ICLASS 2012

12th International Conference
on Liquid Atomization and Spray Systems

Book of Abstracts

September 2-6, 2012 Heidelberg, Germany

12th International Conference on Liquid Atomization and Spray Systems

Local Host: Eva Gutheil, IWR, Universität Heidelberg, Germany

Program Chairs: Eva Gutheil, IWR, Universität Heidelberg, Germany
Cam Tropea, Technische Universität Darmstadt, Darmstadt, Germany

Exhibition: Cam Tropea, Technische Universität Darmstadt, Darmstadt, Germany

Local Organizing Team:
Ellen Vogel (Conference Secretary)
Julia Feuerer
Ingrid Hellwig
Julia Schumacher
Manfred Trunk (Web Administrator)

Book of Abstracts and Proceedings:
Le Cao
Srikanth Reddy Gopireddy
Yong Hu
Rana Muhammad Humza
Hernán Olguín Astudillo

With the support of ILASS Europe and ILASS International

Exhibitors

A2 Photonic Sensors

CD-adapco
www.cd-adapco.com

DANTEC DYNAMICS

FMP TECHNOLOGY GMBH
FLUID MEASUREMENTS & PROJECTS

IGNITE3D ENGINEERING
Convergent Science

KuK
KAMIKA Instruments

LaVision
We count on Photons

LECHLER

OXFORD LASERS
IMAGING DIVISION

Photron

Raydiance
Precision Like No Other

Schlick
Atomizing technologies
Living for Solutions.

Springer
the language of science

SYMPATEC
Sympatec GmbH
System-Partikel-Technik
www.Sympatec.com

TSI
UNDERSTANDING, ACCELERATED

VKT
Video Kommunikation GmbH
Technisches Fernsehen

Foreword

Welcome to the 12th ICLASS meeting. The meeting has been organized by Professor Eva Gutheil of Interdisziplinäres Zentrum für Wissenschaftliches Rechnen, Universität Heidelberg, and representing ILASS Europe. Organizing such a meeting is an enormous undertaking which requires a great deal of planning and careful execution. We are grateful for her exceptional effort in making the arrangements and expediting the many tasks essential to the preparation of a successful conference. Thanks to the persuasive efforts of Professor Oskar Haidn, our conference is taking place at Universität Heidelberg located in the center of the beautiful and charming city of Heidelberg, long recognized for its excellent university, scholarly pursuits, and cultural excellence. Thus, being able to explore the local historical and cultural sites is an added benefit to attending the conference.

Support from many individuals who have contributed their research work for presentation at the meeting is a critical component to the meeting's success. We are grateful for your contributions and the help of many individuals who diligently reviewed the abstracts and final manuscripts. This has helped refine the content and ensures that the presentations are of high quality. Thanks to Professor Cam Tropea for sharing in the responsibility of Program Chair with Professor Gutheil in organizing the technical program. We are also pleased to have three excellent invited lectures highlighting research activity in key areas of interest to the sprays community. We thank Professor Arai, Professor Doerr, and Professor Herrmann for agreeing to present these lectures. As a part of the conference, we are also grateful for Professor Tropea's work as Exhibits Chair in organizing the exhibition of commercial products for spray characterization. Thanks are also due to the exhibitors of the instrumentation who will show their recent developments.

ICLASS began in Japan in 1978 and was followed by a meeting at Madison, Wisconsin in 1981 where the decision was made to form three regional ILASS organizations. A goal of the organizers was to enhance technical communication and collaboration among international researchers and to provide a venue for broadly disseminating this developed knowledge and technology. It may be coincidental but the evolution of the theory involved in atomization and sprays and innovations in the instrumentation and early computer modeling seem to have caught an inspiration from these meetings. There is still much to be done and thus, offering many opportunities for young researchers.

It is evident that we must observe, study, and understand the entire spray flow fields from a phenomenological and physical point of view, including how the liquid breaks up to form droplets, and the two-phase air-liquid mixing that takes place. Coupling of the spray fluid mechanics, gas phase mixing, and evolution of the spray, which are highly important aspects of the process require further investigation. Availability of detailed experimental information on liquid injection, spray formation and spray interaction with the turbulent gas phase flow would significantly advance the technology. With the extraordinary growth in computer power and advances in computational methods, there is promise to deliver truly rigorous scientific models of these spray phenomena. Such a concerted effort is essential if we are to realize any genuine predictive capability in this very complex area of fluid mechanics. Validation of the predictive efforts will require experiments with much greater detail and depth of information than are currently possible, especially in dense spray environments.

Hopefully, at this conference and in these conference proceedings, you will discover that advances are being made toward reaching these goals.

William Don Bachalo, Ph.D.
President, ILASS International

Scientific and Program Committee

Dr. William D. Bachalo, Artium Technologies, INC., USA

Dr. Josette Bellan, Jet Propulsion Laboratory, Caltec, USA

Prof. Günter Brenn, Technische Universität Graz, Austria

Prof. Antonio Cavaliere, Università Degli Studi Di Napoli Frederico II, Italy

Prof. Sanjeev Chandra, University of Toronto, Canada

Prof. Gianpietro Cossali, Università Degli Studi Di Bergamo, Italy

Prof. Masatoshi Daikoku, Hachinohe Institute of Technology, Japan

Prof. François-Xavier Demoulin, CORIA, France

Prof. Joachim Domnick, Hochschule Esslingen, Germany

Prof. Christophe Dumouchel, Université de Rouen, France

Prof. Udo Fritsching, Universität Bremen, Germany

Dr. Hai-Wen Ge, Ford Motor Company, USA

Prof. Iskender Gökalp, CNRS, France

Prof. Mikhail Gorokhovski, École Centrale Lyon, France

Prof. Eva Gutheil, Heidelberg University, Germany

Dr. Chawki Habchi, IFP Énergies Nouvelles, France

Prof. Oskar Haidn, Technische Universität München, Germany

Prof. Yannis Hardalupas, Imperial College London, UK

Prof. Zhen Huang, Shanghai Jiao Tong University, China

Prof. Arvind Jasuja, Cranfield University, UK

Dr. Jan Jedelsky, Brno University of Technology, Czech Republik

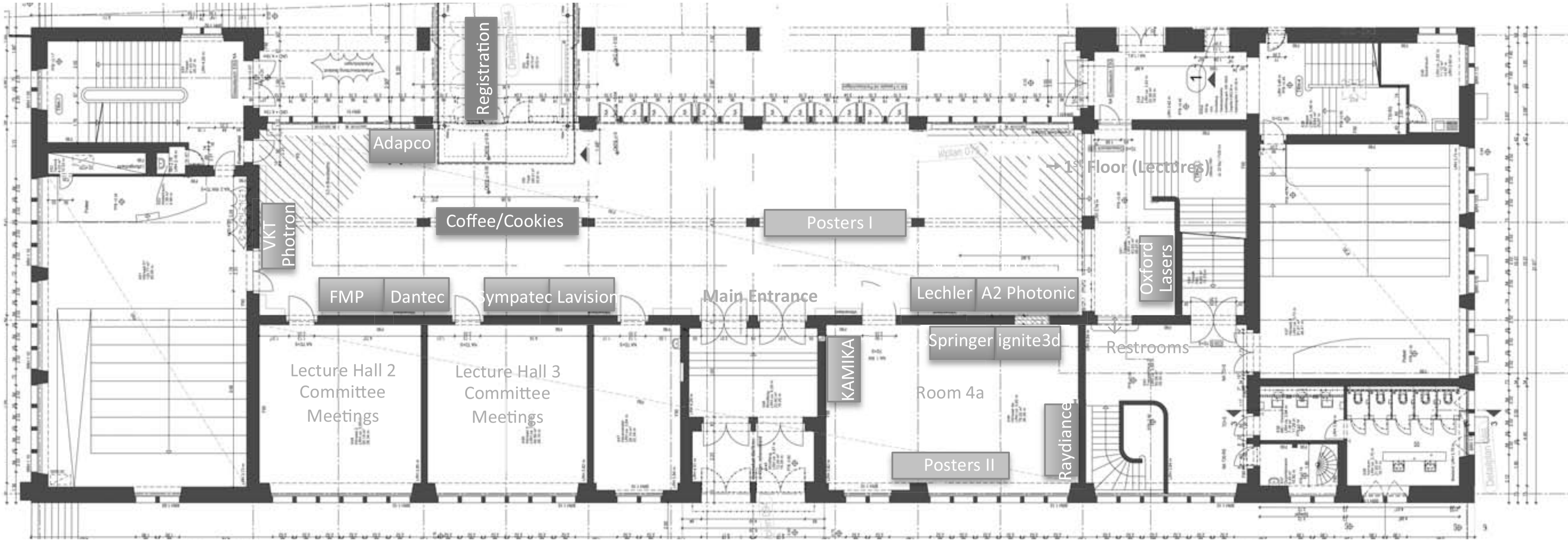
Prof. Nobuyuki Kawahara, Okayama University, Japan

Prof. Reinhold Kneer, RWTH Universität Aachen, Germany

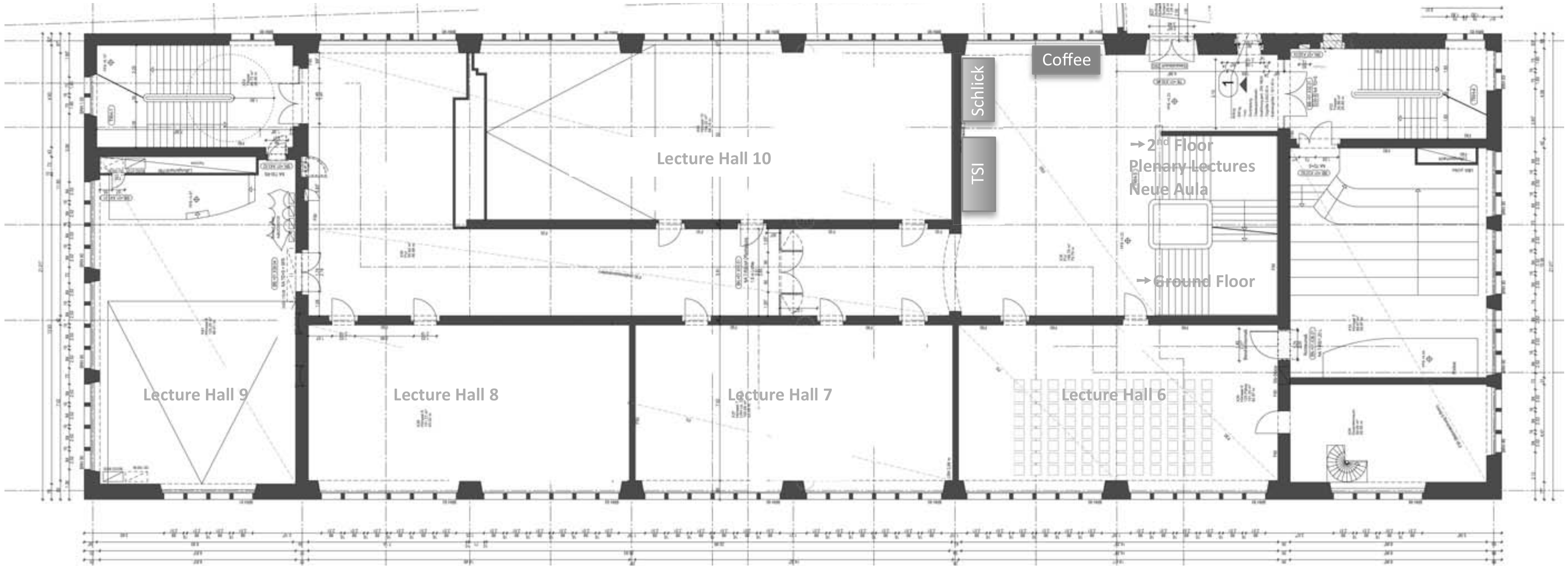
Prof. Fabrice Lemoine, CNRS, France

Neue Universität

GROUND FLOOR



FIRST FLOOR



Scientific Program

Date: Sunday, 02/Sep/2012

5:00pm - 7:00pm	**Sunday Registration** Location: Rottmannsaal
6:00pm - 9:00pm	**Welcome Party** Location: Tiefburg Outdoor activity, please bring warm clothes

Date: Monday, 03/Sep/2012

Pages: **1 – 66**

8:00am - 10:00am	**Conference Registration** Location: Neue Universität				
9:00am - 10:30am	**Conference Opening and Plenary Lecture I** Location: Old Aula Chair: **William Don Bachalo** Chair: **Oskar Haidn** Page: **1**				
10:30am	**Exhibition and Posters** (Pages: **247 - 270**) **Posters I** Location: Foyer Pages: **247 - 256**	**Posters II** Location: Room 4a Pages: **257 - 269**			
10:30am - 11:00am	**Coffee Break**				
11:00am - 12:40pm	**Airblast Atomizers** Location: Lecture Hall 6 Chair: **Min Xu** Chair: **Minori Shirota** Pages: **3 - 6**	**Combustion I** Location: Lecture Hall 7 Chair: **Josette Bellan** Chair: **Benoit Fiorina** Pages: **7 - 10**	**Liquid Jets/Sheet Atomization I** Location: Lecture Hall 8 Chair: **Iskender Gokalp** Chair: **Thomas Kolb** Pages: **11 - 14**	**Pressure Atomizers I** Location: Lecture Hall 9 Chair: **Peter Walzel** Chair: **Ranganathan Kumar** Pages: **15 - 18**	**Diesel Nozzles and Sprays I** Location: Lecture Hall 10 Chair: **Yannis Hardalupas** Chair: **Ezio Mancaruso** Pages: **19 - 22**
12:40pm - 2:00pm	**Lunch Break**		**ILASS International Council Meeting** Location: Lecture Hall 3		
2:00pm - 3:40pm	**Drop Impact I** Location: Lecture Hall 6 Chair: **Marco Marengo** Chair: **Boris Krasovitov** Pages: **23 - 26**	**Combustion II** Location: Lecture Hall 7 Chair: **Raffaele Ragucci** Chair: **Thomas Mosbach** Pages: **27 - 30**	**Liquid Jets/Sheet Atomization II** Location: Lecture Hall 8 Chair: **Vincent G McDonell** Chair: **André Resende Rodrigues Silva** Pages: **31 - 34**	**Alternative Fuels I** Location: Lecture Hall 9 Chair: **Jiro Senda** Chair: **Andreas Kronenburg** Pages: **35 - 38**	**Diesel Nozzles and Sprays II** Location: Lecture Hall 10 Chair: **Lyle M. Pickett** Chair: **Qian Wang** Pages: **39 - 42**
3:40pm - 4:00pm	**Coffee Break**				
4:00pm - 6:05pm	**Drop Impact II** Location: Lecture Hall 6 Chair: **Ana Sofia Moita** Chair: **Norbert Roth** Pages: **43 - 46**	**Combustion III** Location: Lecture Hall 7 Chair: **Ryoichi Kurose** Chair: **Klaus Pachler** Pages: **47 - 50**	**Liquid Jets/Sheet Atomization III** Location: Lecture Hall 8 Chair: **Antonio Lozano** Chair: **Agisilaos Kourmatzis** Pages: **51 - 55**	**Alternative Fuels II** Location: Lecture Hall 9 Chair: **Christian Hasse** Chair: **Seoksu Moon** Pages: **56 - 60**	**Diesel Nozzles and Sprays III** Location: Lecture Hall 10 Chair: **David L.S. Hung** Chair: **Stina Elise Sofie Hemdal** Pages: **61 - 65**
7:00pm - 10:30pm	**ILASS International Board Meeting** Location: Restaurant Weißer Bock				

Date: Tuesday, 04/Sep/2012

Pages: **67 – 128**

8:30am - 9:30am	**Plenary Lecture II** Location: **New Aula** Chair: **William A. Sirignano** Page: **67**				
9:45am - 11:00am	**Drop Impact III** Location: **Lecture Hall 6** Chair: **Junfeng Wang** Chair: **Nobushige Tamaki** Pages: **68 - 70**	**Combustion IV** Location: **Lecture Hall 7** Chair: **William A. Sirignano** Chair: **John Abraham** Pages: **71 - 73**	**Liquid Jets/Sheet Atomization IV** Location: **Lecture Hall 8** Chair: **Franz Joos** Chair: **Volker Gaukel** Pages: **74 - 76**	**Pressure Atomizers II** Location: **Lecture Hall 9** Chair: **Marcus Herrmann** Chair: **Joseph Charles Oefelein** Pages: **77 - 79**	**Diagnostics: Drop Diagnostics I** Location: **Lecture Hall 10** Chair: **Joachim Domnick** Chair: **Lars Zigan** Pages: **80 - 82**
11:00am - 11:25am	**Coffee Break**				
11:25am - 12:40pm	**Modeling/Simulation of Sprays I** Location: **Lecture Hall 6** Chair: **Chawki Habchi** Chair: **Andreas Kempf** Pages: **83 - 85**	**Non-Newtonian (Suspensions) I** Location: **Lecture Hall 7** Chair: **Paul E. Sojka** Chair: **Karl V Meredith** Pages: **86 - 88**	**Droplet Collisions** Location: **Lecture Hall 8** Chair: **Antonio Luis Moreira** Chair: **Ming-Chia Lai** Pages: **89 - 91**	**Electrostatic Sprays** Location: **Lecture Hall 9** Chair: **Saptarshi Basu** Chair: **Christophe Louste** Pages: **92 - 94**	**Diagnostics: Drop Diagnostics II** Location: **Lecture Hall 10** Chair: **Fabrice Lemoine** Chair: **Christian Eigenbrod** Pages: **95 - 97**
12:40pm - 2:00pm	**Lunch Break**		**A&S Editorial Board Meeting** Location: **Lecture Hall 3**		
2:00pm - 3:15pm	**Modeling/Simulation of Sprays II** Location: **Lecture Hall 6** Chair: **François-xavier Demoulin** Chair: **Rudolf J. Schick** Pages: **98 - 100**	**Non-Newtonian (Suspensions) II** Location: **Lecture Hall 7** Chair: **Festus Addo-Yobo** Chair: **Zhixia He** Pages: **101 - 103**	**Multiport Injection** Location: **Lecture Hall 8** Chair: **Laurent Zimmer** Chair: **Norihiko Iki** Pages: **104 - 106**	**Droplet Evaporation** Location: **Lecture Hall 9** Chair: **Günter Brenn** Chair: **Tatsuya Kawaguchi** Pages: **107 - 109**	**Diagnostics: Vapor Measurement** Location: **Lecture Hall 10** Chair: **Eran Sher** Chair: **Alessandro Montanaro** Pages: **110 - 112**
3:15pm - 3:35pm	**Coffee Break**				
3:35pm - 4:50pm	**Modeling/Simulation of Sprays III** Location: **Lecture Hall 6** Chair: **Amsini Sadiki** Chair: **Masaya Muto** Pages: **113 -115**	**Non-Newtonian (Suspensions) III** Location: **Lecture Hall 7** Chair: **Takashi Suzuki** Chair: **Lucio Araneo** Pages: **116 - 118**	**Twin Fluid Atomization** Location: **Lecture Hall 8** Chair: **Franz Josef Durst** Chair: **Seiji Miyashiro** Pages: **119 - 121**	**Alternative Atomizers I** Location: **Lecture Hall 9** Chair: **Udo Fritsching** Chair: **Julien Manin** Pages: **122 - 124**	**Diagnostics: Imaging Techniques I** Location: **Lecture Hall 10** Chair: **William Don Bachalo** Chair: **Simon Gramlich** Pages: **125 - 127**
6:00pm - 7:00pm	**Guided Tour of Heidelberg Castle** Location: **Heidelberg Castle**				
7:30pm - 11:00pm	**Conference Dinner** Location: **Heidelberg Castle**				

