Optical diagnostics, RCM, spray characterisation, alternative fuels, combustion

Figure 1. *Optical Rapid Compression Machine at Advanced Engineering Centre, University of Brighton*

Numerical Investigation of Non-Premixed Combustion of Hydrogen-Enriched Methane and Ammonia in a Reverse-Flow Gas Turbine Combustor

Rachid Renane*[1], Rachid Allouche[2], Hamama Remmal[2]

[1,2] Laboratory of aeronautical science, Institute of Aeronautics and space studies, University of Blida1, Algeria
*Corresponding author: r.renane-aerospatiale@univ-blida.dz

Abstract

This preliminary study presents a numerical investigation of non-premixed combustion of methane and ammonia fuels, with particular emphasis on hydrogen enrichment, in a reverse-flow gas turbine combustor. The objective is to evaluate fuel substitution effects along the transition pathway $CH_4 \rightarrow CH_4/H_2 \rightarrow NH_3/H_2$, while highlighting the role of reverse-flow geometry in improving flow organization, flame stability, and combustion efficiency under gas-turbine-relevant operating conditions. Numerical simulations are performed using ANSYS Fluent 2023 R2, solving the compressible, multi-species, reacting Reynolds-Averaged Navier–Stokes (RANS) equations governing mass, momentum, energy, and species conservation. Turbulence is modeled using the realizable k–ε model, chosen for its improved predictive capability in swirling, recirculating, and rapidly strained flows typical of gas turbine combustors. Turbulence–chemistry interactions are captured through the Species Transport formulation combined with the Eddy Dissipation (ED) and Eddy Dissipation Concept (EDC) combustion models, providing an efficient compromise between computational cost and chemical accuracy. Chemical kinetics are represented using reduced mechanisms: a four-step global scheme for CH_4/H_2 combustion and a seven-step reduced kinetic scheme for NH_3/H_2 flames, capturing the dominant reaction pathways at reasonable computational cost. Simulations are conducted under steady-state conditions with constant fuel mass flow rate, while radiation and gravity effects are neglected. A pressure-based solver with double precision and parallel execution ensures numerical robustness and rapid convergence. The combustor adopts a three-dimensional reverse-flow cylindrical configuration, designed to enhance residence time, mixing quality, and thermal uniformity. The geometry includes an annular fuel injector, a swirl section equipped with eight vanes at a 45° angle, and a downstream dilution zone with multiple air injection holes. A grid sensitivity analysis is carried out, and the final mesh of approximately 2.52 million elements is selected based on axial velocity convergence. Model validation is achieved by reproducing the methane combustion configuration reported by Rajpara (2018), showing good agreement between predicted and experimental normalized temperature profiles at AFR = 50 and an inlet temperature of 300 K. Initial results indicate that methane combustion produces stable flames with peak temperatures near 1883 K (fig.1), whereas pure ammonia combustion exhibits weak heat release with peak temperatures around 1200 K (fig.2). Preliminary NH_3/H_2 results show that adding 15% hydrogen increases peak temperature to approximately 1800 K (+30%), accompanied by a 40% increase in axial velocity, demonstrating enhanced combustion intensity. Future work will focus on hydrogen-enrichment effects on flame speed, pollutant formation, and overall combustion performance, particularly CO_2 emissions for methane and NOx formation for ammonia, toward sustainable gas turbine applications.

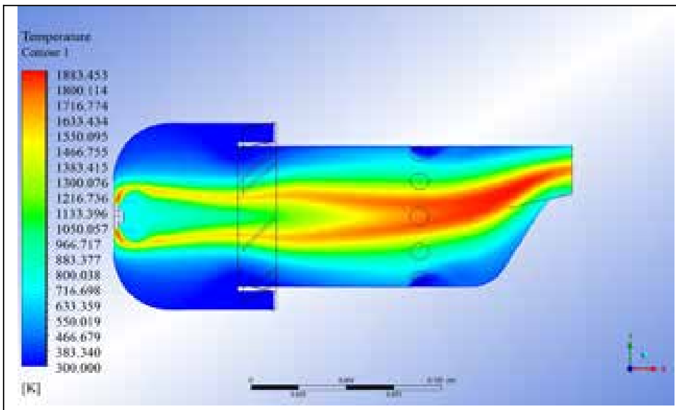


Figure 1. Methane Temperature Contour

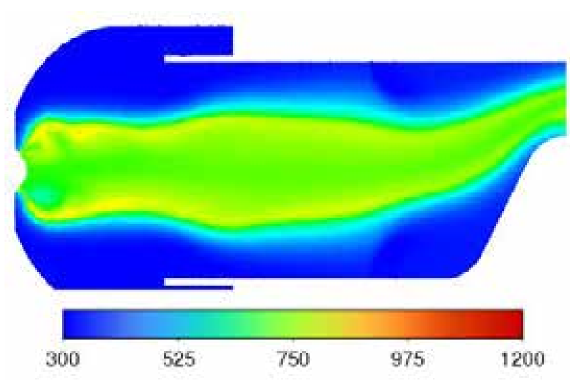


Figure 2. Ammonia Temperature Contour

References

[1] Alfazazi, A., Kumar, S., Behar, O., & Dally, B., 2025. Effects of ammonia substitution on the structure and emissions of non-premixed methane flames stabilized by a bluff-body burner. International Journal of Hydrogen Energy, 170, 151314. https://doi.org/10.1016/j.ijhydene.2025.151314

[2] Rajpara, P., Dekhatawala, A., Shah, R., & Banerjee, J., 2018. Thermal and emission characteristics of reverse air flow CAN combustor. International Journal of Thermal Sciences, 128, 175–183. https://doi.org/10.1016/j.ijthermalsci.2017.12.008

Critical Transition of Ethanol Droplets in Vapor-Rich Environments: A Numerical Study using the REFPROP F8 Helmholtz EOS

R.M. Coelho*[1], P.H.A. Mota[1], J.M. Vedovotto[1], G.S. de Oliveira[1] and R. Serfaty[2]

[1] MFLab, Faculty of Mechanical Engineering, Federal University of Uberlândia, MG 38408-100 BR
[2] Cenpes / Petrobras, Rio de Janeiro, RJ 21941-915 BR
*Corresponding author: coelho_rafa@ufu.br

Abstract

The evaporation and mixing of liquid fuels at supercritical pressures are defining phenomena in modern high-performance combustion systems, including regenerative cooled rocket engines and high-pressure Gasoline Direct Injection (GDI) systems. In these dense spray environments, droplets do not evaporate in isolation but interact with high concentrations of background fuel vapor. While the effects of ambient pressure and temperature are well-documented, the influence of background vapor saturation on the transition from subcritical evaporation to supercritical diffusion remains a critical knowledge gap. Standard cubic equations of state often struggle to resolve the highly non-linear thermodynamic behavior near the critical point, potentially leading to inaccuracies in predicting droplet lifetimes and mixing rates. To address this, this study presents a numerical investigation into the thermodynamic evolution of single ethanol droplets using an Eulerian-Lagrangian solver coupled with the NIST REFPROP database. We employ the high-accuracy F8 Helmholtz energy equation of state to resolve Vapor-Liquid Equilibrium (VLE) and transport properties, offering superior fidelity over traditional models. Following validation against experimental lifetime data for n-heptane at high pressures, we perform a parametric sweep of ethanol droplets in nitrogen at ambient temperatures of 400–1000 K and background vapor mass fractions ($Y_{vap,\infty}$) ranging from 0.0 to 0.75. The study spans pressures from 0.1 to 8.0 MPa ($0.01 < P_r < 1.3$), crossing the critical pressure of ethanol. By analyzing the competition between evaporative cooling suppression and the vanishing latent heat of vaporization, this investigation tests the hypothesis that high local vapor saturation fundamentally alters the thermodynamic trajectory of the droplet. The results quantify the extent to which ambient saturation shifts the pressure thresholds required for the critical transition, providing essential data for optimizing fuel-oxidizer mixing in future propulsion systems.

Keywords: Droplet evaporation, Supercritical pressure, Vapor saturation, Ethanol, F8 Helmholtz EOS.

Interferometric Laser Imaging for Droplet Sizing of Flash-Boiling Ammonia Sprays

Utkarsha Sonawane, Vijayakrishnan Venkatesan, Camille Hespel, Christine Rousselle*
PRISME Laboratory, Université d'Orléans, Orléans, France
*IUF, Institut De France, Paris, France

Abstract

Ammonia as a carbon-free energy fuel has motivated extensive research into its application in internal combustion engines. However, the physical properties of ammonia differ significantly from those of conventional liquid fuels. In particular, its low boiling point (−33.4 °C @ 1 bar) makes liquid-phase direct injection highly susceptible to flash boiling. A detailed characterization of the resulting droplet distribution is therefore essential for understanding spray development and combustion processes. The present study investigates non-flash boiling/ flash-boiling ammonia sprays using **Interferometric Laser Imaging for Droplet Sizing (ILIDS)**. ILIDS is based on the interference between light reflected and refracted by individual droplets, providing spatially resolved information on droplet size.

The experiments were conducted across a range of fuel and ambient conditions that correspond to different degrees of flash boiling conditions of a spray from a single-hole injector. The degree of superheat was characterized by the pressure ratio $R_p = P_{sat}(T_{fuel})/P_{amb}$), covering the non-flash-boiling, transitional, and strong-flash-boiling regimes. Selecting the field-of-view location is critical due to the strong spatial and temporal variations in spray morphology and droplet densities. Therefore, only measurements performed at the spray periphery, approximately 8 mm from the spray axis, provided suitable conditions to apply ILIDS than the dense central region.

The influence of the optical parameters was subsequently investigated. Increasing the numerical aperture by reducing the F-stop increased the diameter of the defocused droplet images and produced larger fringe spacing, thereby improving fringe visibility. A similar tendency was observed as the defocusing distance increased. However, excessive defocusing increased the overlap between neighboring droplet images, making individual interference patterns difficult to resolve.

The results highlight the main challenges associated with applying ILIDS to transient flash-boiling ammonia sprays. In its present configuration, reliable measurements are primarily achievable in sufficiently dilute spray regions. Ongoing optimization of the numerical aperture, magnification, defocusing distance, and field-of-view position aims to minimize droplet-image overlap while maintaining adequate fringe spacing for both fine and coarse droplets. The optimized optical configuration will provide a basis for spatially resolved droplet-size characterization of flash-boiling ammonia sprays and contribute to a better understanding of their atomization behavior.

Keywords: Ammonia spray; Flash boiling; Droplet sizing; ILIDS; Laser diagnostics; Spray characterization; Atomization

Ackowledgement : Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or the European Research Executive Agency. Neither the European Union nor the granting authority can be held responsible for them.

Analytical modelling of finite amplitude oscillation of viscous drops in single mode

Gianpietro Elvio Cossali and Simona Tonini*

Department of Engineering and Applied Sciences, University of Bergamo, Dalmine, Italy
*Corresponding author: simona.tonini@unibg.it

Abstract

Freely oscillating droplets have been studied for over a century because of their fundamental role in interfacial transport phenomena and their significance in numerous civil and industrial applications [1], [2]. The droplet shape is recognized as a key factor affecting the associated transport mechanisms, and several modeling approaches have been proposed in the literature to describe the oscillation amplitude and frequency under specific physical and boundary conditions [3].