Date: Wednesday, 05/Sep/2012

Pages: **129 – 200**

8:30am - 9:30am	**Plenary Lecture III** Location: New Aula Chair: **Jiro Senda** Page: **129**				
9:45am - 11:00am	**Drop Ligament Breakup I** Location: Lecture Hall 6 Chair: **Ta-Hui Lin** Chair: **Carole Planchette** Pages: **130 - 132**	**Interface Modeling I** Location: Lecture Hall 7 Chair: **Xiangyang Zhou** Chair: **Junmei Shi** Pages: **133 - 135**	**Spray Impact I** Location: Lecture Hall 8 Chair: **Mark Allan Linne** Chair: **Julián Palacios** Pages: **136 - 138**	**Alternative Atomizers II** Location: Lecture Hall 9 Chair: **Miroslav Jicha** Chair: **Richard Marcer** Pages: **139 - 141**	**Diagnostics: Imaging Techniques II** Location: Lecture Hall 10 Chair: **Cameron Tropea** Chair: **Elias Kristensson** Pages: **142 - 144**
11:00am - 11:25am	**Coffee Break**				
11:25am - 12:40pm	**Drop Ligament Breakup II** Location: Lecture Hall 6 Chair: **Gianpietro Elvio Cossali** Chair: **Sawitree Saengkaew** Pages: **145 - 147**	**Interface Modeling II** Location: Lecture Hall 7 Chair: **Eva Gutheil** Chair: **Sergei Sazhin** Pages: **148 -150**	**Spray Impact II** Location: Lecture Hall 8 Chair: **Assaad Masri** Chair: **Raul Payri** Pages: **151 - 153**	**Alternative Atomizers III** Location: Lecture Hall 9 Chair: **Nwabueze Giles Emekwuru** Chair: **Christopher F. Powell** Pages: **154 - 156**	**Diagnostics: Imaging Techniques III** Location: Lecture Hall 10 Chair: **Keiya Nishida** Chair: **Thomas Dreier** Pages: **157 - 159**
12:40pm - 2:00pm	**Lunch Break**	**ILASS Asia Committee Meeting** Location: Lecture Hall 3	**ILASS Europe Committee Meeting** Location: Lecture Hall 2		
2:00pm - 3:40pm	**Drop Ligament Breakup III** Location: Lecture Hall 6 Chair: **Soo-Young No** Chair: **David Sedarsky** Pages: **160 - 163**	**Modeling of Primary Breakup I** Location: Lecture Hall 7 Chair: **Christophe Dumouchel** Chair: **Takao Yoshinaga** Pages: **164 - 167**	**Automotive Gasoline I** Location: Lecture Hall 8 Chair: **Yoshio Zama** Chair: **Manuel Schuette** Pages: **168 - 171**	**Alternative Atomizers IV** Location: Lecture Hall 9 Chair: **Masataka Arai** Chair: **Choong Won Lee** Pages: **172 - 175**	**Diagnostics: Imaging Techniques IV** Location: Lecture Hall 10 Chair: **Armin Müller** Chair: **Michael Benjamin** Pages: **176 - 179**
3:40pm - 4:00pm	**Coffee Break**				
4:00pm - 5:40pm	**Drop Ligament Breakup IV** Location: Lecture Hall 6 Chair: **Takao Inamura** Chair: **Gunther Schlöffel** Pages: **180 - 183**	**Modeling of Primary Breakup II** Location: Lecture Hall 7 Chair: **Mikhael Gorokhovski** Chair: **Ilai Sher** Pages: **184 - 187**	**Automotive Gasoline II** Location: Lecture Hall 8 Chair: **Luigi Allocca** Chair: **Daisuke TSURU** Pages: **188 - 191**	**Medical Sprays** Location: Lecture Hall 9 Chair: **Corinne Lengsfeld** Chair: **Caroline L. Genzale** Pages: **192 - 195**	**Diagnostics: Imaging Techniques V** Location: Lecture Hall 10 Chair: **Gilles Bruneaux** Chair: **Kuo-Cheng Lin** Pages: **196 - 199**
5:45pm - 7:00pm	**Annual General Meeting** Location: Lecture Hall 6				

Date: Thursday, 06/Sep/2012

Pages: **201 – 246**

8:30am - 10:35am	**Aerojets & Rocket Sprays** Location: **Lecture Hall 6** Chair: **Oskar Haidn** Chair: **Eugene Lubarsky** Pages: **201 - 205**	**Process Spray** Location: **Lecture Hall 7** Chair: **Masatoshi Daikoku** Chair: **Bin Chen** Pages: **206 - 210**	**Simulation of Primary Atomization I** Location: **Lecture Hall 8** Chair: **Rolf Reitz** Chair: **Miguel Oliveira Panão** Pages: **211 - 215**	**New Developments I** Location: **Lecture Hall 9** Chair: **Qiaoyan Ye** Chair: **Reinhold Kneer** Pages: **216 - 220**	**Spray Impact III** Location: **Lecture Hall 10** Chair: **Douglas G Talley** Chair: **Bernhard Weigand** Pages: **221 - 225**
10:35am - 11:00am	**Coffee Break**				
11:00am - 12:40pm	**Cavitation** Location: **Lecture Hall 6** Chair: **Michael Wensing** Chair: **Tatsuro Wakimoto** Pages: **226 – 229**	**Flashing Sprays** Location: **Lecture Hall 7** Chair: **Christoph Johannes Hassa** Chair: **Karl-Ernst Wirth** Pages: **230 - 233**	**Simulation of Primary Atomization II** Location: **Lecture Hall 8** Chair: **Jose Manuel Pastor** Chair: **Patrick Le Clercq** Pages: **234 - 237**	**New Developments II** Location: **Lecture Hall 9** Chair: **Jeanne Malet** Chair: **Stephen Alexander Schumaker** Pages: **238 - 241**	**Spray Impact IV** Location: **Lecture Hall 10** Chair: **Siegfried Karl Ripperger** Chair: **Teruhiko Yoshida** Pages: **242 - 245**
12:40pm	**Closing**				

Monday

September 3, 2012

Contents

Pressure Atomizers I

Diesel Nozzles and Sprays I

Drop Impact I

Combustion II

Liquid Jets/Sheet Atomization II

Alternative Fuels I

Diesel Nozzles and Sprays II

Drop Impact II

Combustion III

Liquid Jets/Sheet Atomization III

Alternative Fuels II

Diesel Nozzles and Sprays III

The Significance of Fuel Preparation for Low Emissions Aero-Engine Combustion Technnology

Th. Doerr
Rolls-Royce Deutschland Ltd & Co KG, Blankenfelde-Mahlow, Germany
thomas.doerr@rolls-royce.com

Abstract

The expected annual growth rates of air traffic with about 3% for the next two decades are only sustainable if the environmental footprint of engines for aviation is minimised. Besides the progressing stringency of legislative requirements for noise and NOx emissions, customer requirements and market competition are increasingly focussing on the environmental friendliness of aero-engines. Fuel burn und thus CO2 emissions, as the predominant driver for aero-engine design optimisation over the last years has been accompanied by very stringent NOx and noise requirements. A successful new engine design needs a well-balanced consideration of these three parameters and their associated trade-offs. In contrast to previous practise recent approaches for new engine applications are characterised by more revolutionary changes of existing technologies and engine architectures.

The development of combustors for this next generation of aero-engines is predominantly driven by future NOx requirements. Until today only significant improvements of conventional but highly optimised rich burn combustors avoided excessive NOx formation due to aggravated engine cycle conditions with increased combustor inlet pressures, temperatures and significant lower air-to-fuel-ratios. Beside NOx, particulate matter (PM) is another gas turbine combustion emission, which is relevant for local air quality. Engine manufacturers are therefore faced with the need for a reliable combustor design meeting future certification requirements with sufficient margin. Rich burn combustion approach however has a limited potential to cope with future legislative and customer requirements for emission reduction while facing further demanding cycle conditions for lower fuel burn and CO2 emission levels, therefore a revolutionary step towards lean-burn combustion techniques is inevitable.

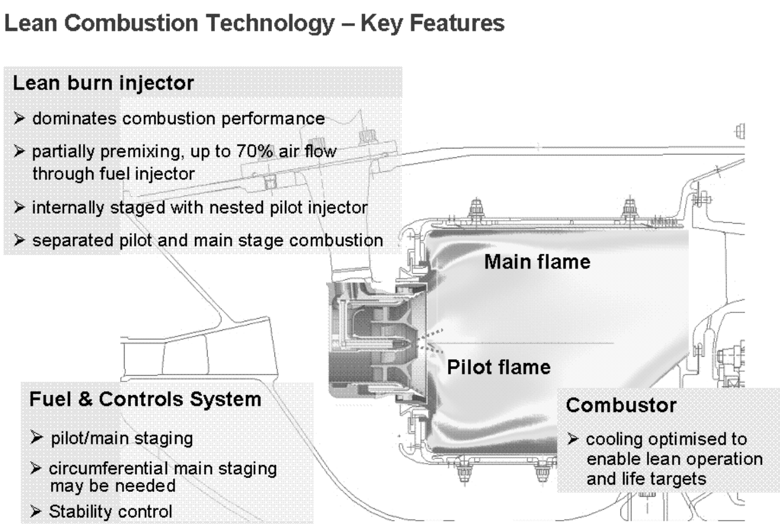


Fig. 1 Rolls-Royce Lean Burn Combustion Technology – Schematic

Lean-burn combustion is identified to offer the highest potential for NOx mitigation contributing to the overall ACARE goal of 80% reduction. Therefore, a significant amount of the air mass flow has to be utilised for fuel preparation and initiation of lean combustion. Lean-burn combustors require fuel staging to obtain full combustor operability and to enable typical aero-engine turn-down ratios while burning lean at high power conditions. The aero- and fluid-dynamic design of the fuel injector is of paramount importance for the combustor performance and a significant portion of the research and development work is focusing on this subject.

Fuel staging is achieved by an internally staged lean-burn fuel injector, which generates a homogeneous fuel-air mixture in the combustor to ensure combustion with reduced peak temperatures at medium to high power operating conditions. The fuel injector configuration features a concentric arrangement of a main fuel stage embedded into large swirling air streams carrying the biggest portion of the combustor air and a nested pilot fuel injector located in the centre.

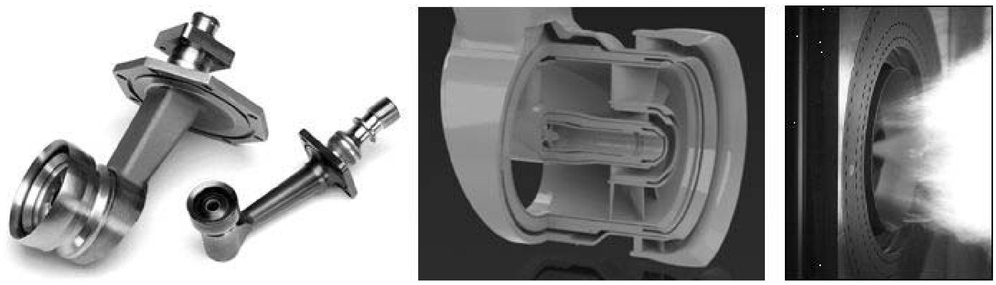


Fig. 2 Rolls-Royce Internally Staged Lean Direct Injection Fuel Spray Nozzle

Lean premixing has to be accomplished in between main fuel injection porting and main flame anchoring location downstream of the fuel injector. The main lean direct injection (LDI) can be arranged with a pre-filming air-blast concept. Within the pre-filmer fuel is distributed over a large surface area resulting in a thin fuel layer exposed to air with high velocity. As a result, the fuel sheet disintegrates into fine droplets being dispersed and evaporated downstream. The fuel rich pilot stage is required for low power operation and stabilisation of the main stage maintaining full combustor turn-down ratios for operability, especially for transient manoeuvres during adverse weather conditions such as hail and rain.

The understanding and steering of the interaction of pilot and main combustion zones is the key technology of lean burn low emissions combustion. Fuel preparation and fuel placement as well as local stoichiometries within the combustor have to be understood and optimised. Both combustion zones have to be sufficiently separated to enable low NOx combustion at reduced flame temperatures at high power conditions, whereas their interaction must be strong enough at low and mid power conditions to avoid emissions and inefficiencies caused by incomplete combustion.

The presentation will illustrate the relevant steps for the development of Rolls-Royce's low emission combustion system focussing on the significance of fuel preparation and premixing. Initially starting from numerical predictions Rolls-Royce and its partners passed through a substantial experimental development and validation programme on single sector, multi sector, full annular combustion rigs and engine demonstrators along with the development and deployment of advanced optical diagnostic methods of DLR.

Ligament and Droplet Characteristics in Prefilming Airblast Atomization

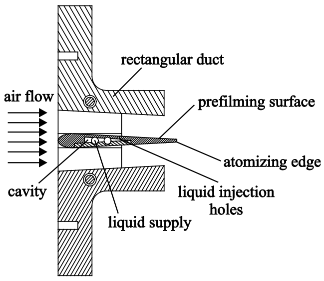
S. Gepperth[*1], A. Müller[2], R. Koch[1] and H.-J. Bauer[1]
[1]Institut für Thermische Strömungsmaschinen, Karlsruher Institut für Technologie (KIT), Karlsruhe, Germany
[2]Lechler GmbH, Metzingen, Germany
sebastian.gepperth@kit.edu and armin.mueller@lechler.de

Abstract

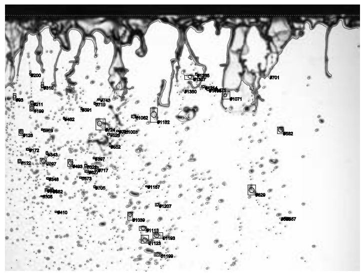
Liquid film breakup of three different geometrical variants of a planar prefilming airblast atomizer have been studied using advanced Particle Tracking Velocimetry (PTV) coupled with ligament tracking. In addition to the variation in geometry, liquids with different physical properties (surface tension, viscosity) have been tested. The liquid mass flow rate and mean air velocities were varied over a wide range of test conditions. A post-processing algorithm was developed to capture the ligament and droplet sizes and their velocities. For the test conditions investigated, it was observed that the mean air velocity and the thickness of the atomizing edge are the key parameters that influence the breakup process. From the database of test cases, correlations for the ligament breakup frequency, Sauter mean diameter and mean velocity of the primary droplets are derived.

Many previous investigations into prefilming atomizers consist of spray measurements at elevated pressures and temperatures using either actual nozzles [1] or simplified planar injectors [2]. Due to limited optical access and difficulties performing measurements in the dense spray regime, most of experimental data consists of downstream drop size measurements and are of limited value for validation of predictions of the primary breakup. To remedy this, the present work focuses on measurements on the ligament formation out of the liquid film and the primary breakup process of these ligaments into initial droplets.

Because sufficient optical access is of primary importance, a two-dimensional abstraction of an airblast atomizer was used for all experiments (Fig. 1a). All measurements were performed at ambient conditions. The mean air velocity could be adjusted between $\bar{u}_g$ = 20 to 70 m/s. The prefilming length, i.e. the distance between the injection holes and the atomizer edge is varied between 20.6 and 47.6 mm and the atomizer edge height is either 1 or 2.5 mm. The film load could be adjusted between $\dot{V}/b$ = 25 to 75 mm²/s, where b = 50 mm represents the width of the wetted surface. Six test liquids were used for the experiments in order to cover a wide range of densities, viscosities and surface tensions.



(a) Prefilmer: length is 20.6mm, atomizing edge thickness is 1mm



(b) Instant shot of ligament formation and break up
$\bar{u}_g$ = 60 m/s, $\dot{V}/b$ = 25 mm²/s, σ = 0.0255 kg/s², ν_l = 1.09x10-6 m²/s

Figure 1: experimental atomizer (a) and evaluated PTV image (b)

In order to investigate the formation of the ligaments and the breakup into single droplets, a commercially available PTV system was used for the image acquisition. For the post-processing of the data a Matlab® routine, developed by the Institut für Thermische Strömungsmaschinen (ITS) [3], was used (Fig. 1b).

Initial results indicate that the governing parameters of the breakup process are the mean air velocity and the thickness of the atomizing edge. It is found that the ligament shape and size and the Sauter mean diameter (SMD) are almost not affected by the prefilming surface length within the range investigated in these experiments. The variation in liquid properties and liquid loading also appears to have a minor effect on the primary breakup itself. The differences in characteristic ligament properties, SMD and droplet velocities of the different test cases studied are compared and the effects of flow conditions, liquid properties and geometry of the atomizers are elucidated. Correlations are derived for the breakup frequency, SMD and mean droplet velocity.

[1]Rizkalla, A. A., and Lefebvre, A. H., *Atomization Journal of Fluids Engineering* 97: 316-320 (1975)
[2]Batarseh, F. Z., Gnirß, M., Roisman, I. V., Selder, M. and Tropea, C., *Exp. in Fluids* 46: 1081-1091 (2009)
[3]Müller, A., Koch, R., Bauer, H.-J., Hehle, M. and Schäfer, O., *ASME Turbo Expo 2006*, Barcelona, Spain (2006)

* Corresponding author: sebastian.gepperth@kit.edu

Spray Characteristics of Prefilming Type of Airblast Atomizer

T. Inamura*, M. Shirota, M. Tsushima, M. Kato,
S. Hamajima and A. Sato
Faculty of Science and Technology, Hirosaki University, Hirosaki, Japan
tina@cc.hirosaki-u.ac.jp and mshirota@cc.hirosaki-u.ac.jp

Abstract

The spray characteristics of a prefilming type of airblast atomizer were experimentally investigated. Firstly, we observed the breakup phenomena at the wall edge of the liquid film flowing over the solid wall using the high-speed video movie. By the observation at the wall edge, the liquid film deforms into the bag-shaped sheet, which consists of a thin liquid film attached to a thick rim. The thin liquid film disintegrates into numerous fine droplets, and the thick ligaments attaching to the wall edge remain. Then the thick ligaments disintegrate into coarse droplets. The new numerical model of a film breakup at the wall edge was proposed. The new model assumes that the mean droplet diameter is determined by the coarse droplets generated by the breakup of a ligament attaching to the wall edge. So, concerning to the new breakup model, we focused on the ligament breakup. The numerically predicted wavelength, ligament diameter and mean droplet diameter were compared with the experiments. The predicted characteristics, such as wavelength, ligament diameter and mean droplet diameter, were almost coincident with the experiments except at low air velocity. The droplet mass flux was measured by the isokinetic probe. The droplet mass flux distributions are expressed by a normal distribution function which is normalized with a maximum flux at the center and half width at half maximum.

* Corresponding author: tina@cc.hirosaki-u.ac.jp

A Study of Gas-Centered Swirl Coaxial Injectors Using X-ray Radiography

S.A. Schumaker[1*], A.L. Kastengren[2], M.D.A. Lightfoot[1], S.A. Danczyk[1] and C.F. Powell[2]
[1]Air Force Research Laboratory, Edwards AFB, CA, USA
[2]Argonne National Laboratory, Argonne, IL, USA

Abstract

Gas-centered swirl coaxial injectors, a specific type of airblast atomizer, are of interest in rocket propulsion applications. These applications require good mixing of the liquid and gas to ensure complete combustion within the engine. While strides are being made on the computational front, predictions of the mass distributions achieved with this type of injector remain too costly or too inaccurate for engineering design. There has been, therefore, a reliance on experimental results. Unfortunately, the mass flow rates and the strong gas phase momentum typically encountered in rocket engines create sprays with high optical densities and render the vast majority of optical and laser techniques ineffective in the near-injector region. Data has been obtainable through mechanical patternation, but the technique has limitations. Time-gated ballistic imaging has also shown promise in rocket injectors but produces quantatitve data only on the size and shape of spray structures. An x-ray radiographic technique with a high-brillance x-ray source (Advanced Photon Source) has been applied to these high-optical-density sprays. To achieve this testing a new, mobile flow facility was constructed; this facility simulates the rocket flows using water and nitrogen instead of fuel and oxidizer. The x-ray radiography technique has been able to measure equilvent path length in gas-centered swirl coaxial injectors at a range of typical operating conditions. These results and their implications for gas-centered swirl coaxial injector performance in liquid rocket engines are discussed.

---

* Corresponding author: Stephen.Schumaker@edwards.af.mil, Paper # 184
Distribution A: Approved for public release; distribution unlimited. Clearance #12181

Experimental Investigation and Spray Characterization of Liquid Jet Atomization of Conventional Fuels and Liquid Bio-Fuels

Soumik Mahapatra[1*], Souvick Chatterjee[1], Swagata Shannigrahi[2], Achintya Mukhopadhyay[1], Swarnendu Sen[1]
[1]Mechanical Engineering Department, Jadavpur University, India
[2]Automobile Engineering Department, MCKV Institute of Engineering, India
hisoumik@gmail.com, souvickchat@gmail.com, swagatashannigrahi@gmail.com, a_mukho@rediffmail.com and sen.swarnendu@gmail.com

Abstract

Liquid atomization has always been a key topic of research in both academic and industrial world because of its importance from both fundamental and applied perspective. The application of efficient atomization ranges from biomedical purposes, inkjet printers and others in the micro scale to Gas Turbine and I.C. Engines in the macro world. Efficient atomization facilitating combustion is crucial for the macro-scale application and is addressed in this study. In this work, using different experimental techniques, fuel spray characteristics like spray cone angle, breakup length, droplet size and velocity distributions are studied. Simple shadowgraphy technique has been used to measure the spray cone angle. The breakup length are measured using an innovative image processing algorithm on images obtained using a laser based imaging technique. In addition, Phase Doppler Particle Analyzer (PDPA) is used to measure the droplet size and velocity. The commercially available fuel injection system used here allowed us to vary the injection pressure over a wide range. Also observed, is the effect of change in viscosity of the fuel on the breakup length and the spray cone angle. Tests were carried out for different commercially available fuels like diesel, kerosene and also bio-fuels like vegetable oils.