The poster describes the mathematical formulation and the derivation of the analytical solutions needed to model finite amplitude oscillation of a viscous drop in single mode. The drop is assumed to oscillate in single mode, i.e. maintaining the shape defined by the function $R(\eta) = R_0(\gamma + a_n(t)P_n(\eta))$ where R is the distance of any point on the drop surface from the centre of mass, R_0 is the equivalent drop radius, $P_n(\eta)$ is the Legendre polynomial of order n (the oscillation mode), $\eta = \cos(\theta)$ is the cosine of the polar angle and a_n is a time dependent finite amplitude. A proper non-orthogonal curvilinear coordinate system is defined in such a way that one iso-surface coincides with the drop surface and deforms with time while $a_n(t)$ is varying, maintaining constant the enclosed volume. The approach represents a generalisation of that used in [4] to model the oscillation of spheroidal droplets. The law of motion for $a_n(t)$ is obtained imposing that the time derivative of the drop kinetic and surface energy is equal to the viscous dissipation. The liquid flow is assumed incompressible, and irrotational and this allows an analytical evaluation, in terms of a truncated series, of the liquid velocity field, and consequently of the drop kinetic energy and of the dissipation term, while the drop surface energy is calculated from the area of the deformed drop surface.

The results of Rayleigh [5] and Lamb [6] for the case of infinitesimal oscillation amplitude are analytically obtained from the model equations, and also the limit for aperiodic decay [7] is correctly found. A comparison with different data sets from the scientific literature and the application of the model to give a physical interpretation of some peculiar characteristics of the drop oscillation dynamics are proposed. The asymmetry in the time spent in the two shape drop configuration for even modes, which is known to increase with the oscillation amplitude for mode $n = 2$, is reproduced, and the effect is found to hold also for higher modes, with different strength. The effect of viscous dissipation of the oscillation frequency, experimentally observed for the mode $n = 2$, is analysed for finite amplitude oscillations of higher modes. The poster will report some preliminary results of the proposed modelling as well as a comparison with available experimental and numerical data.

References

[1] H.J.J. Staat, A. van der Bos, M. van den Berg, et al., "Ultrafast imaging method to measure surface tension and viscosity of inkjet-printed droplets in flight," Exp. Fluids **58**, 2 (2017).

[2] G. Brenn, "Drop shape oscillations," in Droplet Interactions and Spray Processes (Springer International Publishing, 2020), 239-249.

[3] A. Aalilija, C.-A. Gandin, and E. Hachem, "On the analytical and numerical simulation of an oscillating drop in zero-gravity," Comput. Fluids **197**, 104362 (2020).

[4] S. Tonini, and G.E. Cossali, "Analytical model of small- and large-amplitude drop oscillation dynamics," Physics of Fluids **36**, 117147 (2024).

[5] L. Rayleigh, On the capillary phenomena of jets, Proc. R. Soc. Lond. A **29**, 71-97 (1879).

[6] H. Lamb, On the oscillations of a viscous spheroid, Proc. Lond. Math. Soc. **13**, 51-66 (1881).

[7] A. Prosperetti, "Free oscillations of drops and bubbles: the initial-value problem," J. Fluid Mech. **100** part 2, 333-347 (1980).

Optical investigation of cavitation and flash boiling in transcritical cryogenic sprays

Fergus Murphy-Hunt*[1], Campbell Marlow Eastwood[1], Martin White[1], Cyril Crua[1]

[1]Energy and Materials Engineering Research Centre, University of Sussex, UK
*Corresponding author: f.murphy-hunt@sussex.ac.uk

1 Abstract

Prediction of the onset and severity of flash boiling in cryogenic spays is important as it significantly impacts spray morphology, mixing behaviour and heat transfer. Being able to predict and understand the impact of nozzle length to diameter ratio (L/D) on flash boiling behaviour under transcritical operating conditions could lead to improved understanding and design of nozzles within hydrogen refuelling, pharmaceutical production and power generation systems. The influences of nozzle geometry and cavitation on flash boiling behaviour in cryogenic sprays are sparsely researched areas with limited experimental data available. Superheat index and cavitation number are commonly used metrics to predict the onset and severity of flash boiling or cavitation behaviour in nozzles under subcritical conditions, however these metrics do not account for nozzle geometry despite this being known to impact the thresholds at which the phenomena occur. In-nozzle and near-nozzle observations will be conducted using long-distance shadowgraph imaging and transparent nozzles to provide novel experimental data, and improve the understanding of how nozzle geometry and cavitation influence flash boiling development in liquid nitrogen sprays. The impact of L/D ratio (0.5 to 5) on spray morphology and in-nozzle phase change behaviour will be analysed under subcritical and transcritical conditions. The accuracy and applicability of commonly used metrics to predict the onset and scale of flash boiling and cavitation behaviour will be assessed against the new experimental dataset.

Keywords: Cryogenics, Flows, Transcritical mixing, Flash boiling, Cavitation

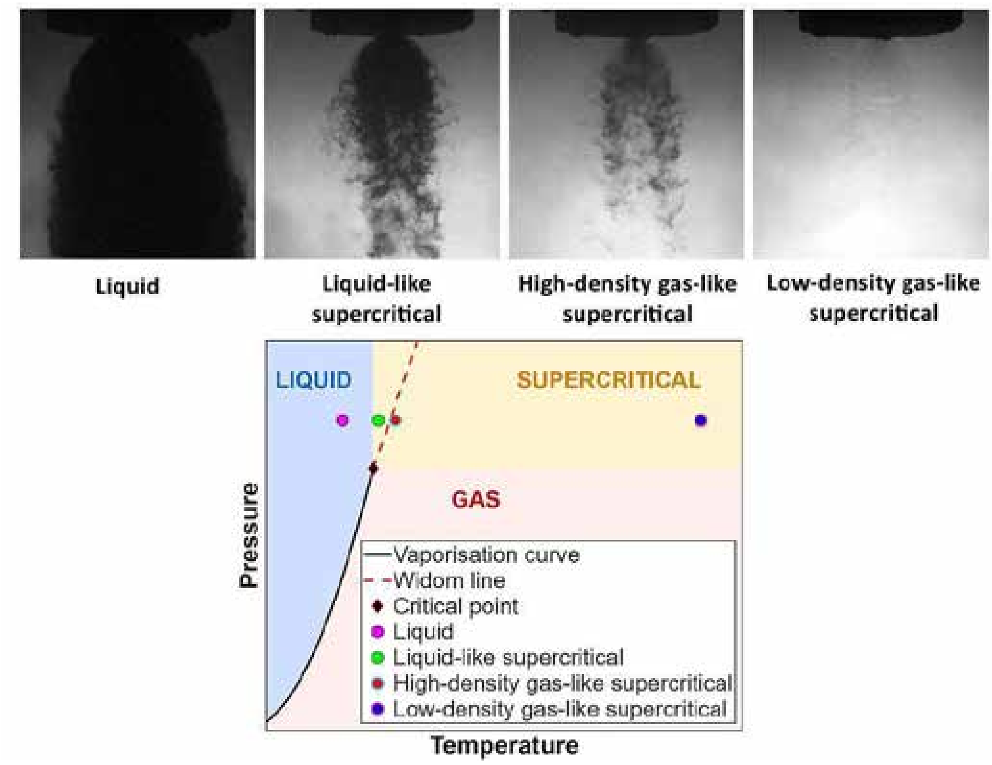


Figure 1. *Flash boiling liquid and supercritical nitrogen sprays into atmospheric conditions with approximate locations on a phase diagram*

Dynamic Analysis of a Solenoid-Driven Diesel Injector Using a One-Dimensional Model for DME Fuel

Jinwoo Lee*[1]

[1]School of Mechanical Engineering, Ulsan College, South Korea
*Corresponding author: jwlee2@uc.ac.kr

Abstract

Low-viscosity alternative fuels such as dimethyl ether (DME) have attracted increasing attention due to their potential for cleaner combustion and reduced emissions in compression ignition engines. However, the physical properties of DME-including low viscosity, high compressibility, and limited lubricity-differ significantly from those of conventional diesel fuel, which can strongly affect the dynamic behavior of fuel injection systems. In particular, these properties influence needle motion, internal pressure evolution, and injection stability in solenoid-actuated diesel injectors.

This study presents ongoing work on the development of a one-dimensional, system-level model for a conventional solenoid-driven diesel injector under low-viscosity fuel conditions. In the investigated injector configuration, the electromagnetic force generated by the solenoid directly actuates the needle against the opposing spring force, and the transient injector behavior is governed by the dynamic balance among electromagnetic force, hydraulic pressure forces, inertial effects, and flow-induced damping. The modeling framework captures the coupled interactions between solenoid current excitation, electromagnetic force generation, needle motion, internal fuel pressure dynamics, and flow through injector orifices. In particular, the model formulation emphasizes accurate representation of fuel compressibility effects and flow-induced damping, which become increasingly important under low-viscosity fuel conditions. This approach enables a more realistic description of transient injector behavior compared to simplified quasi-steady modeling assumptions.

The developed model is applied to analyze transient needle dynamics and pressure evolution during injection events under both diesel and DME fuel conditions. Particular attention is given to the influence of fuel property variations on needle opening delay, closing behavior, and internal pressure fluctuation characteristics. In addition, the analysis highlights the interplay between electromagnetic force buildup and hydraulic pressure response during the initial opening phase, providing insight into the mechanisms governing injection delay and transient instability under DME fuel operation. Preliminary simulation results indicate that low-viscosity fuel conditions can significantly alter the transient force balance acting on the needle, leading to noticeable changes in injector dynamic response compared to conventional diesel operation. Sensitivity trends further suggest that internal hydraulic flow resistance and fuel compressibility play a critical role in determining injector stability under DME fuel conditions.

Future work will focus on further refinement and calibration of the one-dimensional model using experimentally measured data, as well as its extension to investigate injector dynamic behavior over a broader range of operating conditions. The present study aims to provide fundamental insights into the transient behavior of solenoid-actuated diesel injectors and to support system-level modeling approaches for adapting conventional injection systems to low-viscosity fuels. These findings suggest that injector dynamic behavior under low-viscosity fuel conditions cannot be reliably inferred from diesel-based assumptions alone, underscoring the need for fuel-adaptive modeling approaches.

Keywords: Solenoid injector; Diesel injector; DME fuel; Low-viscosity fuel; One-dimensional modeling

Flow Dynamics and Stability Limits of Electrodynamic Sprays through Permeable Electrodes

K. Mehrabi*, F. J. Higuera

Department of Fluid Mechanics, ETSIAE, Polytechnic University of Madrid, Madrid, Spain
*Corresponding author: karim.mehrabi@upm.es

Abstract

This study investigates the flow dynamics and stability limits of dilute sprays of electrically conducting particles generated between two horizontal permeable electrodes. Unlike aerodynamic atomization, the spray is driven by DC electric forces that overcome particle weight and adhesion, initiating a continuous cycle where particles detach from the lower electrode and evacuate through the upper grid. We focus on the collision-dominated regime, occurring when the particle mean free path is small compared to the interelectrode distance.

Using an Eulerian model, we compute stationary distributions of particle number and charge densities as functions of applied voltage, gas velocity, and electrode permeability. This model incorporates an upward vertical gas stream to assist particle transport. A linear stability analysis defines the operational boundaries, identifying a stable regime between the onset voltage and a critical upper threshold. These results provide fundamental criteria for optimizing particle flow and current density in applications such as aerosol combustion and powder metallurgy.

Keywords: Electrodynamic spray, flow dynamics, particle, permeable electrodes, linear stability.

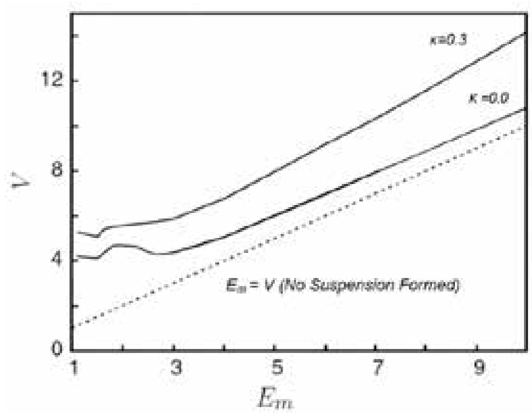


Figure 1. *Stability limits. V: Dimensionless voltage; E_m: Dimensionless Electric field at the lower electrode; k: Electrode permeability (k=0 non-permeable, k=0.3 high permeability).*

Acknowledgements

This work was supported by MCIN/AEI/10.13039 and ERDF under grant PID2023-150329NB-C22.