* Corresponding author: hisoumik@gmail.com

Application of a "Spraylet" Model to the Simulation of Fuel Spray Autoignition

Ch. Eigenbrod*, K. Klinkov, M. Reimert, P. Rickmers
Center of Applied Space Technology and Microgravity (ZARM), University of Bremen, Germany
Christian.Eigenbrod@zarm.uni-bremen.de

Abstract

Modeling of spray selfignition requires significant simplifications due to the high complexity of the physical and chemical processes involved. A turbulent vaporizing two-phase flow describes the physics of mixture formation through a spray in hot air. When modeling mixing of fuel vapor with the surrounding gas, the difference between diffusion rates of components as well as the finite penetration of the diffusion fluxes must be taken into account. The surrounding of an individual droplet or a group of droplets is characterized through significant inhomogeneity. Such inhomogeneity has a great influence on the characteristics of ignition and combustion. Therewith, a fuel oxidation mechanism defines the scores of components that are involved in a diffusive process. In spite of a significant reduction of chemical kinetics, its calculation requires a high performance of computers. Thus, sub-models of the spray simulation must describe the turbulent large scale flow as well as the detailed small scale diffusive transport including chemical reactions.

In this work a new approach for spray ignition simulation is presented. The simulation is split into two tasks. The initial formation of the spray, atomization, vaporization and turbulent two-phase flow are computed in large scale. Chemistry is not taken into account in this part. From this CFD-calculation the droplet´s trajectories and the gas parameters as temperature and mixture fraction along these trajectories are extracted. The latter serve as variable boundary conditions for the second task, the single droplet ignition simulation, named "Spraylet". The "Spraylet" model is solving the transient differential equations of the processes in the liquid and gas phases with variable physical properties. Also the concept of multi-component diffusion is applied. The effects of the presence of neighboring droplets in the flow, usually referred to as "spray" effects, is approximated by taking modifications of the conditions at the outer boundary of the computational domain into account. Thus the model considers the finite rates of diffusion and chemical reactions, as well as spray effects, and allows for spray ignition simulations to predict the most probable instants of ignition of the individually calculated droplet trajectories. The individual ignition delay times can finally be projected onto its trajectories delivering a spatial plane of the most probable instant of ignition – this independent of whether ignition happens soon (heterogeneous ignition) or later (homogeneous ignition). While the CFD part is handling spray formation, turbulence, temperature, pressure and global vaporization, the "Spraylet" calculations take care of droplet related physics and chemistry.

The "Spraylet" model is an enhancement of former single droplet ignition simulations that were validated through hundreds of microgravity droplet ignition experiments. The validation of the "Spraylet"-based simulation was performed by comparison with experimental data obtained from spray ignition experiments performed in a hot blowdown wind-tunnel. N-heptane as liquid fuel was injected into cross flow of air with a pressure of 5 bar and a temperature of 800 K.

The related projects were supported by the German Space Agency (DLR) and the European Space Agency (ESA).

* Corresponding author: Christian.Eigenbrod@zarm.uni-bremen.de

Modeling the Transient Structure of Non-Reacting and Reacting Diesel Sprays

C. Bajaj, M. Ameen, J. Abraham*
School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907-2088, USA
bajajc@purdue.edu, mameen@purdue.edu, jabraham@purdue.edu

Abstract

Modeling the transient structure of reacting diesel sprays accurately is important within the context of multidimensional modeling of flows, sprays, and combustion in diesel engines. In the case of non-reacting vaporizing sprays, vapor jets have been shown to reproduce, with adequate accuracy, the structure of the diesel jet beyond the maximum liquid length of about 100 orifice diameters in high-pressure injection into high-pressure high-temperature environments. In this work, the primary focus is on reacting diesel jets. An unsteady flamelet progress variable (UFPV) model for reacting diesel jets is evaluated. The UFPV model has been proposed for predicting the averaged/filtered chemistry source terms in Reynolds averaged simulations (RANS) and large eddy simulations of turbulent non-premixed combustion. In the model, a look-up table of reaction source terms is generated as a function of mixture fraction Z, stoichiometric scalar dissipation rate χ_{st}, and progress variable C_{st} by solving the unsteady flamelet equations. The formulation assumes that the shape of the $\chi(Z)$ profile can be modeled by an error function which remains unchanged in the presence of heat release. It is also assumed that the probability density functions (pdfs) of Z, χ_{st}, and C_{st} are statistically independent, and presumed functions are employed for the pdfs. Changes in injection pressure, ambient temperature, ambient density, orifice diameter, and ambient O_2 concentration are considered. It is shown that the model is able to predict ignition delay and flame lift-off with reasonable accuracy for all conditions. The ability of the model to predict the lift-off height appears to be related to the mechanism by which the flame propagates from the ignition location to the final stabilization plane along the stoichiometric surface.

* Corresponding author: jabraham@purdue.edu

Influence of Drop Spacing on Burning of an Emulsified-Drop Stream

Y. Y. Lin1, C. K. Chen2, T. H. Lin3*
[1,2,3]Department of Mechanical Engineering, National Cheng Kung University, Taiwan
[3]Research Center for Energy Technology and Strategy, National Cheng Kung University
eva9404@hotmail.com, a0412s@yahoo.com.tw, thlin@mail.ncku.edu.tw

Abstract

Combustion characteristics of water-in-dodecane emulsion drops with various initial spacings were studied experimentally by using a free-falling drop burning apparatus. The initial drop spacings (S_i) were 2.5, 5, 10, 40, 75 (70), 100. S_i (s/d_i) was defined as the ratio of the drop center-to-center distance (s) to the initial drop diameter (d_i). The water content (β) and the oxygen concentration (Ω_{O2}) were fixed at 5% and 21%, while two drop sizes 550 μm and 450 μm were compared. The results showed that the transition of the drop flame occurred for all cases in the experiment. For $S_i > 10$ along the flow direction, the flame around the drops would change from a blue spherical flame to a yellow flame and a wake flame, and the drop flame extinguished later in the downstream region. Soot particles was generated and drops collision and merging occurred to form a flame tube for S_i =2.5 in both cases of d_i = 550 μm and 450 μm. Besides, drop expansion was observed in both cases of d_i = 550 μm and d_i = 450 μm, while micro-explosion only occurred in the far downstream region for S_i = 40, d_i = 450 μm. It was also shown that the emulsion drop evaporation rate was not a constant, and the trend of the drop evaporation rate was strongly influenced by changing the initial drop size.

* Corresponding author: thlin@mail.ncku.edu.tw

Experimental Investigations of the Ignition and Flame Stabilization of a Full Cone Kerosene Spray in a Lab-scale Model Combustor

G.C. Gebel*, T. Mosbach, W. Meier, M. Aigner
Institute of Combustion Technology, German Aerospace Center (DLR)
70569 Stuttgart, Germany

Abstract

Spark ignition of liquid fuel sprays in air and the subsequent flame kernel growth are complex processes, subjected to a variety of mechanisms. A deep understanding is essential for the development of theoretical and numerical models, which contribute to improved efficiency and emission of gas turbines or piston engines. The specific background of this study is the altitude relight of aviation gas turbines. Modern aviation combustor technologies commonly use lean combustion systems, which enforce the challenge of altitude relight. The development of such combustors requests the ability to perform numerical simulations of the ignition process, since an iterative development, including extensive testing, cannot be performed in an economical manner. In this study, the laser-induced ignition of kerosene was investigated in a lab-scale model combustor at well-defined boundary conditions. The experiments were intended to capture the transient ignition process, starting with a flame kernel and completing with a stable flame. The model combustor was a vertically arranged flow channel, which provided good optical access to enable the application of optical measurement techniques. A commercial full cone spray nozzle from Delavan/Goodrich was placed inside the combustor. This nozzle was supplied by two mass flows: a fuel mass flow and an atomization air flow. Ignition was achieved by laser-induced breakdowns. High-speed Particle Image Velocimetry (HS-PIV) was applied to the spray to measure the velocity vectors of the fuel droplets and to verify the suitability of PIV as a tool for spray characterization. An ignition probability study was carried out to determine the ignition probability of the spray flame with respect to the sparking location and to find the most reliable location for subsequent measurements. Simultaneous Planar Laser-Induced Fluorescence (PLIF) measurements on fuel aromatics and hydroxyl radicals (OH) were performed to obtain detailed information about the fuel density distribution and location of the chemical reaction zone during the ignition and flame stabilization. During the experiments the flow channel was perfused by a steady air flow at atmospheric pressure. The standard fuel was Jet A-1 aviation kerosene. Only for the PLIF measurements Exxsol D80 was used, in order to reduce the fluorescence from the fuel aromatics. Systematic variation of the air and fuel mass flows was done during all experiments to determine their impact on the ignition process. It was found that the spray features a narrow core region, where droplet densities and velocities are very high compared to the outer regions. The atomizing air mass flow turned out to have a significant impact on the spray geometry and droplet velocities. The flame stabilization process completed after approximately twenty microseconds. Flame fronts stabilize along the spray cone edge. The results are valuable data for model development and validation.

* Corresponding author: gregor.gebel@dlr.de

Elliptical jet breakup related with the internal nozzle flow

Jung Goo Hong*†, Kun Woo Ku *, Choong Won Lee* and Byung Chul Na**

* Department of Mechanical Engineering
Kyungpook National University
1370 Sankyuk-dong Buk-gu, Daegu, Republic of Korea

**Energy System Research Center
Korea Automotive Technology Institute
303 Pungse-myeon, Dongnam-gu, Chonan-si, Republic of Korea

Abstract

An experimental study was conducted to investigate the atomization characteristics of a circular nozzle and elliptical nozzles of small diameter (0.5mm) under the high injection pressure (1MPa~9MPa). Furthermore, numerical simulations were attempted to investigate the internal flow structure in the circular and elliptical nozzles. This study showed that the disintegration characteristics of the liquid jet of elliptical nozzles were much different from those of the circular nozzle. In the case of the circular nozzle, the surface of liquid jet was much smooth near the nozzle exit under the injection pressures of this study. But, in the case of the elliptical nozzles, surface waves on liquid jet have been generated and grown with increase of the injection pressure. As a result, surface breakup was observed with the increase of injection pressure because a rough column surface caused by growth of surface wave is disintegrated to ligament as the relative velocity between the liquid jet and ambient air increases. Furthermore, the numerical simulations informed that the internal flow structure of elliptical nozzle was quite different from that of the circular nozzle. The internal flow structure of the elliptical nozzle in hydraulic flip was reattached to the orifice wall of the minor axis unlike the flow in the circular nozzle which is detached from orifice wall. It has been concluded that the internal flow structure of the elliptical nozzle has influence on the disintegration characteristics of the liquid jet issued from the elliptical nozzle.

Introduction

We carried out an experimental study on the atomization characteristics of an elliptical nozzle with a small hole diameter (0.5mm) and under relatively high injection pressure (1MPa~9MPa). From this study, it is thought that the atomization characteristics of an elliptical nozzle were related to the internal flow structure characteristics in the orifice.

Experimental setup and method

The experimental setup consists of an injection device, visualization system, pressure measurement and SMD measurement system. Working fluid was water at room temperature. The water pressurized by nitrogen in the surge tank was injected from the nozzle into ambient air. The nozzle was linked to the traverse for movement. Injection pressure was controlled by the pressure of the nitrogen gas, which was controlled by the gas regulator. Injection pressures were measured by a pressure transducer (KELLER, PA-21SR). The pressure signals were acquired by data acquisition board and monitored in real time. The spray image was obtained by the shadowgraph method using a CCD camera (Vieworks, VM-2M 35) and a xenon lamp (Drello, Drelloscope 3020) as the light source. The spray images were automatically stored by the image grabber, which was interlinked with the computer. The macroscopic characteristics of the liquid jet were analyzed from the spray images. The SMD for the variation conditions such as injection pressure, axial distance from the nozzle exit and radial distance of spray from the spray center was measured by using the laser diffraction method

Acknowledgement

This work was supported by Priority Research Centers Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2011-0018392)

†Corresponding author: jghong70@knu.ac.kr

Empirical Scaling Analysis of Atomising Annular Liquid Sheets

D. Duke*, D. Honnery, J. Soria
Laboratory for Turbulence Research in Aerospace & Combustion
Department of Mechanical & Aerospace Engineering
Monash University, Australia

Abstract

Annular liquid sheets are canonical, unstable multiphase shear flows. Many numerical, theoretical and empirical investigations of such flows have been undertaken, however agreement between studies is limited. This is due in part to the lack of agreed definitions for the gas-liquid momentum scaling and the lack of a canonical geometry, among other factors. Few scaling theories for the periodic driving instability that characterises the atomisation mechanism have yet been proposed which show repeatability between different experimental studies. There have been particularly few investigations into sheet thickness effects, due to the difficulty of manufacturing a test nozzle with a variable thickness. We present a large, parametric, experimental study of a non-swirling annular water sheet exposed to separately metered dual air co-flows over an order of magnitude variation in Reynolds and Weber Numbers. Three sheet thicknesses and a wide range of gas co-flow rates have been considered. We have considered a temporal analysis of the primary interfacial instability using Fourier techniques. From empirical data, a geometry-independent temporal scaling based on a non-dimensional momentum ratio is proposed, which shows good agreement with empirical data on low pressure air-water annular flows over a range of geometries, Re & We. This scaling is counter-intuitively different from that of the more well-known planar sheet. Through the use of Dynamic Mode Decomposition, the leading Koopman modes of the primary instability provide a spatially resolved reconstruction of the spatial growth rate and amplification profile of the instability. Sheet thickness effects are observed to dominate over all other scaling variables with regard to the amplitude of the instability, within the limits of experimental error. We demonstrate a comparison between the empirically derived complex wavenumber and linear stability analysis. The measured wavenumber approaches the linear solution as we move toward the nozzle. The temporal scaling of annular sheets may follow a predictable scaling, but the spatial scaling behaviour shows a much more complex behaviour dominated by sheet thickness effects due to both non-linearity and non-parallelism.

*Corresponding author: daniel.duke@monash.edu

Effect of orifice configuration on the penetration height in crossflow

Jun Hee Kim*, Kun Woo Ku*, Hyun Jin Youn*, Jung Goo Hong*†,
Choong Won Lee* and Kyung Yul Chung**

* Department of Mechanical Engineering
Kyungpook National University
1370 Sankyuk-dong Buk-gu, Daegu, Republic of Korea

** Environment and Energy Systems Research Division
Korea Institute of Machinery and Materials
104 Sinseongno, Yuseong-gu, Daejeon, Republic of Korea

Abstract

In previous researches on jet in crossflow (JICF), which is applied for the liquid jet injection system of air-breathing propulsion systems or rocket engine system, more than 20 different correlations of jet penetration have been proposed. But, most of these studies were carried out using the single orifice injector (SOI). In this study, in order to define the interference effects of liquid jet penetration in crossflow, the double orifice injector (DOI) is adopted. First, the jet penetration correlation of SOI according to the crossflow temperature controlled by the vitiated air heater is proposed. The jet penetration height for heated crossflow is lower than that for cold crossflow because of the increase of crossflow velocity despite the lower density. The jet penetration correlation of DOI is derived for variations of injector orifice spacing. In the case of the DOI, since the front liquid jet acts as a shield of the rear liquid jet, the jet penetration with DOI is higher than that with SOI. With the DOI, the rear jet penetration height increased as the nozzle spacing decreases. And, the penetration height correlation for the rear liquid jet with DOI was developed. Therefore, an inverse relationship between nozzle spacing and jet penetration height is expected.

Introduction

The objective of this study is to present a set of penetration heights for double liquid jets injected into crossflow and to describe the influence of the double liquid jets. Thus, the jet penetration height correlation for the single liquid jet injector and double liquid jet injector is determined.

Experimental setup and method

The experimental facility consisted of an elevated ambient pressure supply system, vitiated air heater (VAH), test section, and the pressure and flow rate measurement section. The elevated ambient pressure supply system consisted of a higher-capacity compressor, an air dryer, an air storage tank and a pressure control valve. In this study, the air conditions were set by the vitiated air heater (VAH). The test section had a height of 100mm and a width of 100mm. The visualization windows, made of tempered glass, were installed on 2 sides of the test section, and 1 on the top. In order to maintain constant quantity of crossflow, a sonic nozzle was installed in front of the test section, and to measure and control the pressure of the test section, a pressure regulating valve and an orifice were used. The pressure and flow-rate measurement section were composed of a pressure sensor, data acquisition board, and micro-manometer. Also, the liquid jet penetrations in the JICF were analyzed by the shadowgraph method using a high-speed digital camera and a stroboscope. The shadowgraphs of the liquid jet were taken with Canon EF 50mm f/1.8 lens, and were synchronized with the digital Camera (Canon EOS 20D) and the stroboscope.

Acknowledgement

This work was supported by Priority Research Centers Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology (2011-0018392) and a part of the project titled "Development of TCS System on Green Ship Technology(20110153)" funded by the Ministry of Land, Transport and Maritime Affairs, Korea.

†Corresponding author: jghong70@knu.ac.kr

Investigation of Effect of Nozzle Geometry on Spray Characteristics with a 3D Eulerian Spray Model Coupled with the Nozzle Cavitating Flow

Zhixia He*, Zhaochen Jiang, Juyan Liu, Qian Wang
School of energy and power engineering, Jiangsu university, China
zxhe@ujs.edu.cn j_zhaochen@163.com

Abstract

It is well known that near-nozzle fuel liquid jet development dominated by internal nozzle flow and related fluid-dynamic instability governs the primary break-up process of the injected fuel. The fuel primary atomization controls the mixing of fuel with surrounding oxidant gas, which is significant to achieve highly-efficient and clean combustion for diesel engines. Different nozzle geometry at the same operative conditions can lead to dramatically different behaviors in combustion performance and emissions formation. While the relation between nozzle flow and spray development in the combustion chamber is still a challenging topic with a high improvement potential. Based on the third-generation synchrotrons of Shanghai synchrotron radiation facility (SSRF), the high-precision three-dimension structure of nozzle with detailed internal geometry information were obtained, which is the basis of the subsequent numerical simulation. A 3D-Eulerian Spray multiphase model coupled with the cavitating flow inside the nozzle was put forward to simulate the whole spray process, including the primary atomization and secondary break-up. The nozzle flow and near-field spray were simulated with the VOF multiphase model. At a certain downstream location, where the spray is diluted, this Eulerian spray approach was switched to conventional Lagrangian approach. In this model, the fuel volume fraction of the cell at the switch position determined the droplet size. This entire methodology was validated through the experimental data of liquid spray penetration under non-evaporating chamber conditions. The multi-scheme numerical simulations were carried out by this model to investigate effect of the nozzle geometry and configuration on inner cavitating flow and the subsequent spray characteristics.

Key Words: diesel engine; nozzle; X-rays; cavitating flow; primary atomization; spray model

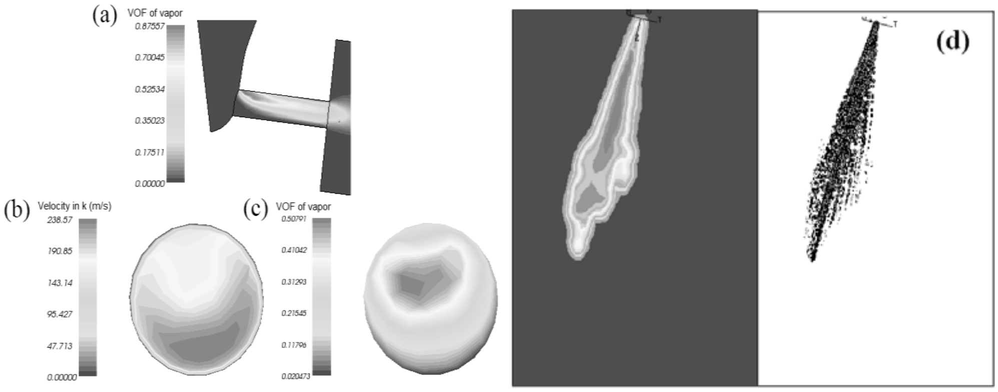


Figure Injectoin pressure 30MPa: (a) volume of fraction of fuel vapor (cross-section), (b) velocity in k direction (vertical direction of orifice outlet),(c) volume of fraction of fuel vapor (orifice outlet), (d) mass fraction of fuel and spray particle distribution.