References

[1] F.J. Higuera, "Analysis of electrodynamic fluidization," J. Fluid Mech. 854, 261-292 (2018).

[2] H. Ling and F. J. Higuera, "A collision-dominated regime in electrodynamic particulate suspension," J. of Aerosol Science 174, 106253 (2022).

[3] Maxwell, J. C. 1881. A Treatise on Electricity and Magnetism. Oxford University Press.

On the impact of nanoparticle shape on droplet retraction over smooth surfaces

Maria Rosaria Vetrano*[1], Marian van Alphen[1], Qixuan Dong[1], Y. Shang[1], D. Fontanarosa[1]

[1]KU Leuven, Department of Mechanical Engineering, Division of Applied Mechanics
and Energy Conversion (TME), B-3001 Leuven, Belgium
*Corresponding author: rosaria.vetrano@kuleuven.be

Abstract

Droplet impact on solid surfaces plays a crucial role in spray cooling, coating, and additive manufacturing. While nanofluids are known to enhance thermal performance, their dynamic impact behavior, particularly during the retraction phase, remains insufficiently understood. Most existing studies focus on spreading to splashing thresholds[1][2], leaving retraction largely unexplored. This work presents an experimental investigation of the retraction phase of nanofluid droplets impacting a smooth sapphire substrate. High-speed visualisation was employed to analyse aqueous glycerol (AG30, AG70) mixtures containing 0.1 wt% carbon nanotubes (CNT, cylindrical shape) and 0.1 wt% Al_2O_3 nanoparticles (spherical shape). The study covers a range of Weber and Reynolds numbers by varying the droplet release height. In total, 25 test cases have been analyzed to support the analysis of spreading/retraction dynamics. Results confirm that nanoparticle addition does not significantly alter the maximum spreading factor. However, pronounced differences emerge during retraction. Three key trends are identified: (i) increasing impact velocity increases retraction speed; (ii) increasing viscosity reduces retraction speed; and (iii) at low base-fluid viscosity (AG30), nanoparticle interactions dominate, leading to a reduced retraction speed that becomes nearly independent of impact velocity. Notably, CNT nanofluids exhibit approximately half the retraction velocity of Al_2O_3 nanofluids within the first 60 ms after maximum spreading. To the authors' knowledge, this is the first systematic experimental study dedicated to the retraction phase of nanofluid droplets. The findings highlight the critical role of nanoparticle shape in post-impact dynamics, with direct implications for surface-engineering applications. Some exemplary curves representing the evolution of the droplet spreading factor (including retraction) are presented in **Figure 1**.

Keywords: Nanofluids, droplet retraction, nanoparticle shape.

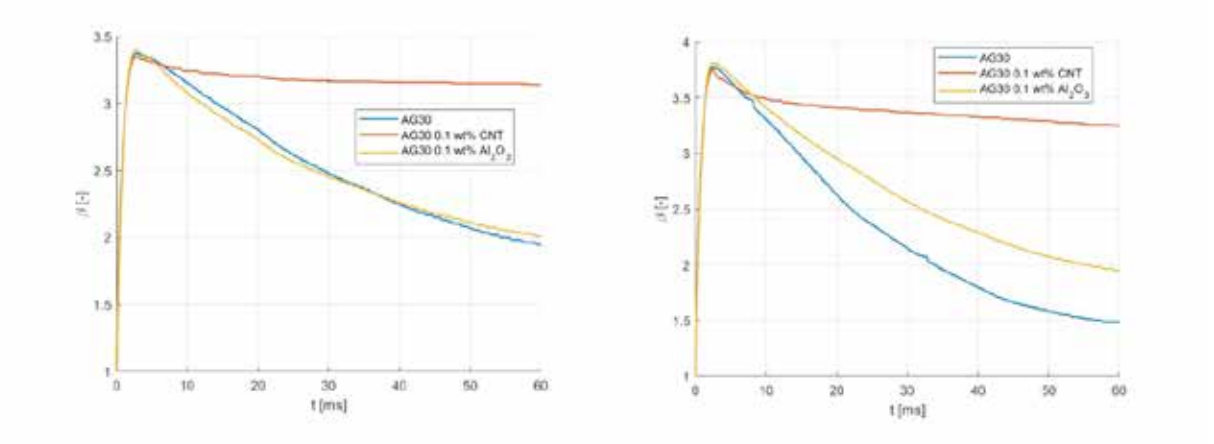


Figure 1. *Spreading and retraction for pure fluid AG30, AG30 0.1 wt% CNT and AG30 0.1 wt% Al_2O_3 at We~180 (left) and We~300*

References

[1] Y. T. Aksoy et al., Role of Nanoparticles in Nanofluid Droplet Impact on Solid Surfaces, *Langmuir, 39* (1), 12-19 (2023).

[2] Y.T. Aksoy, et al., Spreading - splashing transition of nanofluid droplets on a smooth flat surface, Journal of Colloid and Interface Science, Volume 606, Part 1, (2022).

Three-Layer Gas-Phase Modelling for Numerical Simulation of Sodium Spray Combustion in a Sodium-Cooled Fast Reactor Compartment

Junhyeok Kim[1], Byung Jo Kim*[1]

[1]Nuclear Safety Technology Research Dept., KEPCO E&C, Republic of Korea
*Corresponding author: bjokim@kepco-enc.com

Abstract

Sodium spray combustion involves strong coupling among droplet-scale burning, oxygen consumption, heat release, and compartment-scale gas-phase environment. In sodium-cooled fast reactors (SFRs), leaked sodium may disperse into fine droplets and react rapidly with oxygen, causing rapid increases in compartment gas temperature and pressure. This study presents a numerical treatment of sodium spray fire phenomena toward an integrated sodium fire analysis code.

In realistic design-basis accident sodium leakage scenarios, spray combustion is limited above the leak elevation and occurs mainly along the droplet-falling path below the leak. A fraction of unburned sodium reaches the lower region and accumulates on the floor to form a pool-fire region. Thus, spray and pool combustions can coexist while sharing the same gas phase and competitively consuming oxygen, resulting in interdependent behavior. However, existing sodium fire analysis codes employ a lumped-parameter approach with a single well-mixed gas volume, which cannot explicitly represent vertical oxygen distribution or thermal stratification.

To address this limitation, a three-layer gas-phase model was developed within an existing sodium fire analysis framework and compared with a conventional single-volume model. The model accounts for spray combustion during droplet fall, transport of unburned sodium to the lower region, and subsequent pool combustion. It was evaluated against the SNL Surtsey T3 and JAEA Run-E1 experiments, followed by a preliminary application to PGSFR, a previously developed SFR design in Korea. Results indicate that multilayer gas-phase treatment changes local oxygen availability and heat-release distribution, leading to non-negligible differences in burning rate, unburned sodium accumulation, and temperature-pressure response.

Keywords: Sodium-cooled fast reactor, spray/pool fire coupling, Sodium spray combustion

Light field imaging for gas-liquid interfaces in multiphase flows: Bubble/droplet sizes

Ramana Kukkarasi*[1], Adrian Marszalek[1], Jean-Bernard Blaisot[1], Marc Brunel[1]

[1] CORIA UMR 6614-University of Rouen, Saint Etienne du Rouvray, France
*Corresponding author: kukkarar@coria.fr, kukkarasir@alum.iisc.ac.in

Abstract

There has been a significant increase in interest in light field imaging for the volumetric measurements of a wide range of fluid mechanics problems (e.g. 3D PIV, bubble flows & sprays) owing to the large depth of field. In our previous study [1], detailed measurements of bubble location and size distributions for a single bubble regime were presented using a commercially supplied light field imaging package from Raytrix. In this study, we present the successful in-house development of a light field imaging optical system, specifically intended for studies on the gas-liquid interface of multiphase flows. The optical characteristics of the micro lens array (MLA, which is a critical component of a light field imaging system) are varied, and the resultant light field imaging was tested using a high-definition camera. The image processing methodology and the measurements on a bubble flow by the in-house light field imaging system are presented. The poster presentation focuses mainly on the challenges in implementation and critical parameters of MLA, governing the resultant light field. Further, analytical estimation of the resolution and depth of field of the light field image system is discussed, and progress and future directions of the present work are summarised.

Keywords: Multiphase flows, volumetric measurements, light field, micro lens array.

References

[1] P. Schleuniger, R.-A. Herrera Leclerc, M. Brunel, M.-C. Renoult, and J.-B. Blaisot, “Experimental three-dimensional location and size distribution of rising bubbles in a cylindrical column through light field imaging,” Physics of Fluids **36** (2024).

On the Relevance of Nozzle-Internal Surface Roughness on Flash-Boiling Phenomena of Liquid Ammonia

Peter Augustin*[1], Henry Axt[2], Martin Kratz[2], Christian Hinke[2], Wilko Rohlfs[1], Manuel Reddemann[1]

[1]Institute of Heat and Mass Transfer, RWTH Aachen University, Germany
[2]Chair for Laser Technology, RWTH Aachen University, Germany
*Corresponding author: augustin@wsa.rwth-aachen.de

Abstract
Wall roughness inside injector nozzles governs heterogeneous nucleation that triggers flash-boiling during the atomization of superheated liquids such as ammonia. Technically utilized nozzles exhibit roughness and micro-defects that can trap pre-existing bubbles, lowering the energy barrier for bubble growth and promoting nucleation. Consistent with this, prior comparisons between steel and lapped steel nozzles showed that reduced roughness suppresses internal nucleation: smooth nozzles often exhibit only external flash-boiling, whereas rough surfaces favour a transition to internal flash-boiling. We expect that this effect is even amplified for liquid ammonia, whose low viscosity and surface tension increase sensitivity to near-wall flow fluctuations. However, existing transition criteria often fail for ammonia because they neglect explicit surface effects – roughness and wettability – leading to theory-experiment discrepancies. This study introduces a transparent, geometry-controlled nozzle platform to isolate and quantify surface-roughness effects on ammonia flash-boiling. Using the subtractive manufacturing technique Selective Laser-Induced Etching (SLE), we realize well-defined glass nozzles with exit diameters (100 µm) and L/D ratio 2.5, and roughness S_a (1 µm and 10 µm). The superheating degree and injection pressure is additionally varied to map thermodynamical transition criteria. High-speed shadowgraphy visualizes the spray morphology. This framework enables mechanistic mapping of roughness-driven transitions and provides data to refine predictive criteria that explicitly account for surface roughness in ammonia injection.

Keywords: Flash-Boiling, Liquid Ammonia, Selective Laser-Induced Etching, Surface Roughness

Acknowledgements
This work has been funded by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) Project number: 523959480 (SPP2419)

Dispersion Gas Effects on Flash-Boiling Atomization and Stability of Liquid Ammonia Spray Flames

Arirat Arpert*[1], Peter Augustin[1], Lukas Heine[1], Anna Maßmeyer[1], Wilko Rohlfs[1], Manuel Reddemann[1]

[1] Institute of Heat and Mass Transfer, RWTH Aachen University, Germany

*Corresponding author: arirat.arpert@rwth-aachen.de

Abstract

Ammonia enables carbon-free combustion as a renewable fuel. Liquid ammonia (LNH_3) spray combustion offers lower system costs and shorter start-up times. Injection of LNH_3 into a low-pressure atmosphere yields flash-boiling, a superior atomization mechanism for suitable mixture formation to overcome the combustion-adverse properties of ammonia. With a modular LNH_3 spray and combustion test rig featuring a coaxial atomizer with a centric LNH_3 feed and co-axial annular oxygen feed, we investigate flames in a range between 70% and 90% ammonia by lower heating value (E_{NH3}). The remaining energy is supplied by a premixed methane/oxygen pilot flame. The system is characterized by chemiluminescence (NH*, OH*), high-speed flame imaging and high-speed shadowgraphy of the spray in inert and active flame conditions.