---

*Corresponding author: zxhe@ujs.edu.cn

Viscous effects on flows through pressure-swirl atomizers

E. Wimmer, G. Brenn*
Institute of Fluid Mechanics and Heat Transfer, Graz University of Technology, Austria
brenn@fluidmech.tu-graz.ac.at and wimmer@fluidmech.tu-graz.ac.at

Abstract

Pressure-swirl atomizers are widely used in various industrial applications for their reliability in operation, and for the high liquid throughput paired with fine drop size of the sprays. Pressure-swirl atomizers are the standard atomizers in many spray-drying processes in, e.g., the chemical and food industries. An essential part of this kind of atomizers is the formation of a swirl which forces the process liquid into a rotating motion. The swirl may be imposed by an insert in the axial feed line inside the atomizer, or by tangential feed of the liquid into a swirl chamber. In both cases the liquid leaves the atomizer through a fine orifice in the form of an annular liquid sheet, which diverges in the flow direction and, therefore, assumes a conical shape with the thickness decreasing as the distance from the orifice increases.

The liquid flow rate through the atomizer depends, of course, on the driving pressure difference across the atomizer, the relevant geometric properties of the atomizer, and the liquid properties density and dynamic viscosity. One marked characteristic of the flow through pressure-swirl atomizers, which is yet unexplained, is the fact that, for a given atomizer geometry and at a given driving pressure difference, below a certain threshold liquid viscosity, the throughput of a liquid with a higher viscosity turns out higher than for a less viscous fluid. This behaviour of viscous liquid flows through swirl chambers, which is counter-intuitive at the first glance, is the subject of the present investigation.

We describe theoretically the flow through the swirl chamber, decomposing the flow field into three zones: (1) one radial outer zone at large distances from the axis of symmetry of the flow field, close to the swirl chamber wall, (2) one zone closer to the axis of symmetry, inside the radial extension of the inlet slits, and (3) the zone bounded by the gas-liquid interface at the air core. We model empirically the pressure drop across zones (1) and (2) and solve the transport equations for mass and momentum under simplifications appropriate for zone (3). The simplifications base on the negligibility of the axial velocity component in this zone. With the boundary and coupling conditions we arrive at a theoretical description of the flow field in the third zone, which allows us to relate the flow rate to the driving pressure difference for varying liquid properties and atomizer geometries. This modelling approach follows the lines of [1].

Results from the calculations quantify, among others, the liquid throughput and the air core radius. The data yield the trend observed experimentally: at moderate liquid viscosities, a liquid with a higher viscosity leads to a narrower air core, i.e., to a thicker wall-bounded film in the orifice, and, consequently, to a higher throughput than a less viscous fluid. The contribution to the conference will present the theoretical derivation in detail and present computational results in comparison with data from experiments, explaining the observed effect.

References

[1] Khavkin, Y., Strelkov, B. D., Nekhamkin, Y. E., *Teploenergetika* 10: 49-52 (1978).

* Corresponding author: brenn@fluidmech.tu-graz.ac.at

Effects of Injection Pressure on Spray Atomization Characteristics with Measurement Technique Cross-Validation

J. D. Lee[1], A. Saha[1], S. Basu[2], R Kumar[1]
[1]University of Central Florida, Orlando, USA
[2]Indian Institute of Science, Bangalore, USA
jd2lee86@gmail.com, abhishek.saha@ucf.edu, sbasu@mecheng.iisc.ernet.in and ranganathan.kumar@ucf.edu

Abstract

Fuel sprays are produced in various types of engines to form air-vapor mixtures for the combustion process. This mixture quality helps determine the combustion and emission performances. Thus, the transformation from a liquid into spray, in a gaseous medium, has significant importance in models predicting spray characteristics. The main objective of this research is to experimentally study the breakup characteristics of pressure-swirl hydraulic injector nozzles. This study consisted of two nozzles characterized as Flow number (FN) 0.4 and 1.7 respectively and injection pressures (0.3-4MPa) to investigate the effects on the atomization characteristics utilizing solely water. Using the appropriate scaling analysis the nozzles were characterized as a non-dimensional mass flow rate number ($\dot{m}^*$) and the injection pressures transformed into a pressure-Reynolds number which ranged from 7000-26000. The experimental study was conducted using three laser diagnostic techniques, Shadowgraph, PIV (Particle Image Velocimetry), and PDPA (Phase Doppler Particle Anemometry) for a complete study of the atomization process. By utilizing these techniques a cross-validation comparison between the measurements could be made. The results present the effects of non-dimensional mass flow rate number and pressure driven Reynolds number on the breakup characteristics, such as spray angle, droplet velocity, and diameter distributions axially. Utilizing the Shadowgraph technique the spray angle, film length, and breakup characteristics for each nozzle, as a function of Reynolds number, was achieved. Results indicate that both nozzles produce similar profiles for spray angle as a function of Reynolds number, increasing to a critical value at a certain Reynolds number respectively then becoming asymptotic. The PIV, Shadowgraph, and PDPA were all used in order to determine the velocity and diameter distribution throughout the spray cone. PDPA and Shadowgraph were used for diameter measurements and PDPA, Shadowgraph, and PIV for velocity measurements. The cross-validation comparison between the three techniques compared well with each other, but limitations were found. The droplets being around 10-20μm near the nozzle tip, caused this area to be extremely dense and measurements made with the PIV near this region were found to be impossible. The Shadowgraph and PDPA produced adequate results for velocity measurements at this location with the Shadowgraph producing slightly higher values. However, when diameter measurements were made similar to the PIV, the PDPA results produced low validation percentages. An optimal distance of 12.7mm from the injector nozzle was found to achieve measurement comparison between the different techniques. Results indicate that velocity decreases as a logarithmic decay axially. While the diameter decreases away from the nozzle, the droplets however, coalesce to make the effective diameter larger. The effects are functions of Reynolds number.

Removal of Scales in Petroleum Production Tubing Utilising High Pressure Atomisers

G. C. Enyi*, G. G. Nasr, A. Nourian, M.A. El-Kamkhi and M. L. Burby
Spray Research Group (SRG)
University of Salford
Manchester, U.K.
G.C.Enyi@edu.salford.ac.uk

Abstract

Scale is a mineral deposit usually formed on surfaces in contact with water. Scale deposition in petroleum production wells can be attributed to mixing of incompatible waters, changes in thermodynamic, kinetic and hydrodynamic conditions in oilfield operations.

Scale creates serious problems in producing, injection and waste disposal wells. It may restrict and completely plug off production in the formation, tubing or in flow lines. Scale prevention using chemical and mechanical methods are limited in application and depend on the type of well completion method used. This investigation proposes a new technique to address the problem and lay the foundations for a methodology for descaling in-situ production in oil and gas wells. The technique uses flat fan atomiser to produce high water pressure of 6 MPa, flow rate of 23 l/min and high impact force of 0.657 MPa which are used to dislodge scales build-up along the production tubing. Simulated laboratory scale removal rig was designed and built to demonstrate the effects of using overlapping flat fan spray atomisers to remove scales that were formed in oil and gas production tubing. This non-destructive method provides significant advantages over current scale removal methods that involve the use of chemicals or other harmful substances which are impediments to the environment and can also affect the integrity of the pipe.

Three scale samples from oil and gas wells from North Africa and one laboratory prepared candle wax scale were tested using single flat fan atomiser and a combination of two and three high pressure and high impact force atomisers. The Volume of Scale Removed (VSR) was measured experimentally using a combination of atomisers, at different spray angles, downstream distances, and water supply pressures and spraying times. The maximum quantity of scale removed using the soft candle wax was found to be 53 cm^3 at spray cone angle of 30 degree at 75 mm downstream distance from the atomisers exit. Moreover, the volume of scale removed from other three samples was found to be 11.688 cm^3 for the soft gas scale, 13.750 cm^3 for the oil wax scale and approximately 0.989 cm^3 for hard scale sample at 75 mm downstream distance.

KEYWORDS: Scale removal, Flat-Fan atomiser, Overlapping spray

* Corresponding author: G.C.Enyi@edu.salford.ac.uk

Numerical and experimental study of spray coating using air-assisted high pressure atomizers

Q. Ye[1*], B. Shen[2], O. Tiedje[1] and J. Domnick[3]
[1] Fraunhofer Institute for Manufacturing Engineering and Automation
Nobelstr. 12, 70569 Stuttgart, Germany
[2] University Stuttgart, Nobelstr. 12, 70569 Stuttgart, Germany
[3] University of Applied Sciences Esslingen, Kanalstr. 33, 73728 Esslingen, Germany
qiaoyan.ye@ipa.fraunhofer.de, bo.shen@gsame.uni-stuttgart.de,
oliver.tiedje@ipa.fraunhofer.de
and Joachim.Domnick@hs-esslingen.de

Abstract

Spray coating processes are characterized by their relatively large paint wastage and high energy consumption, especially in car industry. The unavoidable overspray, caused by poor transfer efficiency, has to be removed quickly from the paint booth, in order to preserve the painting quality on the subject surface. The therefore needed conditioned air stream results in intensive energy requirement for the disposal of the overspray. Against the background of increasing demands on energy-efficient painting processes, a research project has been launched focusing on increasing the efficiency of atomizers in painting processes. This paper summarizes some results of the current state of the research.

The air supported high pressure atomizer, also known as AirCoat or AirMix, is basically an airless atomizer with additional air flow for improved atomization and homogenization of the spray cone. Compared to the well-known air spray painting, the spray transfer efficiency of airless and air-assisted atomizers is relatively high, corresponding to a lower overspray. Therefore, air-assist atomizing is used in the present experimental and numerical investigations. Fraunhofer diffraction and Laser-Doppler Anemometry were used to measure droplet size distributions and the integral droplet velocity in spray cone. Furthermore, the film thickness distribution on a flat plat was also measured. A commercial CFD code (ANSYS-FLUENT) was used in the numerical studies. Influences of the additional air flow in air-assist gun on the atomization process, the film thickness distribution and the transfer efficiency were studied. A typical result is shown in Fig. 1-2. The experimental and simulation results delivered the necessary information for understanding the painting process using air-assisted guns and for improving the performance of the atomizer.

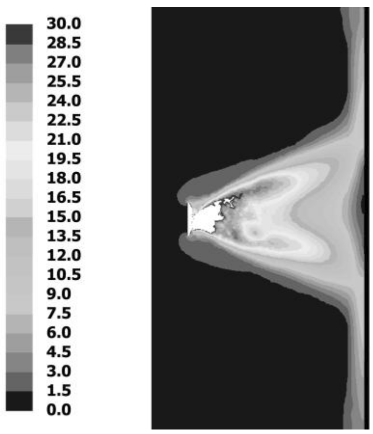


Fig. 1: Air velocity contours (m/s) in spray cone (air-assist system)

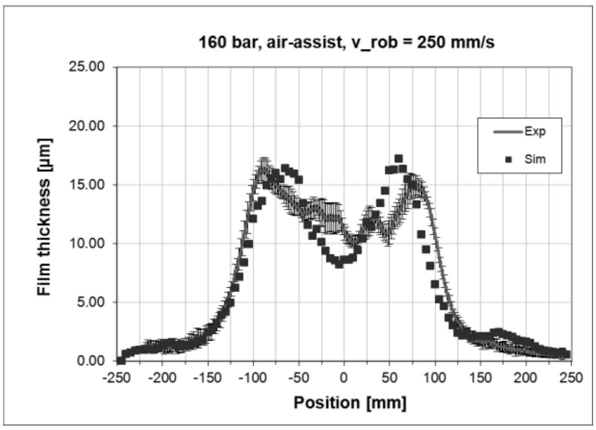


Fig. 2: Comparison of film thickness distributions (air-assist system)

* Corresponding author: qiaoyan.ye@ipa.fraunhofer.de

Improvement of Atomization and Flow Characteristics of Atomization Enhancement Nozzle for Direct Injection Diesel Engine

Nobushige TAMAKI*, Kazuya KATAKAMI
Department of Mechanical Engineering, Kinki University*, Kansai University
1, Takaya Umenobe, Higashihiroshima, Hiroshima, 739-2116, JAPAN*
tamaki@hiro.kindai.ac.jp*, k153715@kansai-u.ac.jp

Abstract

The purpose of this study is to improve atomization characteristics of a direct injection Diesel nozzle, which spray with large spray angle, short liquid core length and small droplet diameter are obtained. In the previous study, the single hole atomization enhancement nozzle, which cavitation occurs in the nozzle hole and strong disturbance is given to the liquid flow in the nozzle hole, and excellent spray characteristics are obtained at relatively low injection pressure, was developed. Moreover, it is demanded improvement of flow characteristics under low injection pressure, that is, even though excellent spray characteristics are obtained at low injection pressure, volumetric flow rate, which is obtained at high-injection pressure, has to obtain under low injection pressure. In this study, it was investigated about atomization of spray of the multi-hole atomization enhancement nozzle, which the single hole nozzle was separated to four nozzle holes (Hole number: N=4) with the same sectional area of the single hole nozzle as one of the four nozzle holes, and aimed to improve atomization characteristics and to obtain excellent spray characteristics. Effects of geometric shapes and dimensions of the multi-hole atomization enhancement nozzle, such as sharp and rounded inlet shapes and combination of hole diameter, on atomization characteristics were investigated. It was investigated that the developed atomization enhancement nozzle with the best hole inlet shape and dimension of nozzle hole was installed at the direct injection Diesel injector, effects of the multi-hole atomization enhancement nozzle on atomization of intermittent spray under high-ambient pressure condition. As a result, it was cleared that in case of the multi-hole atomization enhancement nozzle with hole number of N=4 and rounded inlet shape, breakup length becomes short and spray angle becomes large about 50 percent compared with the actual single hole nozzle (Fig.1). Moreover, volumetric flow rate is increased about 60 percent, and flow characteristic was considerably improved (Fig.2). Atomization characteristics and flow characteristics were improved under low injection pressure. Atomization characteristics of intermittent spray were improved considerably by using the multi-hole atomization enhancement nozzle with hole number of N=4 and rounded inlet shape at high-ambient pressure conditions for the direct injection Diesel engine (Fig.3). Although spray tip penetration of the multi-hole atomization enhancement nozzle is short compared with the actual single hole nozzle, spread of spray becomes wide considerably and homogeneity of spray are obtained.

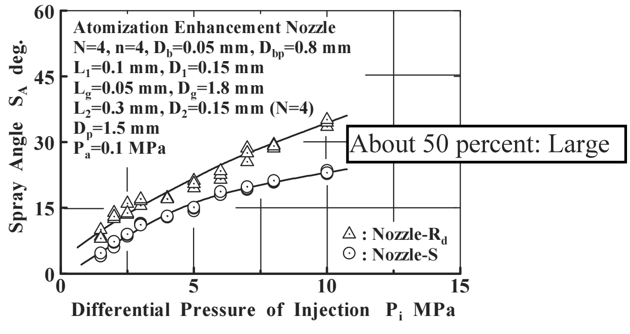


Figure 1 Effect of inlet shapes of nozzle hole on spray angle

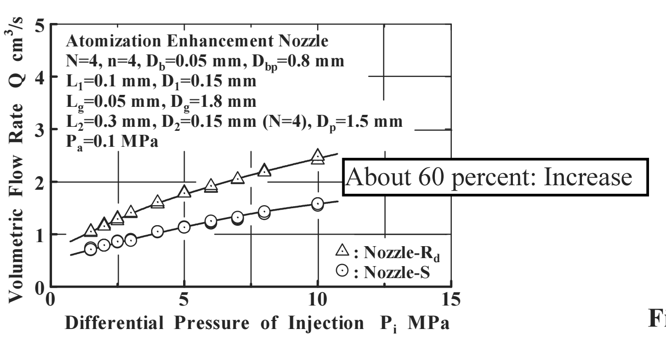
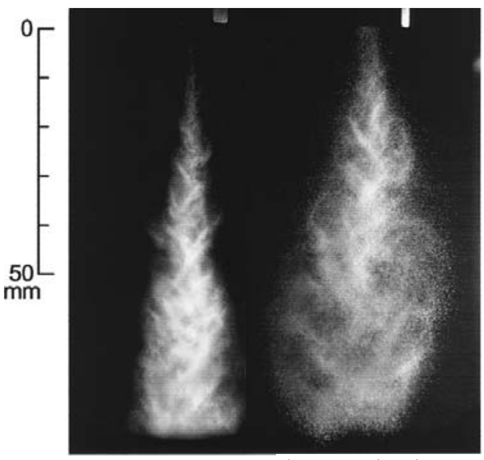


Figure 2 Effect of inlet shapes of nozzle hole on volumetric flow rate



(a) Single Hole Nozzle
(b) Atomization Enhancement Nozzle

P_i=100 MPa, P_a=1.6 MPa, T_a=300 K, $T_{inj.}$=900 us, t=3.0 ms

Figure 3 Atomization of intermittent spray under high-ambient pressure condition

Effect of gas properties on Diesel spray penetration and spreading angle for the ECN injectors

F. Payri, R. Payri*, F.J. Salvador, M. Bardi
CMT-Motores Térmicos, Universidad Politecnica de Valencia, Spain
fpayri@mot.upv.es, rpayri@mot.upv.es, fsalvado@mot.upv.es, mbardi@mot.upv.es

Abstract

The detailed knowledge of the Diesel spray formation is a key factor for the development of robust injection strategies able to reduce the pollutant emissions and keep or increase the combustion efficiency.

In this work, three similar single-hole injectors from engine Combustion Network (ECN) were compared measuring spray penetration and spreading angle under a wide range of test conditions. Several combinations of injection pressures (50 to 150 MPa) and ambient density (7.6 to 22.8 kg/m^3) have been tested in non-evaporative conditions in different gases: Nitrogen and Sulphur Hexafloride (SF_6); moreover, in Nitrogen atmosphere has been executed a temperature sweep from 300 to 550 K, while the same gas densities have been kept.

The single-hole nozzles from the ECN Working Group have been employed and n-dodecane has been used as fuel. The tests have been performed in two different test rigs: one designed for the recirculation of high density gases, like SF_6, at relatively low pressures (max 1 MPa); the other able to control both gas temperature and gas pressure over a wide range (300 – 1000 K and 0.1 – 15 MPa, respectively). Mie scattering imaging technique has been performed using a fast camera and a Matlab routine has been built for the image processing.

The experimental results pointed out some differences in spray penetration for the three injectors related mainly to start of injection transient and differences, spray orifice outlet diameter and spreading angle. A consistent effect of the type of the gas employed on spray penetration and spreading angle has been observed, while ambient temperature appears to have very small effect up 400K; above this temperature some reduction in penetration is appreciable due principally to limitation in the sensitivity of the technique when fuel evaporation is increasing. The non-complete momentum transfer between spray droplets and entrained gas as well as difference in sound speed probably are within the causes of the effect observed.

* Corresponding author: Prof. Raul Payri rpayri@mot.upv.es

Velocity measurement inside a diesel spray by using time-resolved PIV under high ambient density condition

Y. Zama*, W. Ochiai, T. Furuhata, M. Arai
Department of mechanical system engineering, Gunma University, Japan
yzama@gunma-u.ac.jp, t11801209@gunma-u.ac.jp, tfuruhata@gunma-u.ac.jp and arai@gunma-u.ac.jp

Abstract

Recently, exhaust gas recirculation (EGR) ratio and boost pressure of a diesel engine tend to increase for improvement of diesel engine performance and emission. In ultra-high boost engine, compressed gas density in combustion chamber became 3 or 4 times higher than that in natural aspirated (NA) engine. It seems that a diesel spray behavior in high gas density surroundings might differ from that in a conventional diesel engine. There are many reports regarding the effect of ambient gas density on diesel spray behavior. In high gas density surroundings, some researchers investigated spray behavior in terms of spray tip penetration and spray angle. Regarding the investigation of spray behavior and spray velocity, there were some results obtained with Laser Doppler Velocimetry (LDV) and Particle Image Velocimetry (PIV). PIV is velocity field measurement technique with high spatial resolution and is currently applied to various sprays. There are some PIV reports regarding velocity of air entrainment motion around diesel spray. However, a few investigations on the velocity measurement inside diesel spray were carried out. To understand the diesel spray behavior completely, both of instantaneous and time averaged velocity fields inside the spray should be investigated. Therefore, it is necessary to obtain not only instantaneous velocity field but also temporal change of velocity fields for the analysis of transient and average behavior of diesel spray. In this study, velocity distribution inside diesel spray was measured in various gas density surroundings and injection pressures. Maximum velocity gradient positions were evaluated using the Gaussian type distribution function fitted to velocity distribution. Further, effects of ambient gas density and injection pressure on these positions were discussed.

Experimental apparatus consisted of fuel injection system, high pressure vessel and optical system for visualization of a diesel spray. Using a single-hole diesel nozzle with a single shot injection system, diesel fuel was injected into the pressure vessel. Ar^+ ion continuous laser was utilized as a light source, and a laser-light sheet with 1mm thickness was formed with a cylindrical lens and a convex lens. Digital high-speed camera (Vision Research, Phantom V710) was used for capturing sequential images of a diesel spray. Frame rate of high speed camera was set over 0.1Mfps. Ambient gas densities ρ_a in high pressure vessel were 5.8kg/m^3, 11.6kg/m^3, 23.2kg/m^3 and 46.5kg/m^3 under the room temperature condition. Here, ambient gas density of 11.6kg/m^3 was equivalent for a compressed gas density of 3MPa at 800K. This density and temperature condition corresponded to a combustion chamber condition at injection timing of a conventional NA diesel engine. Moreover, various fuel injection pressures were adapted.

According to time-series instantaneous velocity fields inside spray, in high gas density surroundings, cluster of spray droplets near the spray periphery hardly moved, and it was re-entrained into main spray flow. These phenomena were coupled with vortex formation near the spray periphery. Mean velocity distributions inside a diesel spray at 40mm and 60mm from a nozzle were evaluated from over 1000 instantaneous velocity data analyzed with PIV, and the velocity distribution was well fitted with Gaussian function. According to mean axial velocity distribution normalized with center velocity, it was clear that the position of intense mixing zone at Z = 40mm shifted from the periphery to the inside of spray with increasing ambient gas density and injection pressure. However, its trend did not appear for normalized axial velocity distribution at Z=60mm.

* Corresponding author: yzama@gunma-u.ac.jp

Engine Combustion Network: "Spray A" basic measurements and advanced diagnostics

M. Meijer*[1], L-M. Malbec[2], G. Bruneaux[2], L.M.T. Somers[1]
Eindhoven University of Technology Netherlands[1], IFPEN France[2]
m.meijer@tue.nl, louis-marie.malbec@ifpen.fr

Abstract

Diesel spray experimentation at controlled high-temperature and high-pressure conditions is intended to provide a more fundamental understanding of diesel combustion than can be achieved in engine experiments. The Engine Combustion Network (http://www.sandia.gov/ecn) has recently become a leading group in performing comparative studies under standardized conditions by using constant-volume pre-combustion vessels and constant flow test rigs. The purpose of this collaborative effort is to generate a high-quality dataset to be used for advanced computational model development under relevant modern diesel engine conditions.

In this study the pre-burn combustion vessel at IFPEN is used to analyze "spray A" conditions (n-dodecane, 900 K, 22.8 kg/m^3, 15% oxygen). The fuel spray is characterized by applying several advanced optical diagnostic techniques. However before any spray related measurements are executed, boundary conditions inside the vessel are analyzed in great detail. The ambient gas temperature distribution before the fuel injection is measured using fast response thermocouples. The fuel temperature inside the injector is measured and then stabilized by implementing several hardware upgrades. Particle Image Velocimetry (PIV) measurements are taken in order to define ambient gas velocities inside the vessel right before fuel injection.

The spray penetration length over time is analyzed using high-speed Schlieren diagnostics. An optimized backlight illumination set-up is implemented as a replacement of the previously used Mie scattering technique. PIV measurements for reacting and non-reacting sprays are carried out to analyze the fuel spray velocities during an injection event. High-speed imaging of light luminosity and the implementation of various optical filters provides information to define the ignition delay time, ignition location and Flame Lift Off (FLO) locations.

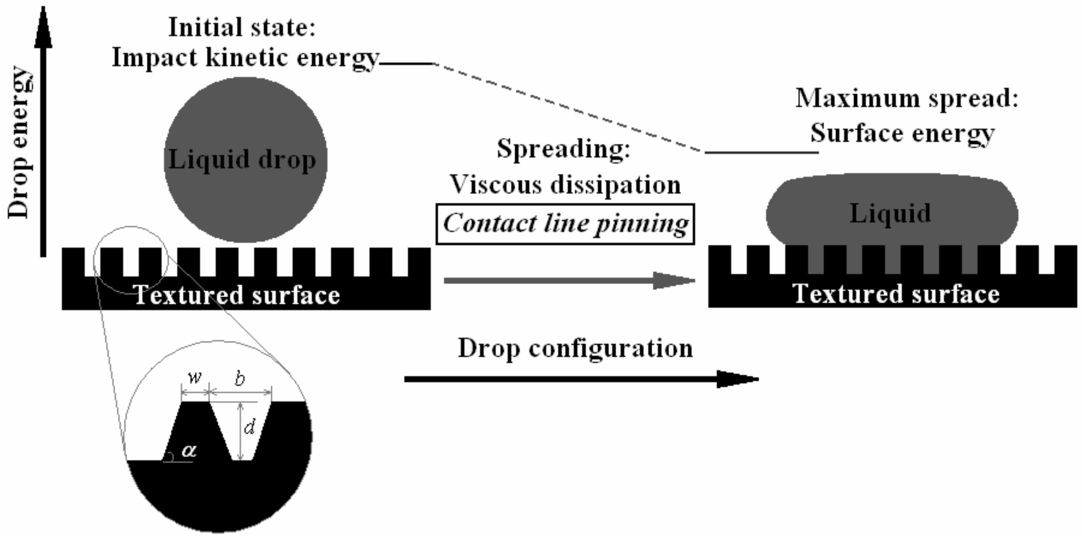
* Corresponding author: m.meijer@tue.nl

Effect of Contact Line Pinning on Maximum Spreading of Liquid Drop Impacted onto Groove-Textured Surfaces

V. Vaikuntanathan*, D. Sivakumar
Department of Aerospace Engineering, Indian Institute of Science, Bangalore 560 012, INDIA
visakh@aero.iisc.ernet.in and dskumar@aero.iisc.ernet.in

Abstract

The present work attempts to develop an understanding on the role of contact line pinning on roughness asperities of target surfaces in the maximum spreading diameter of an impacting liquid drop. Model rough (textured) surfaces of stainless steel material comprising unidirectional parallel grooves were used as the target surfaces. The impacted drop spreading in the direction perpendicular to the grooves experiences the pinning of its contact line as the drop liquid advances over the asperity posts. The highlight of the current study is the modeling of this contact line pinning as an energy loss parameter in terms of the contact angle hysteresis of liquid drop on the target surfaces, and its inclusion in the energy conservation based theoretical models of maximum spreading diameter. Experimental measurements of maximum spreading diameter of impacting drops of water and ethanol-water mixture with varying Weber number (~ 2 to 80) were collected on three groove-textured surfaces with different groove depths and pillar angles. The model predictions clearly capture the experimental observation of a reduction in maximum drop spreading in the direction perpendicular to the grooves compared to that on smooth surface and its trend with *We* for all the drop impact cases studied here. A comparison of the model predictions of maximum spreading diameter of impacting liquid drop perpendicular to the grooves with the experimental data yielded satisfactory quantitative agreement with a maximum error of ~14%. Further, in the limiting case of zero roughness, the model reduces to the one for maximum drop spreading during drop impact on smooth surfaces reported by Ukiwe and Kwok.



---

* Corresponding author: visakh@aero.iisc.ernet.in

Effects of drop and film viscosity on drop impacts onto thin films

D. Banks, C. Ajawara, G. Aguilar*
Department of Mechanical Engineering, University of California Riverside, United States
dbanks@engr.ucr.edu, cajawara@engr.ucr.edu, gaguilar@engr.ucr.edu

Abstract

While drop-film impacts have been studied extensively in the past, little thought has been given to separating the effects of the drop fluid properties and those of the film. Notably in the field of pool fire suppression, sprays impact pools consisting of different fluids, and both the spray properties and the pool properties will govern the resulting behavior [1, 2]. Distinguishing between the film and droplet's effects will also provide insight into behaviors observed on more common same-liquid drop-film impacts.

In this study, the central parameter examined is the fluid viscosity. Using various mixtures of water and glycerol, as well as FC-72, a range of kinematic viscosity covering 3 orders of magnitude (3.8E-7 to 1.1E-4 m^2/s) is examined, while the surface tensions cover a smaller range, from 0.01 to 0.07 N·m. A microliter valve creates mm-scale drops that freefall into the target film at velocities of 0.5 to 3.5 m/s, where a high speed video camera records the impact. The final parameter considered in this study is the ratio of pool depth to droplet diameter, which is examined in three regimes: 0.5 (shallow film), 1.0, and 3.0 (deep pool). This configuration covers Weber numbers from 20 – 3000 and Reynolds from 20 – 14000.

The impact outcomes characterized are coalescence, crown formation, jetting, and splashing of the crown and the jet. These regimes have been defined by past work, and for the same film-droplet liquids they have been mapped to ranges of dimensionless numbers, primarily the Weber number. These dimensionless ranges are examined for differing liquid impacts. To correct for differing drop and film properties, alternative dimensionless criteria are suggested. These criteria are proposed in light of substrate effects in the shallow film regime as well as in the deep pool regime where bottom effects are lessened.

Crown formation and subsequent crown splashing are shown to be governed by the viscosity and surface tension of the film, with minimal dependence on drop fluid properties. With a dimensionless pool depth (H* = pool depth/drop diameter) of 1, for every fluid examined, the critical Weber number for crown formation is on the order of 100 when pool density and surface tension are used. Applying the drop density and surface tension gives no apparent correlation between common dimensionless groups and observed behavior. In contrast, when characterizing the appearance of a post-impact Worthington jet, the drop's properties play a much larger role, with a critical minimum Weber number utilizing drop surface tension and density between 50 and 200 resulting in the appearance of a jet. The impact outcomes in other pool depth regimes are explored, as well.

References

1. Rein, M., *Phenomena of liquid drop impact on solid and liquid surfaces.* Fluid Dynamics Research, 1993. **12**: p. 61-93.
2. Fedorchenko, A.I. and A.-B. Wang, *On some common features of drop impact on liquid surfaces.* Physics of Fluids, 2004. **16**(5): p. 1349-1365.

Corresponding author: gaguilar@engr.ucr.edu

Experimental Measurements of a Liquid Droplet Impinging on a Corrugated Cardboard Surface

Xiangyang Zhou
FM Global, 1151 Boston-Providence Turnpike, Norwood, MA, USA
xiangyang.zhou@fmglobal.com

Abstract

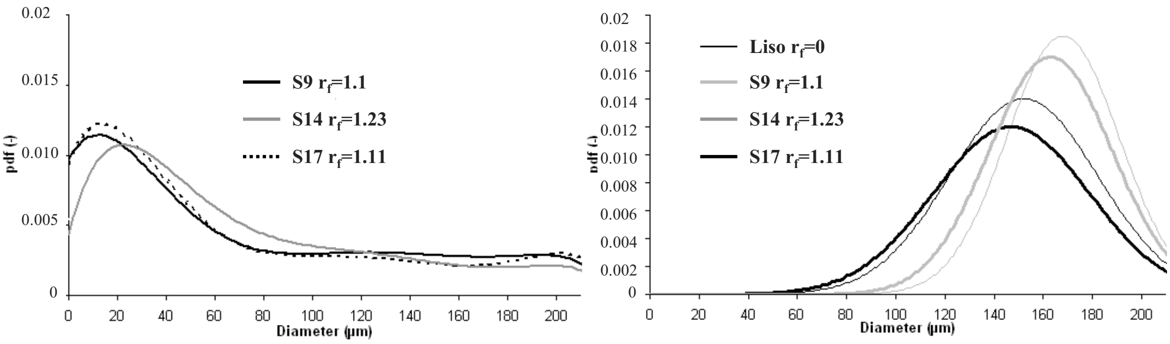
In order to provide data suitable to validate a water-based fire protection model, measurements were conducted to study the impingement process of a liquid droplet upon a corrugated cardboard surface. The amount of mass converted to secondary droplets due to splashing was measured by a weighing balance. The size, velocity and number density of the splashed droplets was measured by a laser-based Shadow-Imaging system. The impact surface was dry unheated cardboard of the kind that is widely used in corrugated boxes and shipping containers. Two liquids, with different surface tension, three droplet sizes, three impact angles and different impact velocities were investigated. Based on the measurements, a critical Weber number was introduced to determine the transition from deposition to splashing. The splashing fraction was expressed as a function of Weber number and impact angle. The distribution of cumulative volume fraction of splash droplets versus a normalized droplet size was correlated as a Rosin-Rammler function. The volume-median droplet diameter was expressed as a function of Weber number and impact angle. The maximum velocity magnitude of a splash droplet was correlated to its diameter and Weber number. These empirical correlations can be incorporated into a numerical model to simulate the splashing fraction, size and velocity distributions of the splashed droplet.

From single droplet impact to micrometric droplet chains: scaling the effect of surface topography

A. S. Moita*, A. L. N. Moreira
IN+, Instituto Superior Técnico, TU Lisbon, Portugal
anamoita@dem.ist.utl.pt and moreira@dem.ist.utl.pt

Abstract

Droplet/wall interactions have been widely explored within the last decade to develop smart interfaces and enhanced surfaces to obtain an accurate control of heat and mass transfer, which is vital, for instance for cooling applications (*e.g.* [1]). Most of the studies reported in the literature address the wetting and spreading behaviour of millimetric droplets, in a trial and error approach to infer on the best wetting and/or topographical characteristics providing the desired control of the spreading behaviour and of the heat transfer mechanisms. The present work follows the experiments reported in Moita *et al.* [2, 3] where criteria have been proposed to develop a systematic methodology to design micro-structured surfaces able to increase the rate of heat removed by single droplets impacting onto hot surfaces. The experiments reported here start with millimetric droplets ($2.5<D_0<3.3$mm) and follow an approach to scale down the patterns to micrometric droplet applications ($80\mu m<D_0<340\mu m$). The surfaces are made from silicon wafers and are micro-patterned with square pillars; the boundary conditions at impact range from dry to wet and the impact velocity, U_0, from 0.88 to 7m/s. The thermal behaviour is inferred from the local heat fluxes, determined from instantaneous temperature measurements made at the droplet-surface interface with fast response thermocouples. These are related to the way that the micro patterns affect the spreading diameter. One of the side effects of modifying the surface topography is to trigger secondary atomization. This is particularly relevant in micro-droplet stream applications and must be well controlled. Hence, the experiments make use of a Phase Doppler interferometer combined with a high speed camera to characterize the secondary atomization generated at droplet impact. The analysis allows to find that, in the homogeneous wetting regime, the parameter $r_f=(2l+\lambda_R)^2/[(2l+\lambda_R)^2+8lh]$ provides the best criterion to optimize the design of the micro pattern when, both, millimetric and micrometric droplets are considered. Here l, λ_R and h stand for the side of the square area, height of the pillars and pitch (distance between them) respectively. Within the film boiling regime, the effect of the topography is clearly more evident. However, with the current surfaces the experiments are on the limits of applicability of Wenzel's theory, so a further scaling down is required. Analysis of the size probability distribution of secondary droplets generated at the impact (Figures below), suggest that large surface patterns act to promote dynamic disintegration which overcomes the thermal induced one, so that the rougher surface leads in fact to smaller secondary droplets.



a) b)

PDF of the secondary droplet size resulting from the impact of a monosize stream of ethanol droplets (D_0=180µm, U_0=3.2 m/s) over the micro-textured surfaces (Liso to S17) within: **a)** nucleate boiling regime ($T_{w,0}$=130°C) **b)** film boiling regime ($T_{W,0}$ =200°C). PDA measurements performed at r=1.2mm, z=1.2mm.

References

[1] Weickgenannt, C. M., Roisman, I. V., Tropea, C., Yarin, A. L., *24th ILASS-Europe, Lisbon, Portugal* (2011).
[2] Moita, A. S., Moreira, Al. L. N., *Exp. Fluids*, 52:679:695 (2012).
[3] Moita, A.S., Tatta, M., Moreira, A. L. N., *24th ILASS-Europe, Lisbon, Portugal* (2011).

* Corresponding author: anamoita@dem.ist.utl.pt

Laser-induced ignition and subsequent flame development along five co-planar monodisperse fuel droplet streams

T. Mosbach*, G.C. Gebel and W. Meier
Institute of Combustion Technology, German Aerospace Center (DLR), Stuttgart, Germany
thomas.mosbach@dlr.de and gregor.gebel@dlr.de

Abstract

The ignition of fuel sprays and the subsequent development of the initial flame are subjected to various mechanisms. A deep understanding of the relevant processes is essential for the development and validation of numerical models of the flame initiation process. This study reports about experiments performed at a generic lab-scale combustor under well-defined boundary conditions contributing to the today's knowledge base of the spray ignition process. The main components of the generic combustor are a flow channel and a vibrating orifice monodisperse fuel droplet generator. The flow channel provides good optical access to the ignition and combustion region. The droplet generator is injecting five monodisperse fuel droplet streams in direction of the air co-flow.

Measurements were conducted at atmospheric pressure for different fuels and test-rig operating conditions, e.g. fuel and air mass flow rates. The following fuels were investigated: Jet A-1 kerosene, a binary mixture of 80 vol.-% n-decane and 20 vol.-% n-propylbenzene, n-heptane, ethanol, and Exxsol D80. The fuel/air two-phase flow was ignited by focused laser radiation to provide well-defined ignition conditions. Ignition was only possible by focusing the laser radiation into the droplet stream. A focal point outside the region of the droplet streams resulted in an almost zero ignition probability. This suggests that the amount of vaporized fuel was below the ignition limit. The lab-scale experiment involved a simplified geometry; the measurements were performed at atmospheric pressure. However, it enabled us to investigate the development of the laser-induced plasma, the subsequent flame kernel formation and the growth and propagation of the initial flame in a two-phase flow. Optical and laser-based measurement techniques were applied to capture the transient flame initiation process with high temporal and spatial resolution. High-speed imaging of the broadband luminosity was used to capture the growth and propagation of the initial flame following the laser-induced flame kernel. The kerosene fuel and hydroxyl radical (OH) density distribution were simultaneously measured using the planar laser-induced fluorescence (PLIF) technique. In general, with this technique the influence of the fuel placement and the influence of the position and shape of the reaction zones and regions of high temperature on the flame initiation process can be studied. The phenomena related to the flame kernel formation process were investigated by high-speed imaging and optical emission spectroscopy techniques. Particle image velocimetry (PIV) measurements were performed to characterize the air co-flow. Broadband luminosity imaging measurements were performed to track the axial-position of the lower edge of the initial flame. Differences in the growth and propagation of the initial flame were observed for identical ignition and combustor operating conditions revealing a stochastic process. It was found that the lower edge velocity of the initial flame increases with increasing fuel mass flow. The PLIF measurements of the fuel and OH density distributions evidence a diffusion flame propagating along the fuel-droplet streams. The flame propagates downstream with a velocity lower than the droplet velocity. Fuel droplets can enter and traverse the flame fronts of the initial flame. A successful ignition in our specific experimental configuration seems to depend not only on the initial heat dissipation, e.g. the thermal area loading, and radical and vapor formation but also on the droplet velocity, e.g. the droplet residence time in the flame and the strength of the induced shear flow layers around the droplet streams. The position of the area influenced by the ignition process in the droplet streams is very reproducible; it is conserved in the flow and can be tracked. The initial droplet velocity at the injector head was derived from the slope of the trajectory of the lower edge position of the influenced area. From the averaged OH PLIF images the centroid of the OH density distribution was determined. The PIV measurement results provide quantitative information about the actual flow-field at the different combustor operating conditions without fuel injection.

This study revealed several findings about the complex processes during the flame initiation. The results provide a valuable data set for the development and validation of numerical models of spray ignition.