The dispersion gas influences both atomization and flame stability. At a constant global equivalence ratio of Φ = 0.68 and E_{NH3} = 73%, reducing the dispersion gas flow rate from 1 to 0 slm causes the flame to extinguish. Introducing 1 slm of dispersion gas reestablishes stable spray combustion, even at a higher ammonia heat fraction of 75%. This indicates that atomization quality, rather than global stoichiometry alone, determines the boundary of lean stability.

Chemiluminescence shows that redistributing oxygen from premix to dispersion gas results in a reduction of NH* /OH* emission intensity and spatial shift in peak emission zones. This suggests that the spatial distribution of oxygen between pilot flame and dispersion gas affects the reaction intensity and location and that the co-annular dispersion gas serves as a critical design parameter to achieve stable LNH_3 flames.

Keywords: Liquid Ammonia, Flash-Boiling, Spray Combustion

Acknowledgements

This work has been funded by the Deutsche Forschungsgemeinschaft (DFG, German Research Foundation) Project number: 523959480 (SPP2419)

Visualization, Measurement, and Analysis of Spray and Flame Propagation in an Ammonia Direct-Injection Engine

Yongrae Kim[1], Cheolwoong Park[1], Chansoo Park[1], Young Choi[1]

[1]Korea Institute of Machinery and Materials (KIMM), Daejeon, Republic of Korea

Abstract

This study aims to establish a platform for simultaneous fuel spray and flame visualization experiments in a single-cylinder engine with direct ammonia injection. By conducting experiments under various conditions of excess air ratio and ammonia mixing ratio, the study seeks to predict the effects on engine power output and combustion stability. To facilitate flame visualization, the single-cylinder engine head was modified. Following the assembly of the head and valve train onto the engine block, the design for the baseline experiments was finalized.

The engine used in this study is relatively large compared to those utilized in previous research involving combustion and flow visualization via optical measurement techniques. Consequently, high-speed camera visualization using an inserted endoscope was deemed a practical strategy, as it allows for flame imaging at the CAD time step without altering the combustion chamber geometry or heat transfer characteristics. To ensure the proper application of the aforementioned visualization technique, it was necessary to address field-of-view issues arising from the discrepancy between the endoscope window size and the overall cylinder head dimensions (specifically, the bore diameter). Consequently, a strategy was implemented to mitigate the limitations of the measurement method such as by calibrating the field of view based on measurements taken in an external environment. The air-cooled endoscope assembly consisted of a mounting sleeve machined into the engine head (custom-fitted to the angle of the head's upper surface) and a quartz glass tube (with either a flat or hemispherical tip) serving as the combustion chamber window. Cooling air was supplied through one of two designated ports, and the endoscope provided an 80° field of view.

We evaluated flame propagation speed, color characteristics, specific crank angle timings, and the probability of flame presence in specific regions. Post-processed flame images allowed us to observe flame formation and propagation characteristics, including the initial spray phase. We could see the leading edge of the burning spray entering into the high-speed camera's field of view alongside the diesel flame as ammonia was injected from the initial ambient temperature and pressure conditions. Within the visualization zone, two spray leading edges were observed impinging on the cylinder wall near the quartz window, spreading vertically, and mixing with each other. Additionally, due to the in-cylinder swirl flow, the flame was observed moving across the field of view before dissipating.

CFD-Guided Geometric Design for Swirling Airflow Control in Ultrasonic Spray Systems

Xuezhou Hou, Paul Vesely & Rudi Schick1

1: Spray Analysis and Research Services, Spraying Systems Co.

Abstract:

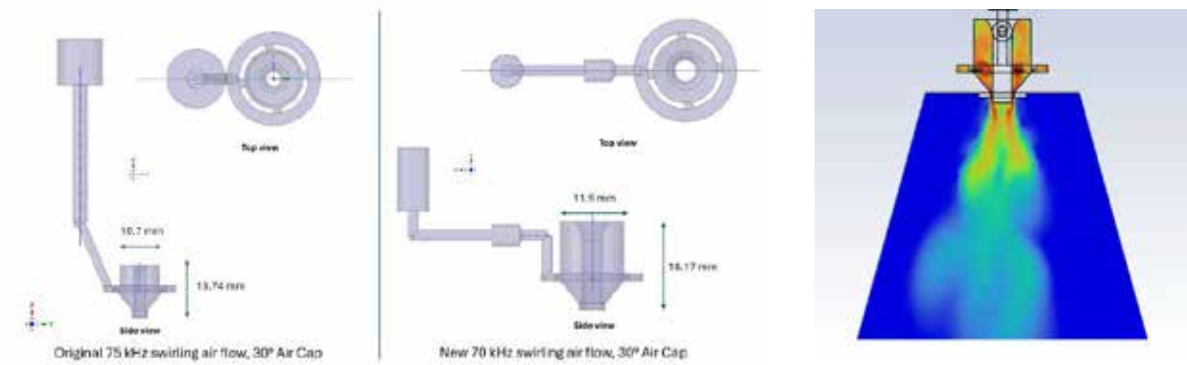
Swirling airflow is a patented design used in SSCo. Ultrasonic Spray Systems to control spray shape and droplet transport, but achieving a stable and symmetric swirl remains challenging. Small geometric differences can significantly affect airflow behavior, leading to inconsistent spray performance. When a successful airflow design was transferred between ultrasonic nozzle platforms, asymmetric spray patterns were observed.

To investigate the underlying cause, two CFD models were developed. A steady-state CFD model was used to evaluate the influence of a single design parameter on swirl generation and to identify an optimized geometry that produces a symmetric swirling flow at the nozzle outlet. A transient CFD model was then used to investigate the evolution of the airflow field over spray distance and assess the persistence of flow symmetry downstream while excluding manufacturing variability. Both analyses quantified the relationship between the key geometric parameter and swirl stability, enabling geometry optimization prior to prototype fabrication.

Experimental spray characterization was subsequently conducted to validate CFD predictions and assess the impact of airflow control on spray symmetry. Ongoing work aims to establish quantitative relationships between nozzle geometry, swirl stability, and spray performance. The combined numerical and experimental approach is expected to provide a systematic methodology for identifying and optimizing the critical design parameters governing swirl-induced spray behavior in ultrasonic atomization systems, thereby improving spray consistency and supporting the development of more robust nozzle designs.

Keywords: Ultrasonic atomization, swirling airflow stability, CFD, flow symmetry, spray characterization, design optimization, etc.

Corresponding author: Xuezhou.Hou@spray.com



(a) *(b)*

Figure 1: (a) CAD models of the two swirling airflow designs; (b) transient CFD model of the optimized swirling airflow for the 70 kHz nozzle.

Sustainable Ultrasonic Atomization with Advanced Spray Analysis for Resource-Efficient Wet Processing

Xuezhou Hou, Paul Vesely & Rudi Schick1

1: Spray Analysis and Research Services, Spraying Systems Co.

Abstract:

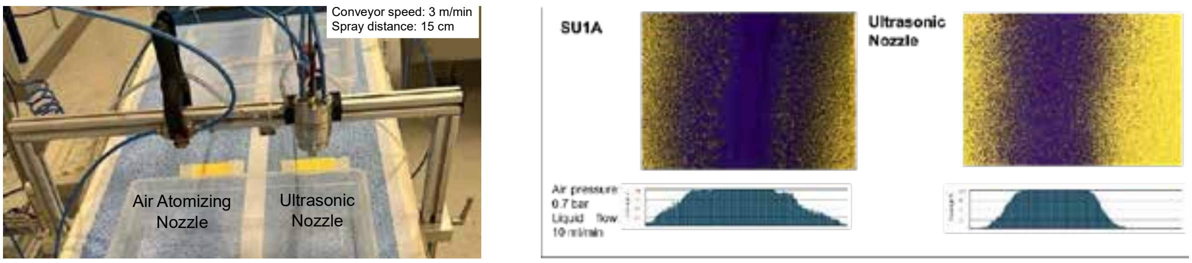
Reducing material consumption while improving process robustness remains a key challenge in semiconductor wet processing applications. This study investigates a 75 kHz ultrasonic atomizer designed for precise low-flow coating processes and compares its atomization and deposition performance with conventional air-assisted atomization.

Droplet size distributions produced by both technologies were characterized using Sympatec laser diffraction measurements. The resulting spray characteristics were correlated with coating coverage obtained from water-sensitive papers and quantified through Surface Deposition Analysis (SDA), providing a direct relationship between droplet size distribution, spray transport, and deposition behavior. In addition, real-time SprayScan® ePT measurements were employed to evaluate spray stability and pattern uniformity, enabling quantitative assessment of process repeatability and robustness.

The combined diagnostic approach is being used to improve the understanding of spray formation and coating performance. Ongoing experimental work aims to establish correlations between droplet size distribution, spray stability, and coating uniformity, with the objective of reducing process variability, material waste, and coating defects while improving deposition uniformity and transfer efficiency. The results are expected to provide new insights into the mechanisms governing ultrasonic atomization and coating performance, supporting the development of more resource-efficient and robust spray processes.

Keywords: Ultrasonic atomization, droplet size distribution, spray diagnostics, transfer efficiency, process monitoring.

Corresponding author: Xuezhou.Hou@spray.com



(a) *(b)*

Figure 1: (a) Experimental spray coating setup using the SUA1 air-atomizing nozzle and ultrasonic nozzle; (b) resulting spray patterns on water-sensitive paper and corresponding coverage distribution analysis.

Flash-Boiling Characteristics of Methanol Sprays: From Bubble Dynamics to Plume Collapse

Vikas Sharma, Guillaume de Sercey, Angad Panesar and Steven Begg*
Advanced Engineering Centre, School of Architecture, Technology and Engineering, University of Brighton, United Kingdom
*Corresponding author: S.M.Begg@brighton.ac.uk

Abstract

Methanol is a promising low-carbon marine fuel due to its renewable production pathways, ease of storage and handling, and near-soot-free combustion characteristics. However, its high latent heat of vaporisation and poor auto-ignition characteristics present challenges for efficient mixture preparation and combustion stability. Despite increasing interest in methanol-fuelled marine engines, the flash-boiling behaviour of methanol sprays under low-pressure conditions relevant to early-injection strategies remains insufficiently understood. This study experimentally investigates the flash-boiling characteristics of methanol sprays using an eight-hole ECN Spray M injector in a low-pressure constant-volume chamber. High-speed shadowgraph imaging and long-distance microscopy were employed to examine the effects of fuel temperature (25–90 °C), injection pressure (50–200 bar), and ambient pressure (0.3–1.0 bar) on spray morphology, vaporisation, plume interaction, and near-nozzle bubble dynamics. Optical intensity analysis and dimensionless scaling using Reynolds, Weber, Jakob, aspect-ratio, and pressure-ratio parameters were performed to characterise flashing behaviour.
The results revealed a transition from non-flashing to fully flashing regimes as the pressure ratio increased from 0.56 to 8.23 (Figure 1). Increasing fuel temperature enhanced spray cone angle, vaporisation, and vapour-phase homogeneity while reducing liquid-core structures. Strong flashing conditions promoted plume interaction, plume collapse, central vapour-core formation, and downstream axial jet development. Near-nozzle visualisation identified injector tip wetting, vapour deposition, bubble growth, and micro-explosion phenomena associated with rapid phase transition. The dimensionless regime maps demonstrated that methanol flash-boiling behaviour is governed by the coupled effects of hydrodynamic atomisation and thermodynamic superheat. These findings provide new insights into the fundamental physics of methanol flash-boiling sprays, demonstrating how thermodynamic superheat modifies spray atomisation, vapour-phase development, plume interaction, and spray collapse under low-pressure conditions. The results establish a framework for understanding methanol flash-boiling behaviour and provide valuable experimental data for the development and validation of spray and combustion models for future methanol-fuelled engine applications.