* Corresponding author: thomas.mosbach@dlr.de

Numerical and Experimental Study of Reduction of NOx on Diesel Combustion by Using Water Injection Systems

D. TSURU*, H. KATO, H. TAJIMA
Interdisciplinary Graduate School of Engineering Sciences, Kyushu university, JAPAN
dtsuru@ence.kyush-u.ac.jp

Abstract

Marine diesel engine has been demanded dramatically to reduce harmful regulated pollutants, which are Particulate Matter (PM) and Nitrogen Oxide (NOx). In particular, NOx regulations for marine engines will require NOx reduction as much as 80% from the level in the first emission standard from IMO. Introduction of water into the combustion chamber has been recognized as an effective measure reducing NOx emissions from medium-speed marine diesels. There are two practical methods of water injection into the cylinder, FWE (Fuel Water Emulsion) and DWI (Direct Water Injection). The former method is to replace the diesel fuel with diesel/water emulsions as an alternative fuel. And the latter method is to inject the water directly into the combustion chamber separately from the fuel with two injectors or a single injector. However, optimization of these water injection systems has not been established yet. In this study, to observe spray propagation and combustion process of fuel-water emulsion and DWI system, experiments were carried out using a Rapid Compression and Expansion Machine (RCEM) with an electronic controlled double-needle type injector. This RCEM has a bore of 240 mm and a stroke of 260 mm. Combustion chamber is cubic shaped volume consisted of 200 x 66 x 80 mm and it has three quartz windows on the walls for objective of visualization. The pressure and the temperature of this combustion chamber at the injection were set to 10 MPa and 750 K, respectively. The combustion processes were taken by photographs directly using high speed camera (Photoron: Fastcam SA-4). Spatial resolution of photographs is 832 x 224 pixels and frame rate is 20,000 fps and exposure time is 5.8μs. In addition, temperature distribution of diesel flame was obtained from photographs analysed by Two Colour Method (TCM). After combustion, CO, CO2, O2, THC and NOx in the burned gas were measured by using exhaust gas analyser (HORIBA: MEXA-7100). Simulation of spray propagation and combustion processes were also carried out using three-dimensional CFD code: KIVA3V in order to evaluate the effect of water vapour distribution on cylinder temperature and NOx formation. In the DWI system, numerous and immiscible droplet collisions should happen inside the merging sprays from closely-located injection holes for water and fuel. Authors had developed the new collision models for simulating the complicated colliding behaviour of these immiscible droplets and implemented into KIVA3V code. Concentric water-in-oil type droplet had to be newly introduced as an outcome of oil and water droplet coalescence. And different treatments in calculating its breakup and evaporation processes were considered. Concentric water-in-oil type droplets were injected as an emulsion fuel in case of emulsion system. It has been concluded that when the water was injected appropriately, the temperature of flame became lower than in conventional diesel combustion. NOx emissions were reduced up to 40% with water addition of 25% of fuel mass in both the water injection technologies. The computational results were in agreement with experimental measurements and provided detailed information on the mixing process.

Key words: diesel sprays, modeling, emulsion, direct water injection, marine diesel engine

* Corresponding author: dtsuru@ence.kyushu-u.ac.jp

Experiments on turbulent ethanol spray flames in EEC conditions

H. Rodrigues*, M. J. Tummers, D. J. M. Roekaerts
Department of Multi-scale Physics, Delft University of Technology, The Netherlands
H.R.Correiarodrigues@tudelft.nl and m.j.tummers@tudelft.nl

Abstract

An experimental study on the structure of ethanol turbulent spray flames in Excess Enthalpy Combustion (EEC) conditions and cold coflow was conducted Delft Spray in Hot Coflow burner. The setup consists in a spray issuing upwards in either a coflow of ambient air or hot-vitiated combustion products mixture. Different flame structures were observed depending on the coflow Reynolds number, injection pressure and coflow temperature and oxygen volume fraction. For the spray flame in cold air, a double flame structure that consists of two diverging flame fronts originating at the leading edge of the reaction zone and an increase of the lift-off height is observed with increasing injection pressure. In EEC conditions double flame structure is absent, flame exhibits low chemiluminescence and no lift-off height changes are attested with increasing injection pressure. Using Laser Doppler anemometry and Phase Doppler anemometry, we report space and time resolved measurements of the turbulent flow field and droplet spatial spectrum for a number of measurement locations in flames. Droplet radial distributions revealed important difference in the spray structure and the evidence of non-reacting droplet along the spray edges for both cases. The modeling of the flame structures observed represents a challenge for combustion scientists due to the richness of observed physics phenomena.

---

*Corresponding author: H.R.Correiarodrigues@tudelft.nl

Effect of Disturbance of Inlet Spray Velocity on Flame Structure

T. Kitano, R. Kurose*, S. Komori
Department of Mechanical Engineering and Science, and Advanced Research Institute of Fluid Science and Engineering, Kyoto University, Yoshida-honmachi, Sakyo-ku, Kyoto, Kyoto 606-8501, Japan
kitano.tomoaki.88x@st.kyoto-u.ac.jp, kurose@mech.kyoto-u.ac.jp, and komori@mech.kyoto-u.ac.jp

Abstract

From the viewpoint of environmental protection and energy security, it is extremely important to reduce CO_2 emitted by the consumption of fossil fuels in internal combustion engines such as gas turbine engine and diesel engine for energy conversion and propulsion devises. In order to optimally design and operate such equipments, precise prediction of the combustion behavior is essential. However, since combustion is a complex phenomenon, the prediction of the combusting flow behavior has been based on the engineers' experiences and the reliable prediction technique has not been well developed yet. In particular, spray combustion is a complex phenomenon in which liquid fuel is used and the dispersion of the liquid fuel droplets, their evaporation, and the chemical reaction of the fuel vapor with the oxidizer take place interactively at the same time. Hence, the underlying physics governing these processes has not been well understood [1],[2],[3],[4]. One of the most important issues of combustion research is the prediction and suppression of combustion instabilities. In spite of a large number of studies, however, the mechanism of the combustion instabilities has not been well clarified yet e.g.[5]. In particular, the mechanism of combustion instabilities for spray combustion is hardly understood. Recently, de la Cruz Garci'a et al. [6] investigated the self-excited oscillation in a kerosene spray flame by experiment and found that frequency of the self-excited oscillation depends on the degree of mixing of air and fuel droplets. However, the trigger of the self-excited oscillation is not clear.

In tis study effects of disturbances of inlet flow and spray velocities and internal pressure on spray combustion field are investigated by means of two-dimensional direct numerical simulation (DNS). n-decane ($C_{10}H_{22}$) is used as liquid spray fuel, and evaporating droplets' motions are tracked by a Lagrangian manner. The pressure perturbation is captured by employing a pressure-based semi-implicit algorithm for compressible flows [7]. The frequency and magnitude of the inlet disturbance are set at 800 Hz and up to 50 %, respectively, and the internal pressure is set at 0.1 or 0.5 MPa.

The main results obtained in this study are summarized as follows. Firstly, the pressure perturbation in the spray combustion filed is enhanced by combustion reaction and increase in internal pressure, but it is not affected by disturbances of inlet flow and spray velocities very much. Secondly the frequency indicating the peak of power spectra of pressure perturbation depends on neither the inlet disturbance nor the internal pressure.

References

[1] Nakamura, M., Akamatsu, F., Kurose, R., Katsuki, M., *Physics of Fluids* 17:123301 (2005).
[2] Watanabe, H., Kurose, R., Hwang, S., Akamatsu, F., *Combustion and Flame* 148:234-248 (2007).
[3] Watanabe, H., Kurose, R., Komori, S., Pitsch, H., *Combustion and Flame* 152:2-13 (2008).
[4] Baba, Y., Kurose, R., *Journal of Fluid Mechanics* 612:45-79 (2008).
[5] Lu,F.K., *Progress in Astronautics and Aeronautics* 210 (2005).
[6] de la Cruz García, M., Mastorakos, E., Dowling, A.P., *Combustion and Flame* 156:374-384 (2009).
[7] Moureau, V., Berat, C., Pitsch, H., *Journal of Computational Physics* 226:1256-1270 (2007). Kee, R.J., Rupley, F.M., Miller, J.A., *Sandia Report* SAND89-8009B (1989).

*Corresponding author: kurose@mech.kyoto-u.ac.jp

Experimental study of a horizontal shear-driven liquid film approaching a sharp corner. Critical conditions.

E. Bacharoudis*[12], H. Bratec[3], L. Keirsbulck[1], J-M. Buchlin[2], L. Labraga[1]
1: Univ. Lille Nord de France, 59000 Lille, France
UVHC, TEMPO, 59313 Valenciennes, France
2: von Karman Institute for Fluid Dynamics, Rhode-St-Genése, Belgium
3: Robert Bosch Produktie N.V., Tienen, Belgium
bacharoudis@vki.ac.be, herve.bratec@be.bosch.com,
laurent.keirsbulck@univ-valenciennes.fr, buchlin@vki.ac.be and
larbi.labraga@univ-valenciennes.fr

Abstract

The present work concerns the experimental study of a horizontal liquid film (order of 1-2 mm thickness) driven by an external turbulent air flow and its separation due to the presence of a convex sharp corner. The liquid film is distilled water ($\sigma = 72$ mN/m) flowing over a polycarbonate surface with critical surface energy of $\sigma_c = 31$ mN/m which implies conditions of the partial wetting regime. The liquid film is formed free on the wall surface without side wall borders which would restrict the expansion towards the spanwise direction. Besides the characterization of the liquid film, main objective of this study is the experimental investigation of the critical conditions dictating the onset of the atomization phenomena at the edge of the corner.

The experimental facility designed and constructed for the purpose of the film separation study is consisted of three main parts, namely, the wind tunnel, the liquid supply system and the test section. The test section is a semi-open configuration of a corner with an angle of 90° made by polycarbonate. Qualitative results of the film behavior have been extracted from high speed recording. Planar laser induced fluorescence (PLIF) has been used in order to visualize the cross section of the liquid film in the spanwise direction at the edge of the corner. A Matlab code has been developed to process the images and collect information for the interface of the liquid film. The mean film thickness at the edge of the corner has been obtained. The mean width of the liquid film at the edge of the corner has been measured from single images taken by a digital SLR camera and the mean film velocity has been calculated by measuring the flow rate of the liquid film. The droplets generated downstream the corner have been collected with a flat collector and weighed by means of a digital balance with accuracy of 1 mg. Critical conditions are considered the flow conditions on which the liquid film begins to be atomized at the sharp corner and the resulting atomized mass presents an intermittent layout.

The liquid film formed on the horizontal flat plate and the mass of the droplets produced after the corner have been studied for different flow conditions. The experiments show that there is a weak dependency of the film width on time. Depending on the flow conditions, the liquid film depicts two different trends. Far above the critical conditions, the film is atomized continuously but its rate is slowly reduced due to the slow variation (expansion) of the film width, while at the critical conditions, the atomization presents an intermittent behavior and the effect of the width is vanished.

The main objective of this project is the identification of the critical conditions for the onset of the liquid film atomization at the corner. Measurements of the local characteristics of the shear-driven liquid film have been conducted under critical conditions and presented here. Generally, the critical mean film thickness and the critical mean film velocity drop as the external air flow is increased and the liquid flow rate is reduced. The dimensionless RMS of the film thickness which comprises a mean to measure the interface fluctuations seems to be reduced. The role of the film width on the onset of the film atomization has been investigated by developing and comparing two different films coming from slits with different lengths ($w_{slit} = 100mm$ and $w_{slit} = 45mm$). The results show that the onset of the film breakup depends primarily on the wavy interface but it is independent on the film width. The mean local characteristics of the film have been correlated using two dimensionless numbers, the film Reynolds number (based on the mean film thickness and velocity) and the aerodynamic Weber number (based on the velocity difference of the external air flow with the film and the mean film thickness). An empirical formula has been derived from the experimental data. A first comparison of the experimental data under critical conditions with proposed analytical models in literature showed that further theoretical development is required to predict accurately the onset of the critical conditions.

*Corresponding author: bacharoudis@vki.ac.be

Airblast spray characteristics of planar liquid films in longitudinal gas-phase shear layers at various ambient pressure conditions

K. Yoshida[1], K. Ide[1], S. Takahashi[2], K. Matsuura[*3], J. Iino[4], Y. Kurosawa[3], S. Hayashi[2] and Y. Ohta[1]

1: Department of Mechanical Engineering, Waseda University, Japan
2: Department of Mechanical Engineering, Hosei University, Japan
3: Aviation Program Group, Japan Aerospace Exploration Agency, Japan
4: Advanced Science & Intelligence Research Institute Corp., Japan

keisuke-yoshida@asagi.waseda.jp and matsuura.kazuaki@jaxa.jp

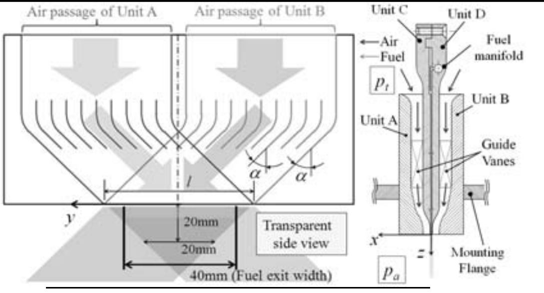
Abstract

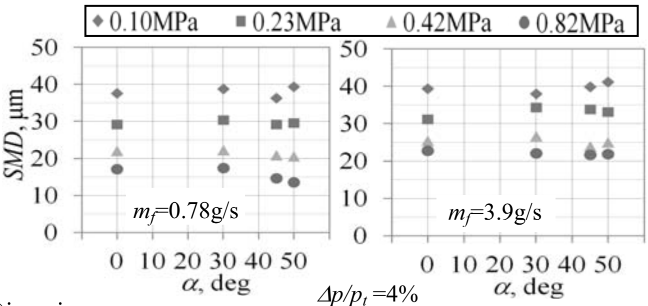
Recently, the present authors investigated the effects of swirl combinations on the spray characteristics of a double-swirl filming-type airblast atomizer in relation to aero-engine applications. For a normalized air pressure drop across the atomizer ($\Delta p/p_t$) of 4%, they showed that when changing the outer swirl vane angle (*OSA*) from +20 to +60deg with the inner swirl vane angle (*ISA*) fixed at -45deg (counter swirl combinations with circumferential shear), the minimum *SMD* is obtained at *OSA* between +45 to +50deg. Possible explanations for this tendency would include (a) the effects of the intensity of circumferential shear, (b) weaker interaction of the liquid film with the outer swirling air for larger *OSA*, (c) the effects of deceleration of air velocity downstream by spreading of the swirling flows and so on. However, what is the dominant factor on atomization quality is not well identified yet.

The motivation of the present study is to obtain a clue for a better understanding of the results above, by firstly focusing on the effects of shear intensity on atomization characteristics. For this purpose, a planar airblast atomizer was developed with which the effects (b) and (c) caused by centrifugal forces are eliminated. It was designed so that the two air streams, having the same axial velocity but opposite spanwise velocity components to each other, meet up at the exit of the atomizer to form a longitudinal shear layer to which a planar kerosene liquid film is injected and atomized, as schematically illustrated in the left figures below. The shear intensity was varied by changing the angles of the guide vanes and the air passages (α) of 0, 30, 45 and 50deg.

The performance of the atomizer in terms of air flow field characteristics was firstly evaluated by single-phase numerical simulations. The spanwise non-uniformity of the absolute velocity and the flow angle is to acceptable degree and the atomizer was proven to be working almost as expected.

Its spray characteristics were then investigated by the phase Doppler anemometry, at various test conditions including elevated ambient pressure cases (p_a=0.10-0.82MPa). The results for $\Delta p/p_t$=4% are mainly discussed. As a result, the clear tendency on the effect of shear strength on atomization enhancement is only observed for the higher ambient pressure and the lower fuel flow rate cases (m_f=0.78g/s), as shown in the right figures below. This is mainly due to the relative population increase of small droplets rather than the decrease of the large ones. For the rest, the tendency is not clear. A comparison of the present results with those of the counter-swirl airblast atomizer above suggests the clear effects of swirl combination on the droplet sizes observed in the previous work is not primarily due to the local circumferential shear intensity near the atomizer lip but to other effects such as widely distributed vorticity field by the swirl and/or the centrifugal effects. In addition to this main conclusion, it was also revealed that the effects of p_a and $\Delta p/p_t$ on *SMD* are primarily explained by those of air momentum, a single parameter, and that those of the height of fuel flow passage (0.2 and 0.5mm) are small.





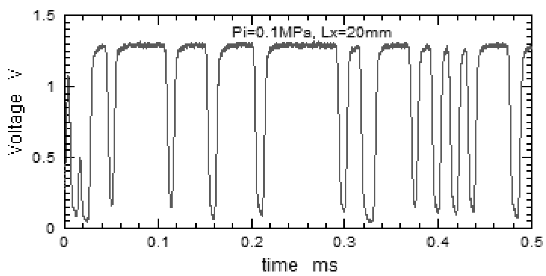
* Corresponding author: matsuura.kazuaki@jaxa.jp

Disintegration of Thin Liquid Jet Injected from Several Tens Micrometer Hole

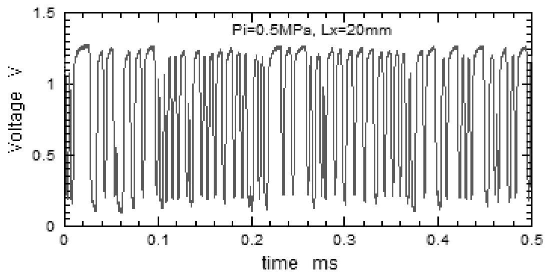
Norihiko Iki
Energy Technology Research Institute, National Institute of Advanced Science and Technology, (AIST), Japan
n-iki@aist.go.jp

Abstract

Water jet from the thin plate nozzle is already disintegrated into droplets array at 10mm of ***Lx*** (distance from the nozzle hole exit). The droplets move inline. The interval of the droplets array becomes larger with ***Lx***. The number density of droplets is estimated to decrease with increase of ***Lx*** and ***SMD*** increases with ***Lx***. Therefore, some droplets combine into one because the relative velocity of droplets is small. Since ***SMD*** at far from nozzle exit depends on the droplets collapse, ***SMD*** at far from nozzle exit can be influenced by various disturbances caused by share force with circumstance air, complicated hole shape, etc. Although breakup length should be discussed with initial diameter of a droplet, there are not enough data to evaluate quantitatively. Therefore time changes of the light intensity caused by the shadowgraph of moving droplets are measured. Since laser beam size at focus point is 0.07mm, the measurement volume is not so larger than droplets. The shadow of the droplet decreases light intensity extremely when the droplet passes through the central axis of the laser beam. When the distance of droplets is larger than the laser beam size, light intensity reaches the maximum. The number of shadows of droplets increases with injection pressure ***Pi***. The time interval of shadows passing through the measurement volume increases with ***Lx***. In the case of ***Pi***=0.5MPa, number of shadows of droplets increases and the time interval of shadows passing through the measurement volume is shorter than that in the case of ***Pi***=0.1MPa as shown in figure 1. These phenomena agree with the decrease of ***SMD*** and the increase of the velocity of the droplets. Data at lager ***Lx*** are obtained since the droplets moves stably. When ***Pi*** increases to 0.7MPa, each shadow of a droplet is hard to be separated at ***Lx***=10mm. The movement of droplet is not as stable as that in the case of ***Pi***=0.5MPa. In the case of ***Pi***=1.0MPa, shadow of a droplet cannot be separated well to the other shadows. The autocorrelation function related with the frequency of the droplet passing through the measurement volume. The first peak point corresponds to the time difference of the next droplets if the interval of the droplets are constant and droplet size is constant. The first peak point shift to zero with increase of ***Pi***. The first peak point increase extremely with ***Lx*** as shown in figure 2. This means that the number of droplets decrease with ***Lx***. That is the coalescence of droplets occurs extremely near nozzle exit.