Keywords: Up to 90 characters max including spaces, to make sure it remains on a single line.

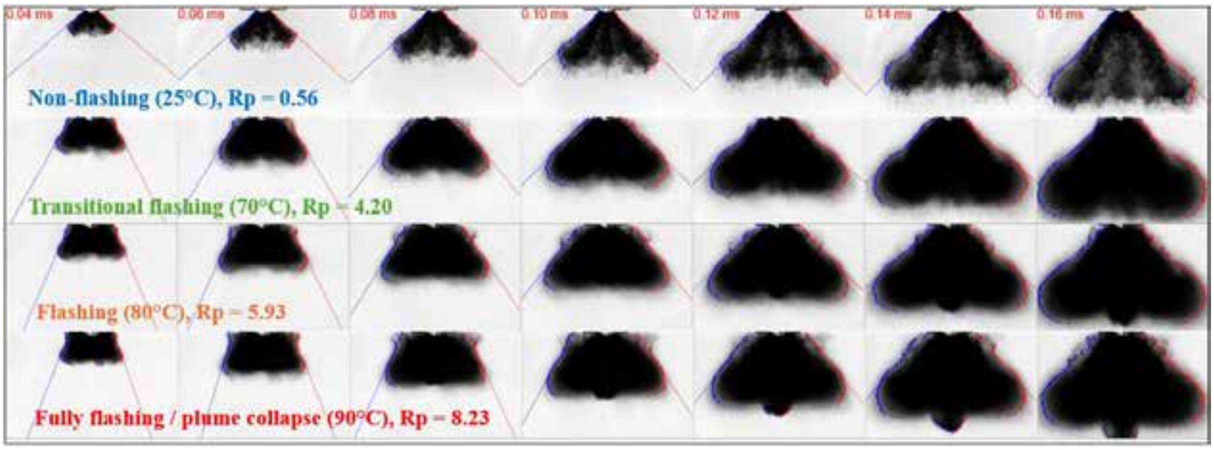


Figure 1. Temporal evolution of methanol spray morphology at $P_{inj} = 100$bar and $P_{amb} = 0.3$bar for fuel temperatures of 25–90 °C. The transition from non-flashing to strong flash boiling is accompanied by increased spray expansion and plume interaction, while further increases in $R_p$produce comparatively smaller changes in the macroscopic spray morphology.

Validation of Spray Characteristics of Hydraulic Axial Full Cone Nozzles Using the Smoothed Particle Hydrodynamics (SPH) Method

Author(s): Dr.-Ing. Jens Stefan Markus[1], stefan.markus@spray.com,
Pierre Hellard[2], hellard@divecae.com

Affiliation: [1] Design & Development, Spraying Systems Co., Germany
[2] Product Engineering, DiveCAE, Germany

Abstract

Inside a hydraulic axial full cone spray nozzle, a swirling free-surface flow is generated by a vane before exiting the nozzle orifice. The unsteady behavior of this highly swirling two-phase flow is typically simulated using the unsteady Volume of Fluid (VOF) method combined with a Reynolds Stress turbulence model in conventional finite-volume CFD. However, the fine mesh resolution required for such simulations significantly increases computational time and cost.

As an alternative, the spray characteristics are evaluated using the mesh-free Smoothed Particle Hydrodynamics (SPH) method implemented in Dive CAE, which employs a specialized weakly compressible SPH (WCSPH) formulation. The use of GPU acceleration enables substantial reductions in computation time compared with conventional CPU-based implementations. In addition, semi-analytical integral boundary conditions are used to improve wall-boundary accuracy.

Axial full cone nozzles are used in a wide range of industrial applications. Their flow capacities range from 0.5 L/min to several thousand L/min, while their spray angles typically range from 30° to 120°. This study investigates three representative nozzles:

- Large-capacity nozzle: 320 L/min at a rated water pressure of 0.5 MPa and a spray angle of 120°, used for desalination applications.
- Medium-capacity nozzle: 6 L/min at a rated water pressure of 0.5 MPa and a spray angle of 60°, used for cooling steel during continuous casting.
- Small-capacity nozzle: 0.6 L/min at a rated water pressure of 0.5 MPa and a spray angle of 45°, used for cooling electric motors.

The simulated spray characteristics, including liquid pressure and flow rate, spray angle, spray pattern volume-flux distribution,are compared with experimental data obtained using optical techniques, laser-sheet imaging, and patternator measurements under identical operating conditions. Since the predicted droplet size distribution is directly dependent on particle resolution in the mesh-free SPH method, this aspect is not addressed in the present study and will be investigated in future work. The objective of this ongoing research is to establish best-practice guidelines for minimizing computational cost while maintaining acceptable prediction accuracy for design optimization. The proposed methodology has the potential to reduce development costs by enabling early-stage virtual design optimization prior to the manufacture and testing of prototypes for application-specific requirements.

Keywords:
Axial Full Cone Spray Atomization, Computational Fluid Dynamics, Smoothed Particle Hydrodynamics, Spray Characterization, Design Optimization

Development of Experimental Test Platforms for Aerospace Propulsion Research: Example Results on 3D-Printed Pintle Injector Sprays and Shock–Droplet Interaction

Min Son*[1], Max Wenderoth[1], Moritz Drost[1], Tobias Sander[1], Lars Zigan[1]

[1]Institute of Thermodynamics, Universität der Bundeswehr München, Germany
*Corresponding author: Min.Son@unibw.de

Abstract

Liquid-fueled aerospace propulsion systems, e.g., liquid rocket engines and high-altitude air-breathing engines, typically operate under extreme conditions. Therefore, investigating the breakup and mixing characteristics of sprays under such conditions is important to develop these engines. For spray investigations in harsh environments, the Institute of Thermodynamics at Universität der Bundeswehr München has recently developed two experimental test platforms; the Cold-flow Spray Testbed for Advanced Rocket Injectors (CSTAR) and the Double-diaphragm Shock Tube for Shock–droplet Coupling Research (DiSCO). CSTAR is designed to investigate the static and dynamic spray characteristics of novel rocket injectors, whereas DiSCO focuses on the fundamental breakup morphology of shock–droplet interactions relevant to supersonic combustion in ramjet and SCramjet applications, as well as high-altitude ignition.

To verify the capabilities of both test platforms, two representative studies were performed. For CSTAR, a slit-type pintle injector for a 500-N-class lunar-lander thruster was designed, and a pintle with 12 slits was manufactured using laser-sintering additive manufacturing. Two different pintle-tip geometries were investigated using high-speed shadowgraphy while varying the momentum flux ratio. Based on the recorded spray images, the spray angles and radial spray distributions were compared to determine optimal operating conditions for soft-landing scenarios. For DiSCO, the breakup morphology of droplets exposed to a shock wave was investigated using high-speed shadowgraphy to capture the short-duration breakup process. The injector, shock-tube operation, and high-speed camera were successfully synchronized, and the bag breakup of multiple droplets was observed after the shock wave passed.

These verification studies confirmed that both test platforms can be operated as intended. Further improvements, e.g., a single-hole droplet dispenser for DiSCO and Mie-scattering measurement for CSTAR, will be conducted to obtain more precise quantitative and qualitative experimental data and to support fundamental research on spray and breakup phenomena relevant to aerospace propulsion systems.

Keywords: Experimental Test Platform, Spray, Pintle Injector, Shock–Droplet Interaction, Aerospace Propulsion

Acknowledgements

This research is partially funded by dtec.bw – Digitalization and Technology Research Center of the Bundeswehr under the project MaST – Macro/Micro-Simulation of Phase Decomposition in the Transcritical Regime. dtec.bw is funded by the European Union – NextGenerationEU. The authors gratefully acknowledge the technical support and research infrastructure provided by Universität der Bundeswehr München. The authors thank Jan-Erik Brune and Prof. Christian Mundt for supporting the shock-tube experiments.

Numerical Assessment of Multi-Jet Injection into turbofan exhaust

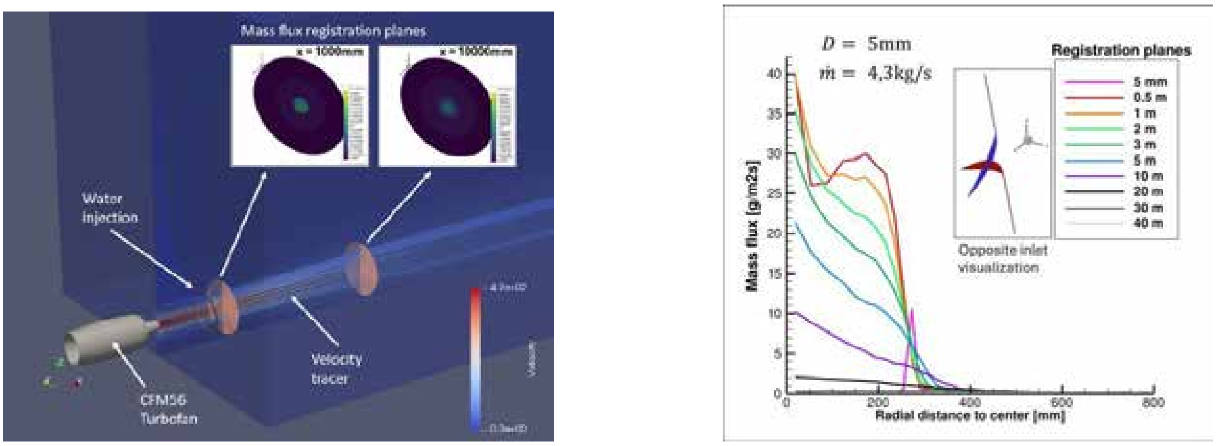
Holger Huck[1], Corine Kieffer-Roth[1], Hannah Scheffold[1],
Benedict Enderle[1], Georg Eckel[1], Andreas Huber[1]

[1]German Aerospace Center, Institute of Combustion Technology, Stuttgart

Abstract

Non-volatile particulate matter, primarily consisting of ultrafine soot particles, is one of the main aircraft emissions affecting local air quality around airports [1]. One possible approach for reducing these emissions is to inject water droplets into the exhaust stream with the aim of capturing and removing particulate matter. Post-maintenance ground tests represent a suitable operating case for local exhaust treatment, as the aircraft remains stationary while its engines are operated at different power settings. As an initial step towards such a treatment concept, this work numerically investigates multi-jet water injection into the separate bypass and core flows of a representative A320/CFM56 exhaust configuration [2]. Spray penetration, evaporation and water distribution are assessed as key requirements for achieving contact between the injected droplets and the particle-laden exhaust flow.

The continuous gas flow from the velocity profile is calculated with the DLR in-house CFD solver ThetaCOM and subsequently used as background flow for Lagrangian spray simulations with the in-house code SPRAYSIM. Water is injected through 16 equally distributed nozzles arranged around the exhaust and directed towards the core center; the spray setup is schematically shown in Fig 1a). The investigated parameter range includes nozzle diameters from 2.5 to 10 mm and water mass flow rates from 1 to 10 kg/s per nozzle. The simulations consider semi-empirical atomization models in a jet-in-crossflow initialization, secondary breakup and evaporation [3]. Spray penetration and distribution are evaluated from droplet trajectories and radial water mass-flux profiles at several downstream registration planes, shown for one case in Fig 1b).



a) b)

Fig 1: a) Visualization of the setup behind the turbine. The injection is considered a jet-in-crossflow configuration. Two mass flux registration planes downstream are shown. b) Radial distribution of water mass flux trough registration planes downstream showing good coverage for the presented case. Two opposite jets are shown to visualize the overlap and increased mass flow close to the center.

The outcome of the investigated range was a representative configuration with a nozzle diameter of 5mm and a water mass flow rate of 4.3kg/s per nozzle, providing complete penetration and downstream spreading. The sprays penetrate from the bypass stream towards the central exhaust region. Increasing the nozzle diameter shifts the generated droplet-size distribution towards larger droplets. Their greater inertia improves radial penetration but reduces their response to the gas flow and therefore limits downstream spreading. The mass-flux profiles further show how the initially concentrated jets spread across the exhaust with increasing downstream distance.

The simulations resolve the penetration and downstream distribution of multiple interacting sprays within the non-uniform core and bypass flows of a turbofan exhaust. The results highlight the balance between radial penetration and spray spreading when selecting suitable injection conditions. Building on the established ThetaCOM-SPRAYSIM setup, further investigations will address variations in injection geometry and operating conditions which reduce evaporation. Future extensions may also include interactions between the liquid phase and selected particulate matter.

References

[1] ICAO Committee on Aviation Environmental Protection, Impacts and Science Group, *Air Quality and Climate Impacts: Interdependencies and Trade-Offs of Aviation Emissions*, International Civil Aviation Organization, 2025, ICAO report.

[2] Afkari et al., *Evaluating Bypass Effects of Advanced Turbofan Engines on Contrail Formation Using Large Eddy Simulations*, Journal of Aircraft, vol. 63, 2026, https://doi.org/10.2514/1.C038376.

[3] Eckel et al., *Semi-Empirical Model for the Unsteady Shear Breakup of Liquid Jets in Cross-flow*, Atomization and Spray, vol. 26, 2016, https://doi.org/10.1615/AtomizSpr.2015011185.

Stabilization of Ethanol-in-Oil Emulsion Produced with an Injector Nozzle In-Line with a Microchannel Mixer

Arthur V. S. Oliveira*[1], Michael Bailly[2], Jérôme Bellettre[2]

[1]GOTAS/LETeF, Department of Mechanical Engineering, São Carlos School of Engineering, University of São Paulo, São Carlos, Brazil
[2]Laboratoire de Thermique et Énergie de Nantes, CNRS, Nantes Université, Rue Christian Pauc, B.P. 50609, Nantes, 44306, France
*Corresponding author: jerome.bellettre@univ-nantes.fr

Abstract

Ethanol-in-oil emulsions are attracting interest as renewable fuels for compression-ignition engines, allowing a low-cost, low-viscosity component to be blended into vegetable oil without the equipment penalties associated with pure vegetable oil combustion. However, ethanol and vegetable oils are only partially miscible, and emulsions produced by simple mechanical mixing typically phase-separate within minutes. In this work, ethanol-in-sunflower-oil emulsions were produced by co-injecting both fluids, each delivered by an independently controlled volumetric piston pump, through an automotive direct-injection nozzle placed in-line with a microchannel mixer immediately upstream of the injector. Emulsions containing 20, 30 and 40%v ethanol were sprayed at a constant total flow rate of 100 ml/min and collected for stability assessment. Visual inspection of the collected samples showed that emulsions with up to 30 %v ethanol remained visually homogeneous for at least 24 h, only phase-separating after 72 h, whereas the 40%v mixture never formed a stable emulsion (Figure 1a). Droplet size distributions (Figure 1b), obtaining from images collected with a high-resolution optical microscope, show a similar primary droplet size for both emulsions (D10 ≈ 5 μm), but the 40 %v emulsion exhibits a long tail up to ~23 μm and a much larger Sauter mean diameter (D32 = 12.1 μm vs. 5.2 μm for 20 %v), consistent with droplet coalescence and its instability. Remarkably, this stability window lies within a composition range that, according to the ternary liquid-liquid equilibrium of the ethanol-sunflower oil system, should thermodynamically split into two liquid phases already at 20 %v ethanol — indicating that the injector-microchannel arrangement produces a metastable emulsion well beyond the equilibrium miscibility limit.

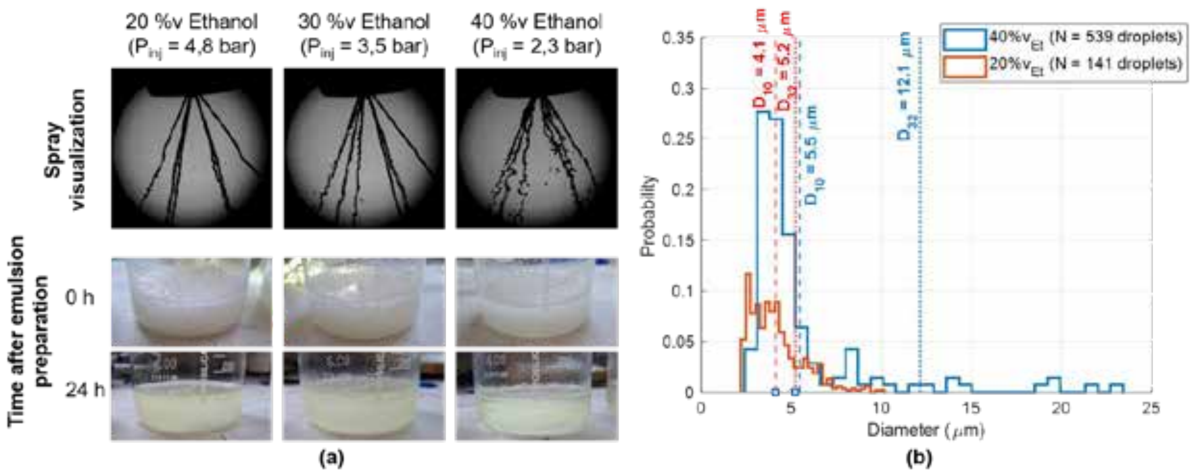


Figure 1. *Results for ethanol-in-oil emulsion preparation: a) spray images (100 ml/min, high-speed shadowgraphy) and emulsion appearance at 0h and 24h after preparation, for ethanol-in -oil emulsions from 20 to 40 %v ethanol; b) probability density function of the ethanol droplet size for 40%v and 20%v ethanol content in the emulsion, including mean diameter (D_{10}) and Sauter mean diameter (D_{32}) values.*

Keywords: Ethanol-oil emulsion, spray injection, microchannel mixer, emulsion stability

Spatial evolution of liquid sheets formed by impinging jets under subcooled and flash-boiling conditions

Hernan Amaya[1], Bartosz Kaźmierski[1], Łukasz Jan Kapusta*[1]

[1]Institute of Heat Engineering, Warsaw University of Technology, Poland
*Corresponding author: lukasz.kapusta@pw.edu.pl

Abstract

Impinging-jet atomisation relies on momentum exchange at the collision point to promote liquid sheet breakup and finer droplet formation compared to free jets. While superheating of the injected liquid (flash boiling) is known to affect efficient momentum exchange and alter macroscopic spray structures, the micro-scale fluid dynamics occurring within the immediate collision zone and their effect on the downstream liquid structures remain unexplored. This study investigates local sheet morphology at the impingement point (12 mm from the nozzles) and downstream sheet evolution (19 mm from the nozzles), aiming to link near-field structural changes with downstream breakup consequences across orthogonal visualisation planes.

The experimental system delivers deionised water through two 0.7 mm cylindrical nozzles arranged at a 30° inclusion angle across gauge pressures ranging from 1.1 to 7.5 bar (Re≈13,000-81,000, We≈1,600-13,300). High-resolution microscopic shadowgraphy images were acquired at an effective spatial resolution of 3.3 μm/pixel from both parallel and perpendicular optical axes relative to the jets' plane.

Under subcooled conditions (40°C), jets' collision produced structured liquid sheets adhering to classical hydrodynamic breakup regimes. Under moderate-flash-boiling conditions (120°C), perpendicular-plane visualisations revealed that thermal effects disrupt the incoming jets prior to the collision point, altering momentum alignment at the collision origin. Downstream, superheating amplifies out-of-plane sheet flapping, driving violent edge perforation and primary ligament detachment. The spatial comparison shows that the flash boiling accelerates early film breakup compared to subcooled injection.This affects downstream spray formation, producing a progressively wider, fan-shaped expanding liquid sheet with intensified edge-ligament detachment, internal film perforation, and enhanced lateral droplet dispersion downstream.

Keywords: Impinging jets, Flash boiling, Liquid sheet breakup, Surface wave instabilities.

Acknowledgements

This research was funded in whole or in part by the National Science Centre, Poland, within the framework of the SONATA programme, grant number: 2020/39/D/ST8/00947. For the purpose of Open Access, the author has applied a CC-BY public copyright licence to any Author Accepted Manuscript (AAM) version arising from this submission.

Reduced Gas Consumption and Enhanced Powder Fineness for Additive Manufacturing by High-Temperature Gas Atomization

Anne K. Geppert[1*], Aline Weicht[1], Stefan Evers[2], Lydia Achelis[2], Volker Uhlenwinkel[1], Udo Fritsching[1,2]

[1]Leibniz Institute for Materials Engineering - IWT, Bremen, Germany
[2]Particles and Process Engineering, Faculty of Production Engineering, University of Bremen, Germany

*Corresponding author: geppert@iwt.uni-bremen.de

Abstract
Spherical metal powders are a critical feedstock for various Additive Manufacturing processes. Industrially, they are predominantly produced via gas atomization of molten metals, a process that typically involves high consumption of inert gases. Elevating the gas temperature during atomization presents a promising pathway to significantly reduce gas usage while maintaining or enhancing powder quality. As gas temperature increases, gas density decreases, leading to reduced mass throughput, but simultaneously increasing gas velocity and viscosity, factors that promote the formation of finer particles. While current industrial hot-gas atomization systems operate up to 400°C, flow simulations suggest that even higher temperatures could further decrease gas consumption offering substantial cost savings, particularly with expensive gases like argon, and improved environmental performance in terms of CO_2 footprint.

To investigate the effects of high gas temperatures on atomization, a novel High-Temperature Close-Coupled Atomizer (H-CCA) was developed, capable of operating up to 1000°C. Experiments with steel melt showed gas consumption reduced by up to 50% at 700-800°C compared to cold-gas atomization, while the fine fraction (<20 µm) increased by 25-30% at 800°C and 14-16 bar. Copper alloy (CuSn6) powder was analyzed via dendrite arm spacing to assess cooling rates, which were validated against numerical simulations. An existing solidification model was adapted to account for hot-gas conditions. This study establishes foundational design principles for next-generation H-CCA systems, enabling accurate prediction of median particle size and solidification rates - key factors for the industrial implementation of this energy-efficient and resource-saving technology.

Keywords: melt atomization, high-temperature gas atomization, powder, cooling rates.

References
[1] S. Hussain, C. Cui, L. He, L. Mädler, V. Uhlenwinkel, *Journal of Materials Processing Technology,* 2020, 282, 116677, https://doi.org/10.1016/j.jmatprotec.2020.116677.

Advanced Optical Instrumentation for Spray, Droplet, and Particle Characterization

Mason D. Leandro*[1,2], William D. Bachalo[1], Khalid M. Ibrahim[1], Gregory A. Payne[1]

[1]Artium Technologies, Inc., Sunnyvale, California, USA
[2]Earth and Planetary Sciences, University of California Santa Cruz, USA
*Corresponding author: mleandro@artium.com

Abstract

Accurate characterization of droplets and particles is essential to the development and control of atomization, precision droplet deposition, coating, combustion, pharmaceutical, atmospheric, and aircraft icing processes. Translating optical diagnostic methods from laboratory demonstrations into practical industrial and field instruments introduces additional requirements for stability, robustness, and ease of operation. Instruments must maintain alignment, operate over broad ranges of particle size and concentration, adapt to changing test conditions, and provide results that can be readily interpreted by engineers and operators. Artium Technologies, Inc. develops highly refined optical instrumentation that translates advances in spray diagnostics into reliable and easy-to-use measurement systems for laboratory, production, wind tunnel, and airborne applications.