(a) ***Pi***=0.1MPa



(b) ***Pi***=0.5MPa

Figure 1 Variation of light intensity caused by shadows of droplets (***Dn***=0.03mm, ***Lx***=20mm)

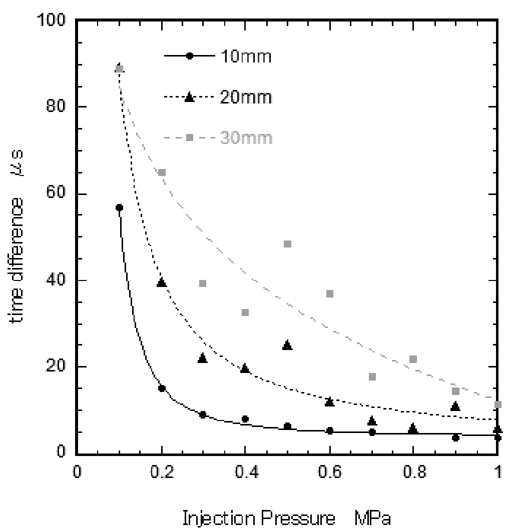


Figure 2 The first peak point of autocorrelation of light intensity caused by shadows of droplets (***Dn***=0.03mm)

Empirical Correlations for Breakup Length of Liquid Jet in Cross Flow-A Review

Soo-Young No
Dept of Biosystems Engineering, Chungbuk National University, Korea
sooyoung@chungbuk.ac.kr

Abstract

The injection of liquid into a high-speed cross flow in combustion systems can be found in the practical applications such as diesel engine, gasoline engine, gas turbine, ramjet and scramjets etc. The application of chemicals to crops in agricultural field and the injection of liquid friction modifiers onto the rail surface are another application of cross flow. Numerous studies have, therefore, been conducted to characterize the liquid jet atomization process in a cross flow. Previous studies of liquid injection in uniform and non-uniform cross flows include analyses of controlled and uncontrolled liquid jets atomized in both subsonic and supersonic airstream. The primary breakup process including column and surface breakup, breakup length, penetration height, jet width, droplet size and droplet velocity have attracted considerable attention. A detailed survey of the empirical correlations related to liquid jet trajectory and penetration height available in the literature was carried out many investigators. As one of atomization characteristics, breakup length is of prime importance in air-breathing propulsion systems.

In this study, the empirical correlations for the prediction of breakup length of liquid jet in cross flow are reviewed and classified. Many empirical correlations have been developed to predict the breakup length of liquid jet in a cross flow. The breakup length can be divided into column fracture height and column fracture distance. Around ten and twelve different correlations have been developed to predict the column fracture height and column fracture distance, respectively. It is known that the breakup length of liquid jet in a cross flow is a basically function of the liquid to air momentum flux ratio. However, Weber number, liquid-to-air viscosity ratio and density ratio, Reynolds number or Ohnesorge number were incorporated in the empirical correlations depending on the investigators.

The existing correlations for column fracture height can be classified as three groups such as momentum flux ratio form, Weber number form, and other parameter form. For column fracture distance, the existing correlations can be categorized as four groups such as the constant form, momentum flux ratio form, Weber number form, and other parameter form. It is clear that there exist the significant discrepancies of predicted values by the existing correlations even though many correlations have the same functional form. The possible reasons for discrepancies will be summarized as the different experimental conditions such as jet operating condition and nozzle geometry, measurement and image processing techniques introduced in the experiment, difficulties in defining the breakup location etc. The evaluation of the existing empirical correlations for the prediction of breakup length of liquid jet in a uniform cross flow is required.

Parametric Study of Fuel Impact on Spray Behavior using High-Speed-Visualization

M. A. Reddemann*, F. Mathieu, D. Cordes, R. Kneer
Institute of Heat and Mass Transfer
RWTH Aachen University
Aachen, Germany

Abstract

In this work, fuel influence on spatial and temporal spray behavior is studied experimentally for a given Diesel-nozzle geometry. Fuels of different molecular groups with various physical and chemical properties are investigated by a parametric study. In addition to conventional Diesel, three alcohols, two alkanes, four silicon oils, two furans, two esters and one ether are used. The spray behavior of each fuel is analyzed for non-evaporating conditions utilizing fifteen different operation points with varied injection pressure and ambient density. The method of choice is high-speed visualization, providing a detailed temporal and spatial view on the spray propagation process. The spray evolution is analyzed with respect to characteristic macroscopic spray parameters such as spray cone angle, penetration length and integrated spray volume. It is found that the influence of fuel properties is reduced with increasing ambient density. An empirical correlation for the macroscopic cone angle as a function of Reynolds number and density ratio is derived and it is found that for non-evaporative ambient conditions mixture formation is mainly driven by density and viscosity. It is shown that the resulting air-fuel-ratio as the main influencing variable for resulting engine emissions is strongly influenced by fuel dependent spray behavior.

---

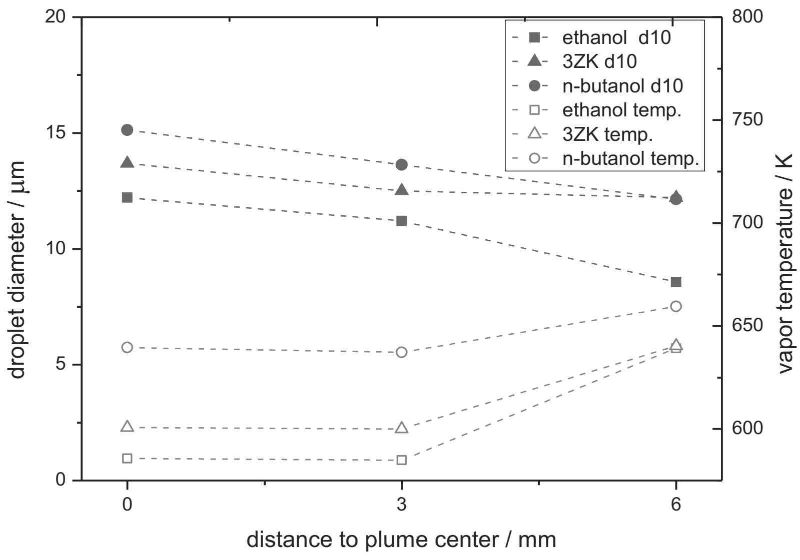
*Corresponding author: reddemann@wsa.rwth-aachen.de

Biofuel Droplet Evaporation Rate of a DISI Spray by Laser-induced Fluorescence and Phase Doppler Anemometry

Tobias Knorsch*, Lars Zigan, Johannes Trost, Michael Wensing, Alfred Leipertz
Dept. Engineering Thermodynamics (LTT) and Graduate School in Advanced Optical Technologies (SAOT), FAU Erlangen-Nuremberg, Germany
Tobias.Knorsch@ltt.uni-erlangen.de and Lars.Zigan@cbi.uni-erlangen.de

Abstract

The atomization and evaporation of gasoline sprays with bio-components differs depending on the respective alternative fuel blend physiochemical properties. This work focuses on estimating the biofuel evaporation rate of sprays at stratified charge conditions. One specific spray plume is analyzed in terms of local droplet size verified by local vapor concentration and temperature. Depending on the operating conditions different physiochemical properties were found to dominate the atomization and evaporation behavior. For moderate ambient temperature and pressure high-boiling point components show a strong influence on the droplet size and temperature distribution in the sprays. However, at elevated temperature the evaporation rate changes completely. Due to a high degree of evaporation taking place, cold spots of 125 K temperature difference appear inside the spray and during the spray process. In the center of the spray plume a maximum cooling of 89 K due to the higher droplet density in those areas compared to the more dilute outer positions are detected. However, when comparing data of two diffident boundary conditions with an ambient temperature difference of 200 K, the measurement positions in the outer regions of a spray cone always show higher temperatures. Smaller droplets as a measure of progressed evaporation are found there. For fuel mixtures with higher evaporation enthalpy the temperature difference between spray center and spray boundary is more pronounced than for fuels with a lower evaporation enthalpy – despite their higher boiling point. Overall, it can be stated that for the droplet evaporation at stratified supercharged conditions, the evaporation enthalpy is a dominating physiochemical property.



fuel	boiling point / °C	evap. enthalpy / kJ/kg
ethanol	78	938.2
3ZK	69-118	844.6
n-butanol	118	637.7

Figure 1 Absolute droplet sizes and spray vapor temperatures at 30 mm distance to the nozzle tip for increased gas pressure and temperature (0.8MPa, 673K) with corresponding key physiochemical parameters at ambient conditions (table)

* Corresponding author: tobias.knorsch@ltt.uni-erlangen.de

Hollow-cone Spray of Viscous Liquid in High-pressure Gas Environment - Experimental Investigation for the Application of New Liquid Fuels to Gas-turbine -

T. SUZUKI*,
Dept. of Mech. Engng, Toyohashi University of Technology, JAPAN
H. NISHIDA, N. HASHIMOTO and Y. OZAWA
Energy Engng Research Lab., Central Research Institute of Power Industry, JAPAN

Abstract

Bio-liquid fuels and non-conventional fossil oils have attracted attentions recently as alternative energy resource for gas turbine power plant. It is convenient that the swirl atomizer can be used to spray these fuels, as was used in the current plants. To obtain fundamental knowledge which is indispensable to survey the applicability of these new liquid fuels for the plant, the experimental investigations were performed on the hollow-cone spray of viscous liquids in high-pressure gas environment. Test liquid employed were water, diesel fuel, palm methyl ester (PME) and silicone-oil #10. Ambient gas was nitrogen of room temperature. Test spray nozzle, Delavan oil burner nozzle 60°A-0.85, was installed in pressure vessel of about 14ℓ. Experimental investigations were performed within the range of liquid flow-rate from 50 to 140cc/min and the range of ambient-gas pressure from 0.1 to 1.0MPa. The behavior of injected liquid-sheet and ensuing spray-flow were observed in detail by flash photography. The apical-angle and the breakup-length of conical liquid-sheet and the cone-angle of spray were evaluated from the photographs. The Sauter mean diameter of spray was measured by the laser diffraction method, and the spray flow field was investigated by the PIV technique. The following conclusions were deduced: (1) The injected hollow-cone spray tended to contract at high ambient-gas pressures. The spray contraction could be observed in wider range of liquid flow-rate when liquid was viscous. (2) The breakup-length of liquid-sheet decreased with increase of the ambient-gas pressure and the liquid flow-rate. The higher viscosity the liquid had, the longer the breakup-length became. (3) The spray cone-angle decreased with increase of the ambient-gas pressure, although the apical-angle of liquid-sheet was almost constant. The spray cone-angle of non-viscous liquid became smaller with increase of the liquid flow-rate, but the spray cone-angle of viscous liquid did not depend so much upon the liquid flow-rate. (4) The Sauter mean diameter of spray decreased with increase of the liquid flow-rate. The mean diameter increased slightly with increase of the ambient-gas pressure. However, the mean diameter was not affected so much by the liquid-viscosity within this experimental range. (5) The surrounding gas should be drafted into the spray. The velocity of drafted-flow did not change so much with downstream-distance, did not change so much with the ambient-gas pressure, and increased linearly with increase of the liquid flow-rate. (6) When the viscosity of liquid was high, several peculiar atomization manners were also observed; roll-up of smooth liquid-sheet, perforation of liquid-sheet and spray pulsation. The experimental range for each manner was examined, and the mechanism of spray pulsation was discussed.

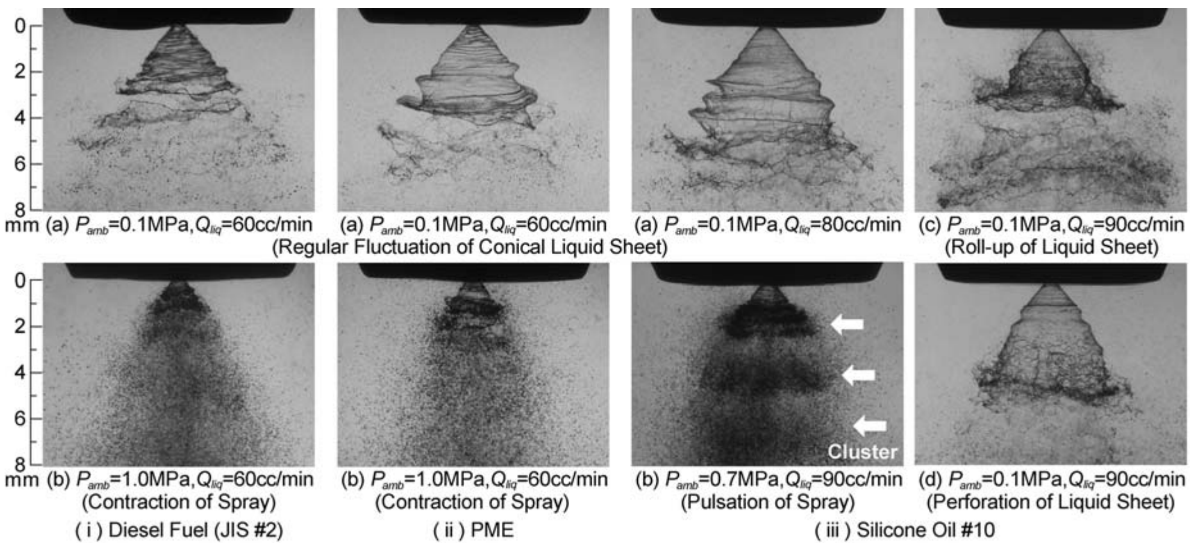


Figure 1 Flash photograph of hollow-cone liquid film.

* Corresponding author: takashi@me.tut.ac.jp

Experimental investigation of spray characteristics of fuel blends having low cetane number and high volatility in a diesel fuel injection system

M. J. Cárdenas*, D. Cordes, R. Kneer
Institute of Heat and Mass Transfer, RWTH Aachen University, Germany
cardenas@wsa.rwth-aachen.de and kneer@wsa.rwth-aachen.de

Abstract

High speed images were used to experimentally investigate the spray characteristics of fuel blends having low cetane number and high volatility properties using a common diesel fuel injection system. Low cetane-number fuels have been chosen, as their resistance to autoignition provides a sufficient ignition delay enabling improved mixing of air and fuel. As the mixing rate depends on the local fuel vapor concentration, high volatility, which assures faster vaporization, also tends to improve the mixing. Therefore, an ethanol-diesel blend and a gasoline-diesel blend have been investigated. One fuel blend was composed of 29.4% ethanol and 68.6% diesel fuel and 2% 1-dodecanol to improve miscibility and the other blend consisted of 40% gasoline and 60% diesel fuel. For Both blends a standard diesel fuel was used, which also served as reference for comparison.

Single injections have been visualized using a high speed camera and compared to standard diesel fuel sprays at different ambient conditions and injection pressures. Typically, 50 injections were recorded to extract the penetration length and spray cone angle of the spray liquid phase from the images.

For all ambient conditions and injection pressures diesel presented the highest spray penetration length values for the same injection time. The results showed that at lower ambient pressures the spray penetration length values of both blends were similar. Once either the ambient or the injection pressure was increased, the ethanol-diesel blend showed higher penetration length values than the gasoline-diesel blend.

For the ambient temperature variation the penetration length values for each fuel were almost equal, while the values of the spray cone angle decreased with increasing ambient temperature. In any other cases, the behavior of the spray cone angle values was inverse to the behavior of the penetration length values for all fuels.

* Corresponding author: cardenas@wsa.rwth-aachen.de

CFD simulations of multi-hole Diesel injector nozzle flow and sprays for various biodiesel blends

C. A. Chasos*, C. N. Christodoulou and G. N. Karagiorgis
Mechanical Engineering Department, Frederick University, Cyprus
eng.cca@fit.ac.cy, eng.cc@fit.ac.cy and eng.kg@fit.ac.cy

Abstract

There is recent interest for the utilisation of renewable and alternative fuels and this is regulated by the European Union, which for the case of biodiesel currently imposes a lower limit of 7% by volume of biodiesel blending in diesel fuel. The biodiesel physical characteristics, as well as the biodiesel blending percentage in diesel fuel affect the injector and nozzle flow, the spray characteristics and the resulting air/fuel mixture and subsequently the combustion quality and the overall engine performance. In the present study, the computational fluid dynamics (CFD) methodology is applied, where the effects of biodiesel blending percentage in diesel fuel on the Diesel injector internal flow and spray characteristics are investigated and compared in order to identify the connections with the engine performance reported in literature. Four test fuels were examined, namely, pure diesel fuel, 10 % by volume of biodiesel blending in diesel fuel, 50 % by volume of biodiesel blending in diesel fuel and pure biodiesel, denoted B0, B10, B50 and B100, respectively. The fuels physical properties were determined from literature and the European standard for biodiesel fuels. For the investigations, a multi-hole Diesel injector geometry was modelled, and for the test fuels three-dimensional simulations of the injector internal flow were carried out. The commercial CFD code STAR-CD was employed for the investigations, where the Eulerian single-phase modelling methodology was used for the injector flow simulations and the Eulerian/Lagrangian two-phase flow modelling methodology was used for the spray simulations. From the injector internal flow simulations of the test fuels, the nozzle exit velocity was determined. The nozzle exit velocity and the physical properties of test fuels were used in an empirical expression to calculate the injected spray angles. The mass flow rate of the fuel, the nozzle geometry data and the spray angle were used as input in the Reitz-Diwakar spray atomisation model. The transient spray simulations were carried out using the Reitz-Diwakar spray atomisation model, along with the Reitz-Diwakar droplet break-up model and O'Rourke inter-droplet collision model. From the simulations, it was found that cavitation takes place at the nozzle inlet for all test fuels and with increasing biodiesel blending percentage the turbulence level in the injector nozzle is reduced. The resulting spray angle decreases with increasing biodiesel blending percentage by approximately 10 to 20 % from pure diesel B0 to pure biodiesel B100 fuel. Comparisons of the simulated sprays of the test fuels were performed, and B0 and B100 sprays at high pressure and temperature chamber conditions were compared against published spray photographs and penetrations. It was found that the resulting spray penetration increases when the biodiesel blending percentage is increasing. From the validation against the experiments, the shape of the body of the predicted sprays was wider than the experimental sprays, and the spray penetration was slightly overpredicted for both pure diesel and pure biodiesel sprays.

*Dr. Charalambos Chasos: eng.cca@fit.ac.cy

Investigation of the Internal Dynamics of Diesel Nozzles by Time-Resolved Laser Doppler Vibrometry

C. Crua*, M. R. Heikal
Centre for Automotive Engineering, University of Brighton, UK
C.Crua@brighton.ac.uk and M.R.Heikal@brighton.ac.uk

Abstract

Experimental and numerical data show that high frequency pressure fluctuations and vibrations exist inside modern diesel injectors. Even though nozzle vibrations may influence the shape and stability of sprays, no thorough spectral investigations have been reported and limited data are available. Experiments were performed to measure time-resolved vibration spectra of diesel injector nozzles using three dimensional laser vibrometry, needle lift sensor and fuel pressure transducer. The vibrometer, which measures the velocity of a vibrating object using the Doppler effect, was used to scan injector nozzle tips during the injection event. In order to allow a comparative investigation of the effect of nozzle type and orifice diameter, the nozzle library included custom-built single-orifice nozzles with VCO and minisac geometries. All nozzles were tested at injection pressures ranging from 60 to 140 MPa. A spectral peak was found around 7 kHz for all nozzles and every injection pressure. Further evidence of a similar frequency was obtained from the pressure sensor and injector needle lift sensor. This frequency is proposed to be caused by the injector's needle oscillation in the axial direction. Oscillations between 35 and 45 kHz where observed during the needle opening phase. For the 200 µm nozzle orifices, these oscillations were found to progressively extend into the steady state injection period as the injection pressure was increased. This was suggested to indicate the presence of quasi cyclic cavitation.

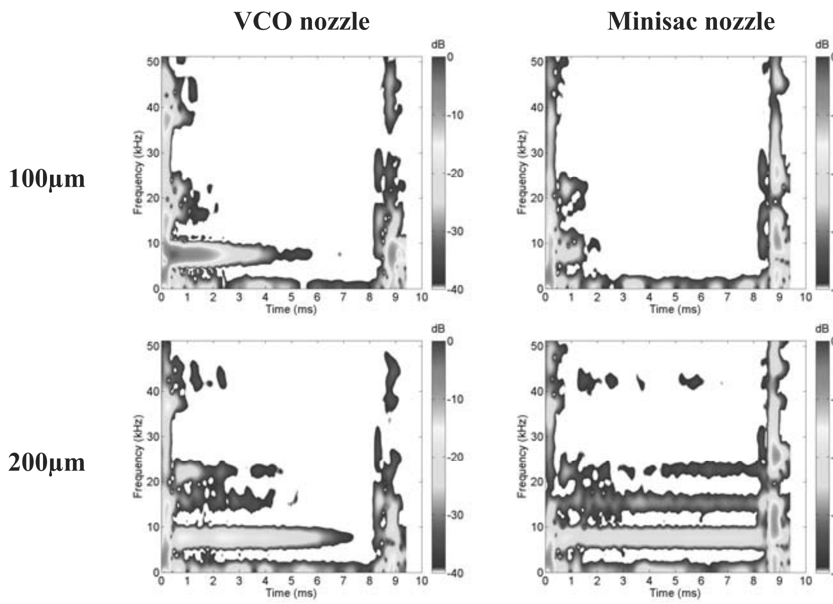


Figure 1. Spectrograms for single-hole VCO and minisac nozzles, with orifice diameters of 100 µm (top row) and 200 µm (bottom row) recorded for injections with 140 MPa pressure

* Corresponding author

Internal Flow Visualization of a Large-Scaled VCO Diesel Nozzle with Eccentric Needle

T. Oda[1*], K. Ohnishi[1], Y. Gohda[2], T. Sumi[1] and K. Ohsawa[1]
[1] Department of Mechanical Engineering, Tottori University, Japan
odate@mech.tottori-u.ac.jp, kenichifuji@gmail.com,
sumi@mech.tottori-u.ac.jp, ohsawa@mech.tottori-u.ac.jp
[2] Mitsubishi Material Techno Co., Japan
kzhr.55344.mp4-20@ezweb.ne.jp

Valve-covered-orifice (VCO) diesel nozzles are usually employed in order to reduce unburned hydrocarbon emissions of diesel engines. Radially eccentric location of needle markedly influences formation of cavitating bubbles inside nozzle and atomization. This result suggests that flow patterns inside the nozzle strongly affect production of cavitating bubbles and primary breakup of liquid jets. Therefore flow visualization was carried out to investigate the effects of eccentricity of a needle inside a VCO diesel nozzle on flow structure of internal flow and primary atomization.