Phase Doppler interferometry (PDI, aka PDA, PDPA) is the central technology in many of these systems. PDI provides simultaneous, non-intrusive measurements of the diameter and velocity of individual spherical droplets in polydisperse sprays. Depending on the optical configuration, one, two, or three velocity components may be measured simultaneously with droplet diameter. These measurements can be used to determine size and velocity distributions, number density, volume flux, and other spray statistics. In industrial test facilities, wind tunnels, and airborne measurement campaigns, available test time is often limited and costly. Efficient spray characterization therefore requires statistically representative measurements at an optimized data rate, coverage of broad droplet size distributions with minimal reconfiguration, and accurate characterization of the effective sample area used to convert droplet counts into concentration-based quantities.

Recent PDI developments at Artium have focused on these aspects of measurement efficiency and are described further in our oral presentation, “Recent Advances in Phase-Doppler Interferometry for Efficient Spray Characterization,” Abstract #105. One development addresses signal coincidence, which occurs when multiple droplets occupy the probe volume simultaneously. The measured droplet concentration and Poisson statistics are used to estimate the probability of coincidence and automatically select, in real time, the receiver slit width that maximizes the effective sample area while maintaining an acceptable coincidence probability. When coincidence inevitably occurs due to random droplet arrivals, a patented signal parsing approach separates signals generated by individual particles and recovers measurements that might otherwise be rejected or produce errors. Controlled testing with monodisperse droplet streams showed that this approach substantially increased the recovered droplet count without introducing measurable bias.

A second development expands the dynamic droplet size range available at a single instrument setting to approximately 100:1 (i.e., 1-100 μm). Broad polydisperse sprays may otherwise require repeated measurements at several detector gain settings followed by merging of the resulting distributions. The expanded dynamic size range reduces the need for repeated measurements and avoids uncertainty associated with changes in spray conditions between acquisitions. Finally, a probe volume mapping method combines a high-resolution motorized traverse with a monodisperse droplet generator to quantify each dimension of the effective sample area under controlled laboratory conditions. The measurements verify sizing performance throughout the detectable region, evaluate assumptions regarding probe volume geometry, and provide laboratory-derived corrections that improve the accuracy of concentration-based quantities. Together, these developments extend PDI capability into more challenging spray regimes, including higher concentrations and broader size distributions, while improving confidence in derived spray metrics.

These processing and validation methods are implemented across several PDI instrument configurations. Artium's Modular PDI systems are intended for research and development applications requiring flexibility in measurement geometry, size range, velocity components, and working distance. Compact diode pumped solid state lasers and self-contained transmitter and receiver assemblies provide stable optical systems with a long lifetime. Automated setup procedures and computer-controlled slit aperture selection adapt the optical and sampling parameters to changing spray conditions with minimal user intervention. The systems can also be integrated with automated traverses to map spray size, velocity, concentration, and flux.

The Turnkey PDI, or TK-Series, systems use the same measurement principles but are factory configured and calibrated for applications in which consistent setup and operation are priorities. Their fixed optical arrangement eliminates routine field alignment and is designed to maintain calibration over the life of the instrument. Inkjet Droplet Measurement (IDM) systems are PDI instruments specifically designed for inkjet printhead development, nozzle optimization, and quality control. They provide real-time measurements of droplet size, velocity, and trajectory without disturbing the droplet stream. Automated positioning allows nozzle arrays to be scanned for variations in drop volume, speed, and trajectory.

For atmospheric research and aircraft icing applications, PDI flight probes package the transmitter, receiver, signal processing electronics, and measurement region within a compact aerodynamic structure. Dual range configurations combine two overlapping PDI channels, allowing cloud droplets and supercooled large drops to be measured with a single instrument over a nominal range of approximately 1-2000 µm. Integrated heaters limit ice accumulation on exposed components, while high speed acquisition supports measurements aboard research aircraft and in high speed icing wind tunnels. The individual particle measurements provide size and velocity distributions together with derived cloud properties such as number concentration and liquid water content.

PDI is complemented by our Particle Imaging (Particle-I) systems for applications in which particle morphology or non-sphericity is important. Multiple optical configurations provide high resolution images of droplets, ice crystals, and other particles from micron to millimeter scales. Short duration LED illumination (20 ns or less) minimizes motion blur, while double pulse configurations enable particle tracking velocimetry. These systems have been applied to laboratory sprays, mixed phase icing clouds, supercooled large drop studies, and atmospheric flight measurements. Our large field-of-view disdrometer extends the approach to simulated and natural rain environments where particle sizes up to several millimeters have been recorded, and our patent-pending Scheimpflug imaging approach enables three-dimensional particle imaging while increasing the measurement volume by approximately an order of magnitude.

The development of these instruments has depended on sustained interaction among instrument developers, academic researchers, government laboratories, and industrial users. Applications include precision deposition, inkjet printing, spray and aerosol research, cloud microphysics, aircraft and engine icing, combustion diagnostics, and environmental particulate measurements. Tests aboard aircraft and in icing tunnels, propulsion facilities, and high speed rain environments provide feedback unavailable from laboratory development alone. This experience has driven improvements in optical packaging, automated setup, signal validation, environmental protection, data analysis, and application-specific software. Artium's research and development program has been supported by 37 Small Business Innovation Research awards and has produced 20 issued or pending patents. Our researchers have been the recipients of several awards including one Lefebvre Award, two Marshall Awards, two Tanasawa Awards, and numerous certificates of recognition from the National Aeronautics and Space Administration (NASA). We continue to extend the range, accuracy, and practicality of optical diagnostics while working with the spray research community to translate scientific advances into dependable measurement tools for emerging industrial and research challenges.

Keywords: Phase Doppler interferometry, spray characterization, particle imaging, spray diagnostics

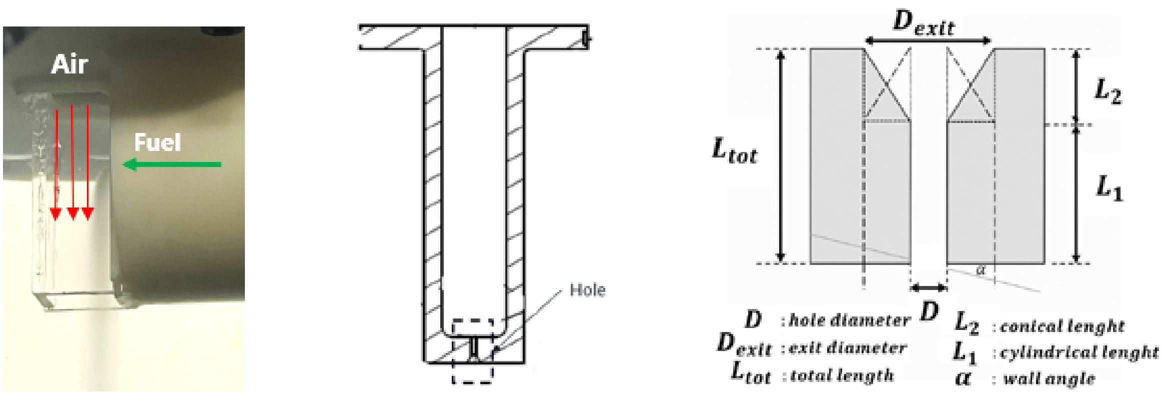
Impact of Nozzle Shape and Dimensions on liquid Jet-In-CrossFlow Spray characteristics for the Design of a Multi-Staged Fuel Injection System

Amandine Tomba*[1], Ahlem Ghabi[1], Romain Janodet[1], Yann Meslem[1], Laurent Kubek[2], Frantz Gouriou[2], Jean-Philippe Le Ru[2]

[1] Safran Aircraft Engines, Combustion design office, Villaroche, France
[2] CERTAM, Technopole du Madrillet, France
*Corresponding author: amandine.tomba@safrangroup.com

Abstract
The optimization of fuel injection systems is a crucial step in the design process of an aircraft engine, as it conditions both the performances and the operability (e.g. high-altitude relight) of the combustion chamber. It is also critical to reduce pollutant emissions. To this aim, a lean-combustion technology is developed, the design of a Multi-Staged Injection System (MSFI) being the starting point. In particular, it is necessary to understand the underlying physics of jet-in-crossflow atomization in such systems, at the relevant operating points in order to improve fuel-air mixing in the swirler. Thus, an extensive experimental campaign has been performed to study the impact of a single nozzle on liquid spray characteristics using the setup of Fig. 1, supplemented by high-fidelity CFD calculations. A particular focus has been made on operating points with low momentum flux ratios ($q < 4$) and gaseous Weber number ranging from 100 to 2400 reported in Tab. 1, which is relevant for the targeted application. Six nozzle shapes (cylindrical hole "type A", divergent nozzle "type B", convergent nozzle "type C", cylindrical step-hole "type D" and cylindrical hole with a slanted inlet wall "type E") were studied with dimensional variations identified by a number of parameters summarized in Tab. 2. The results of this campaign, partially presented in Fig. 2, feature comparisons of liquid jet penetrations obtained using shadowgraphs and of droplet size distributions (SMD, PDF) with the first mode and the filming mode identified from a laser diffraction particle size analysis for each operating point. The first mode depend on operating conditions, whereas the second mode consisting of larger droplets characteristic of a filming phenomenon.

Keywords: Experimental atomization; Laser diagnostics; Jet-in-crossflow; Nozzle design



(a) (b) (c)

Figure 1. *(a) Experimental setup with jet development In the CrossFlow, (b) Single-orifice injector schematic, (c) Schematic of the different injection orifice designs*

Operating point	We	q	M	U_a (m/s)	W_a (g/s)	U_f (m/s)	W_f (g/s)
1	1520	0.24	0.70	242.5	41.36	4.57	0.71
2	233	0.83	0.27	95.0	16.19	3.34	0.52
4	118	1.00	0.20	67.6	11.52	2.61	0.41
7	118	1.31	0.20	67.6	11.52	2.98	0.47
12	118	4.00	0.20	67.6	11.52	5.21	0.81
14	1520	4.00	0.70	242.5	41.36	18.72	2.92

Table 1. *Representative selection of operating points defined by the momentum flux ratio (q), the gas Weber number (We), the Mach number (M), the air and fuel velocity U_a and U_f, and the air and fuel mass flow rate W_a and W_f*

ID	D (mm)	L_{tot} (mm)	L_{tot}/D	D_{exit} (mm)	L_1 (mm)	L_2 (mm)	Ra (μm)	α (°)
				Cylindrical hole				
A1	0.5	1	2.0	0.5	1	0	1.6	0
A2	0.5	3	6.0	0.5	3	0	1.6	0
A3	0.5	3	6.0	0.5	3	0	0.8	0
A4	1.0	3	3.0	1.0	3	0	1.6	0
A5	1.0	6	6.0	1.0	6	0	1.6	0
				Cylindrical hole with a diverging nozzle				
B1	0.5	3	6.0	1.66	2	1	1.6	0
B2	0.5	2.4	4.0	1.66	1.38	1	1.6	0
B4	0.5	3	6.0	2.0	2	1	1.6	0
				Cylindrical hole with a converging nozzle				
	D = D_{exit}			D_{max}				
C1	0.5	3	6.0	1.66	2	1	1.6	0
				Cylindrical step hole				
D1	0.5	3	6.0	1.66	2	1	1.6	0
				Cylindrical hole with a slanted inlet wall				
E2	0.5	3	6.0	0.5	3	0	1.6	65
E4	0.5	3	6.0	0.5	3	0	1.6	40

Table 2. *Geometrical parameters of the different injection hole configurations*

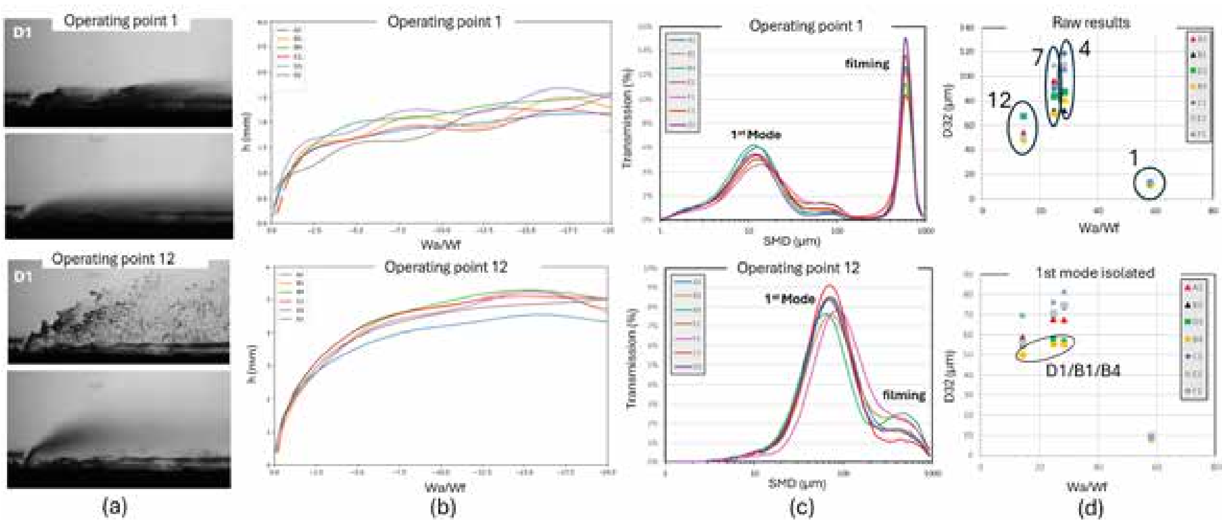


Figure 2. *(a) Example of instantaneous (top) and time-averaged (bottom) shadowgraph images for configuration D1 and operating points 1 and 12, (b) Corresponding penetration curves and (c) Probability Density Function (PDF) with the 1^{st} mode and the filming mode identified for all configurations, (d) Sauter Mean Diameter (or D32) as a function of the Wa/Wf ratio considering all operating modes (top) and with the 1st mode isolated (bottom)*

A Retrofit Electrostatic Pressure Swirl Atomizer for Improving Industrial Spray Performance Without Nozzle Redesign

Baraya E.A.*[1], Ibrahim I.A.[2], Gad H.M.[2]
[1]Suez Canal Authority, Dredging Department, Ismailia, Egypt
[2]Mechanical Power Engineering Department., Faculty of Engineering, Port Said University, Egypt
*eslam.ahmad@eng.psu.edu.eg

1. Industrial Challenge

Pressure swirl atomizers are extensively used in industrial burners, gas turbines, boilers, and other liquid-fuel spray systems because of their simple design and reliable operation. However, improving atomization performance usually requires increasing injection pressure or redesigning the nozzle geometry, leading to higher energy consumption, increased pumping requirements, greater manufacturing costs, and limited operational flexibility. A practical retrofit solution capable of improving spray quality without modifying the existing hydraulic design remains an important industrial requirement.

2. Industrial Solution

This work presents an Electrostatic Pressure Swirl Atomizer (ESPSA) that combines conventional hydraulic swirl atomization with electrohydrodynamic charging while preserving the original hydraulic geometry. The proposed system introduces the electric field as an additional operating parameter without increasing injection pressure or modifying the atomizer design, allowing existing pressure swirl atomizers to be upgraded through electrical integration only.

3. Technology Description

The ESPSA was manufactured with identical hydraulic dimensions to a conventional PSA. Electrical insulation was achieved using polyethylene components capable of safely withstanding high-voltage operation while maintaining unchanged internal flow characteristics. Both atomizers were tested under identical operating conditions to isolate the influence of electrostatic charging. Spray cone angle, breakup length, droplet-size distribution, SMD, spray concentration, and spray momentum were experimentally evaluated using validated optical imaging, ImageJ analysis, inline patternation, and momentum measurement techniques.

4. Industrial Performance

The proposed ESPSA produced significant improvements in atomization performance (Figs. 3–7 and Table 1). Increasing the applied voltage from 10 to 40 kV increased the spray cone angle from 52° to 74°, reduced breakup length by approximately 94%, and decreased SMD from 63 μm to 30 μm. Compared with the conventional PSA under identical hydraulic conditions, the ESPSA achieved up to 70–75% reduction in SMD, transformed the spray from a hollow cone into a partially filled cone, and produced more uniform spray concentration and momentum distributions.

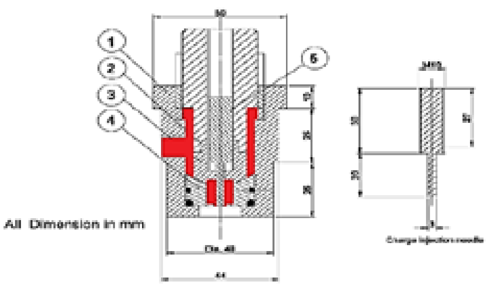


Fig. 1. Detailed dimensions of the Electrostatic Pressure Swirl Atomizer (ESPSA).

Fig. 2. Components and assembly of the conventional PSA and electrostatic pressure swirl atomizer (ESPSA).

5. Industrial Benefits

The proposed technology enables existing pressure swirl atomizers to achieve finer atomization without increasing hydraulic power or redesigning the injector. The improved spray quality is expected to enhance fuel-air mixing, increase combustion efficiency, reduce fuel consumption and pollutant emissions, improve process stability, and provide electrically adjustable spray characteristics without additional mechanical complexity. Because the original hydraulic nozzle remains unchanged, implementation only requires electrical insulation and high-voltage integration, making the proposed ESPSA a practical low-CAPEX retrofit solution that can be readily integrated into existing industrial systems.

Fig.3 Spray shape at different P_{inj} with L_o/D_o=0.22 , WxH=1.5x1.5mm², Ø = 90° and L_s/D_s = 0.93 for ESPSA of 30 kV after image processing.

6. Industrial Applications

The proposed ESPSA can be readily integrated into industrial burners, gas turbines, boilers, liquid-fuel combustion systems, spray drying, coating technologies, agricultural spraying, and other industrial spray processes where improved atomization quality, higher process efficiency, and lower emissions are required.

Spray Parameter	PSA	ESPSA
Cone Angle	58°	70°
Breakup	17 mm	5 mm
SMD	75 µm	21 µm
Spray Pattern	Hollow Cone	Filled Cone
Spray Momentum Distribution	Hollow-core, non-uniform (Peripheral concentration)	More uniform, center-filled (Uniform radial distribution)

Table 1. Performance improvements achieved by the proposed ESPSA under identical operating conditions.

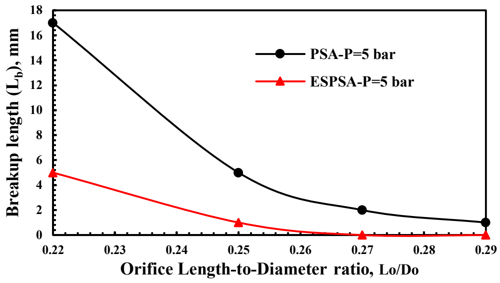


Fig.4 Effect of L_o/D_o on Lb for PSA and ESPSA of 20KV, P=5 bar, Ø = 90°, D_s=10 mm and WxH =1.5x1.5 mm²

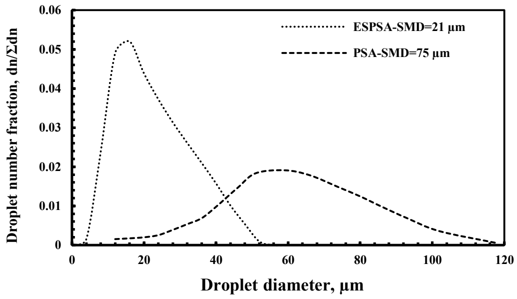


Fig.5 Droplet size distribution at P = 4 bar, WxH =1.5x1.5 mm², L_o/D_o = 0.27, L_s/D_s = 0.775 for PSA, and ESPSA of 40 kV.

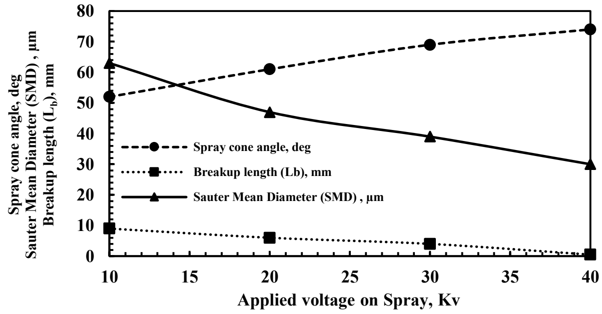


Fig. 6. Effect of Applied Voltage on the SCA, SMD and L_b of the ESPSA at fixed geometry.

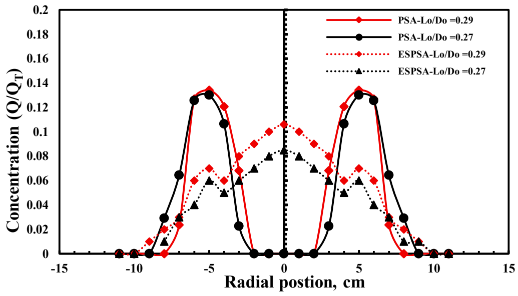


Fig.7 Effect of L_o/D_o on SCD of ESPSA at 40 Kv and PSA, at P = 9 bar, h = 22 cm, WxH =1.0x1.0mm² and L_s/D_s = 0.775.

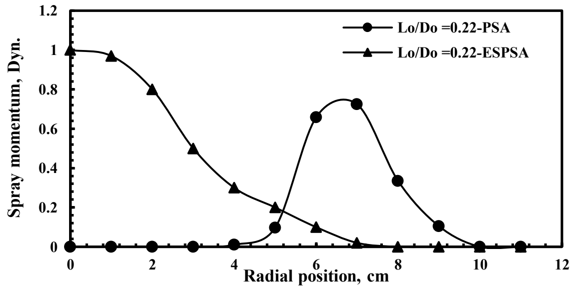


Fig. 8 Radial distribution of spray momentum for PSA and ESPSA at L_o/D_o = 0.22, at P = 4 bar, WxH= 1.5 × 1.5 mm² and L_s/D_s=0.664.

7.Industrial Significance

Unlike previous electrostatic atomization studies, the proposed ESPSA preserves the original hydraulic geometry, allowing the observed improvements to be directly attributed to electrostatic charging rather than nozzle modification. This provides a reliable benchmark for evaluating electrohydrodynamic enhancement in industrial pressure swirl atomizers and facilitates future industrial adoption of electrostatic retrofit technologies.

8.Conclusion

The proposed ESPSA demonstrates that electrohydrodynamic assistance can be successfully integrated into existing pressure swirl atomizers without modifying the original hydraulic design. The technology provides a practical, low-cost retrofit solution capable of significantly improving atomization performance while enhancing energy efficiency and reducing emissions, making it highly attractive for future industrial combustion and advanced spray applications.