A 10 times large-scaled VCO nozzle had two nozzle holes as shown in Fig. 1. The diameter of both the nozzle holes was 2mm and the length was 8mm. Reynolds number of the flow inside the nozzle hole of the large-scaled VCO nozzle achieved maximum value of approximately 40000, which was nearly the same as that of real-size diesel nozzles. The needle, which was incorporated into the nozzle, was manipulated by a three-dimensional traverse with micrometers. Fine polymer particles were employed as tracer particles for flow visualization, so that photographs of stream lines of the particles are obtained.

Most of discussion in this study focuses on the behavior observed at relatively low needle lift, and the needle is manipulated vertical to both the holes. The visualization shows that flow pattern strongly depend on radial location of the needle, and two modes of flow patterns are observed. On the contrary the flow pattern is almost independent of injection pressure. Flow visualization for first mode of internal flow, which causes the solid cone spray, indicates almost straight stream with weak swirl motion inside the hole when the eccentric needle is located close to the nozzle center. High injection pressure leads to cavitating bubbles at both contraction regions while vortex cavitation is not observable. On the other hand, the second mode of internal flow is encountered when the distance of the needle from the nozzle center is further increased beyond the radial location of the first mode. The flow pattern of the second mode shows strong swirl motion, so that the hollow cone spray is mainly appeared. The upperside contraction region at the hole entrance remains in the second mode although the lowerside contraction region at the hole entrance becomes undetectable. High injection pressure causes a vortex cavitation inside of the hole. Spray cone angle is compared with the swirl angle of flow inside the hole which is obtained from a flow pattern as shown in Fig. 2. The swirl angle was defined as the angle between a centerline of the hole and a stream line across the center line by using the photograph obtained in order to evaluate swirl motion along the hole. This result shows a similar trend to that of the spray cone angle.

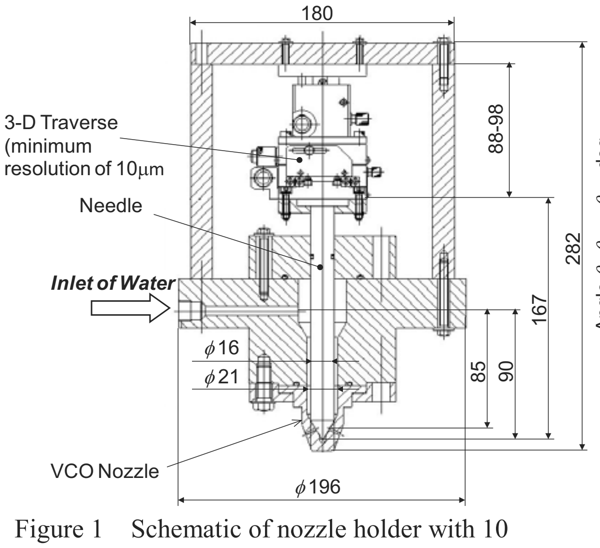


Figure 1 Schematic of nozzle holder with 10 times large-scaled VCO nozzle.

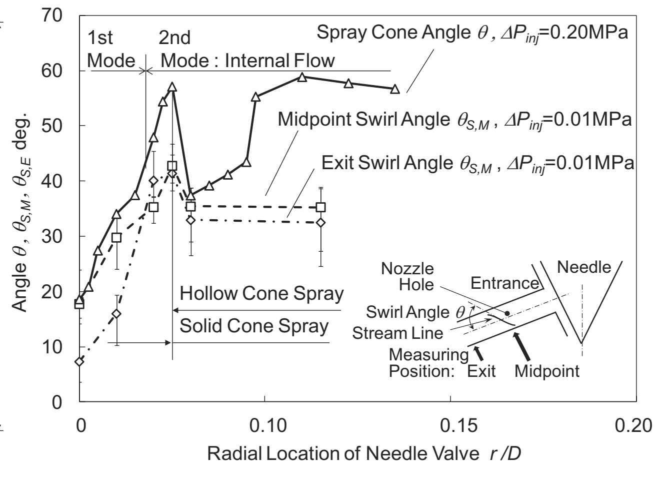


Figure 2 Comparison between spray cone angle and swirl angles. (L_h=0.3mm, ϕ=90°)

* Corresponding author: odate@mech.tottori-u.ac.jp

Near Nozzle High-Speed Measurements of the Intact Core for Diesel Spray

A. Heilig*, M. Kaiser, F. Dinkelacker
Institute of Technical Combustion (ITV)
Leibniz University of Hanover
Welfengarten 1a, 30167 Hannover, Germany

The optical connectivity method is applied for the first time for near nozzle measurements of the intact core length of stock high pressure diesel injectors up to 220 MPa (2200 bar) with high speed imaging. The method is based on an approach by Roosen (1991) and the so called optical connectivity method of Charalampous et al. (2009) for the measurement of the intact core length of sprays. To achieve this, the light is guided through an optical access into the nozzle tip to illuminate the liquid jet along its intact core length from inside. So far, this approach has been applied to rather low pressure injectors or specially designed nozzles.

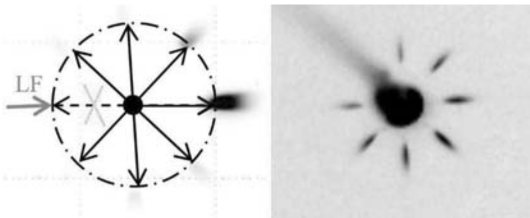


Figure 1. Exemplary, inverted raw pictures: inserted light fiber (LF) through 9 o'clock nozzle hole (additionally visualized pitch circle of nozzle holes respectively direction of the remaining holes) (l.) and centrally coupled light fiber through nozzle tip (r.) (spray core at the 11 o'clock position is hidden by the slightly off focus fiber in the foreground)

For the application on modern common-rail diesel injectors with injection pressures above 200 MPa (2000 bar), two approaches are being evaluated now. In the first approach, an optical fiber is coupled into one of the nozzle holes of a standard heavy duty diesel injector (Fig. 1, l.). Doing so, it was possible to determine the intact core length for the hole on the opposed side. First high speed measurements of the unsteady spray behaviour with up to 25 kHz resolution were feasible. In the second approach the light is guided into the sac hole volume through an additionally drilled access on the center of the nozzle tip (Fig. 1, r.). Here, the lengths of all spray cores could be measured simultaneously. A stable operation was possible up to injection pressures of 220 MPa (2200 bar). Results with transient high speed imaging measurements are then shown. Additionally, a simultaneous application with high speed Mie scattering imaging of the spray shape is presented within this work.

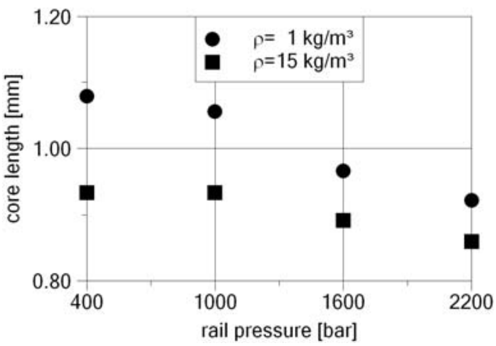


Figure 2. Mean values between $t = 1500\ \mu s$ and $t = 2500\ \mu s$ for variations of the rail pressure and density

*Corresponding author: heilig@itv.uni-hannover.de

Energy balance for single or multiple droplet chains impinging onto a hot slab in the Leidenfrost regime

P. Dunand, M. Gradeck*, G. Castanet, F. Lemoine, D. Maillet
LEMTA CNRS, University of Lorraine, France
michel.gradeck@univ-lorraine.fr and guillaume.castanet@ensem.inpl-nancy.fr

Abstract

This work aims at estimating the energy balance during a single droplet chain or multiple droplet chains interaction with a hot wall whose initial temperature is above the Leidenfrost point. The corresponding experimental data are required for validating submodels for heat transfer between a hot wall and sprays. these physical submodels are subsequently used in CFD modeling.

In the Leidenfrost regime, impinging droplets can experience only three different regimes : perfect bouncing, bouncing with a satellite formation and splashing. Modeling heat transfer in these regimes requires the knowledge of several parameters such as the spreading diameter, heat transfer at the wall, the thickness of the vapour layer, its temperature as well as the liquid heating during the impingment and the evaporation rate. In the splashing regime, data about secondary droplets are also required. Up to now, no experimental data base giving all these parameters are available because of the difficulty to measure simultaneously all these parameters. In this experimental work, some of these parameters have been measured simultaneously during the interaction of the droplet with the heated wall while other ones have been estimated by means of an energy balance.

A monodisperse droplet chain (diameter 80-220μm) is interacting with a hot wall above the Leidenfrost point. A dedicated experimental set-up designed for characterizing heat transfer during the impingement was developped. The slab is a thin nickel plate heated by electro-magnetic induction up to a temperature above the Leidenfrost point. The temperature of the rear face of the nickel sample is measured using an infrared camera and the heat removed from the wall and caused by the presence of the droplets is estimated using a semi-analytical inverse heat conduction model. At the same time, the temperature of the droplets is measured using the two-colour Laser-Induced Fluorescence thermometry which has been extended to imagery for the purpose of these experiments. The measurements of the variation in the droplet temperature occurring during the impact allows determining the liquid sensible heat. Coupling these two measurement techniques allows to estimate the heat flux associated to liquid evaporation. The evaporated mass of the droplet during the impingment is derived from the energy budget of the impingment. A wide range of parameters were tested : Weber number, droplet diameter, droplet injection temperature, injection temperature, corresponding to the bouncing and splashing regimes. Finally, the preliminary results of multiple monodisperse droplets chains are presented. The database reveals that, in the case of small droplets, the cooling mechanism is mainly related to evaporation and the part of the sensible heat gained by the droplets increases as the droplet diameter increases.

On Computational Investigation of the Supercooled Stefan Problem

A. Criscione*, D. Kintea, Z. Tuković†, S. Jakirlić, I. V. Roisman, C. Tropea
Institute for Fluid Mechanics and Aerodynamics, Technische Universität Darmstadt, Germany
†Department of Mech. Engineering and Naval Architecture, University of Zagreb, Croatia

Abstract

Supercooled droplets often occur in clouds which are located at heights which aircrafts usually have to pass during start and landing. When these droplets impact on the surface of the aircraft, the crystallization of the water is triggered and this results in the formation of ice layer on the body of the aircraft. This phenomenon well known as airframe icing is recognized as a significant aviation hazard. It leads to increased aerodynamic drag and weight, associated with a reduction in lift and thrust. Exemplarily only in the U.S. airframe icing accidents led to 819 deaths within the last 19 years.

Supercooled water-ice transition results to be at least a three step process: nucleation, cooperative birth of critical nuclei in the whole sample and growth of the macroscopic solid phase. Concerning the last step two phases are to be considered, a supercooled liquid and a solid phase, and in between of these an interface or moving boundary separating the adjacent phases. The solidification process is described by the energy equation in each phase. The solidification rate is determined by the balance of the heat fluxes at the solidification front. It is governed by the Stefan condition.

In the present work a computational model for the macroscopic freezing mechanism under supercooled conditions relying on the physical and mathematical description of the two-phase Stefan problem is formulated. The relevant numerical algorithm, based on the finite volume method, is implemented into the open source software OpenFOAM©. For the numerical capturing of the moving interface between the supercooled and the solidified liquid an appropriate level set formulation is utilized. The heat transfer equations are solved in both the liquid phase and solid phase independently from each other. At the interface a Dirichlet boundary condition for the temperature field is imposed and a ghost-face method is applied to ensure accurate calculation of the normal derivative needed for the jump condition, i.e. for the interface-velocity in the normal direction. For the sake of updating the level set function a narrow-band around the interface is introduced. Within this band, whose width is temporally adjusted to the maximum curvature of the interface, the normal-to-interface velocity is appropriately expanded. The physical model and numerical algorithm are validated along with the analytical solution.

Understanding instabilities is the first step in controlling them, so to quantify all sorts of instabilities at the solidification front the Mullins-Sekerka theory of morphological stability is investigated. If there is only a low supercooling of the fluid, the interface is stable, i. e. the amplitude of the pertubation decays exponentially, whereas the amplitude rises if the supercooling reaches a critical value. In case the supercooling is greater than the critical value pertubation grows exponentially, i. e. the flat interface is not stable. If the frequency of the pertubation is raised in the case of an unstable interface it will at some point make the interface stable again, which is a result of the Gibbs-Thomson relation. This effect could be simulated with the described code as well.

In summary, we show excellent agreement between our computations and the theoretical results of Stefan's freezing model. Furthermore, we quantitatively check the morphological instability of the pertubated solidification front and compared it against the Mullins-Sekerka theory, obtaining agreement within a few percent. In future work, we intend to demonstrate that our computational model is also able to describe dendritic growth.

*Corresponding author: a.criscione@sla.tu-darmstadt.de

Rebound map for water drop impacts on tilted surfaces

F. Villa, C. Antonini, I. Bernagozzi, N. Ongari, M. Marengo*
Department of Industrial Engineering, Università degli Studi di Bergamo,
Viale Marconi 5, 24044 Dalmine (BG), Italy.

Abstract

Normal and oblique impacts of water drops on dry solid surfaces were studied experimentally, to investigate the conditions for drop rebound. Data from literature suggest that rebound of a drop from a surface can be achieved when wettability is low, i.e. when contact angles, measured at the triple line (solid-liquid-air), are high. However, no clear criterion exists to predict when a drop will rebound from a surface, and which is the key wetting parameter to govern drop rebound (e.g. the "equilibrium" contact angle, θ_{eq}, the advancing , θ_A, or the receding, θ_R, contact angles, contact angle hysteresis, $\Delta\theta$, or any combination of these parameters). As such, experimental tests were conducted to study impacts of millimetric water drops on different dry solid surfaces with variable wettability (i.e. with variable θ_A, θ_R, and $\Delta\theta$), on hydrophobic and superhydrophobic surfaces. The study was focused at performing a phenomenological investigation of drop impact, at understanding in which conditions a drop rebound, and at evaluating drop rebound time (time shift between impact and detachment of the drop from the surface). Impacts were performed on horizontal and tilted surfaces, to evaluate the effect of surface inclination on drop impact outcome. Experimental conditions were: impact speed in the range $0.8 < V < 4.1\text{m/s}$, constant drop diameter $D_0 = 2.55\text{mm}$ (constant), Weber numbers in the range $25 < We < 585$, Ohnesorge number $Oh = 0.0022$ (constant), surface advancing contact angles $108° < \theta_A < 169°$, and receding contact angles $89° < \theta_R < 161°$, and surface inclination angle $0° < \alpha < 80°$.

For normal impact tests, it was found that receding contact angle is the key wetting parameter to control drop rebound: drop rebound was observed only on surfaces with receding contact angle is higher than ~100°; also, drop rebound time decreases monotonically by increasing receding contact angle; a drop rebound map was proposed, accordingly. The analysis of oblique impacts onto tilted surfaces led to the definition of six different impact regimes (see Figure 1). Also, the following was found: on SHS ($\theta > 150°, \Delta\theta < 10°$), surface inclination generally enhances drop rebound and shedding from the surface, reducing drop rebound time up to 30%; on hydrophobic surfaces (with receding contact angles higher than ~100°), rebound was never observed for surface inclination up to 45°; the maximum inclination, α_{max}, at which drops rebound was found to depend on impact Weber number.

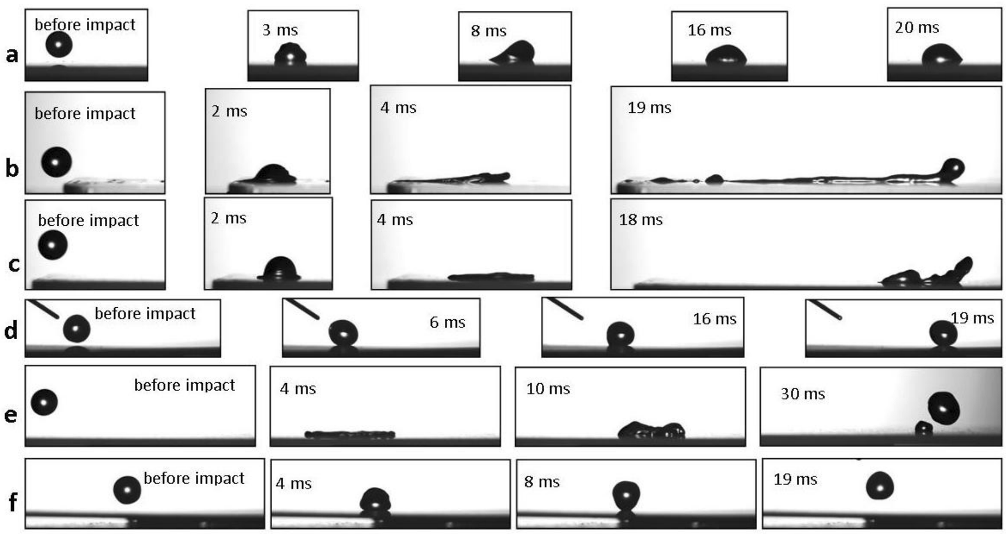


Figure 1: Outcomes of water drop impact onto various tilted substrates: (a) deposition, (b) slug, (c), sliding, (d) rolling, (e) partial rebound, and (f) rebound.

* Corresponding author: marco.marengo@unibg.it

Velocities and size of outgoing droplets after impact on a wall heated above the Leidenfrost temperature

P. Dunand[1,2], G. Castanet[1,2*], P. Villedieu[3], O. Caballina[1,2], F. Lemoine[1,2]
1: Université de Lorraine, LEMTA, UMR 7563, Vandoeuvre-lès-Nancy, F54500, France
2: CNRS, LEMTA, UMR 7563, Vandoeuvre-lès-Nancy, F54500, France
3: ONERA, Toulouse, France
guillaume.castanet@univ-lorraine.fr

Abstract

The interactions between sprays and hot walls occur in a wide variety of applications like spray cooling in the metal processing industry or the direct injection engines. However, their industrial integration remains difficult due to poor understanding of the flow and heat transfer characteristics. In the present study, the emphasis is placed on the impact of droplets onto a wall above the Leidenfrost temperature. In the experiments, chains of equally spaced and mono-sized droplets are impinging onto a smooth nickel slab heated by induction [1]. Shadow imaging using a high-speed camera allows visualizing the droplet impacts and an innovative drop sizing method combined with Particle Tracking Velocimetry, is implemented for the image processing. Shadow imaging offers benefit of measuring size of droplets of irregular shape while providing directly their spatial distribution. Detection of the object shadow outlines is undertaken using the zero of the laplacian. Then a watershed transform of the particles and the curvature of the object's outlines are examined in order to separate overlapping objects. Tracking of the particles allows adding correlations between droplet sizes and velocities. It results in a Lagrangian description of the flow, particularly valuable for the modeling of the impacts. This algorithm is based on Multi-Hypothesis methods, which are preferred technique in multiple targets tracking systems in cluttered environment [2]. This approach has been adapted to the case of droplet splashing. It allows taking into account events such as appearance of a new droplet, temporary vanishing droplets, droplet coalescence and final disappearance of droplets. A plate dotted with calibrated discs allows evaluating the size measurement accuracy and its sensitivity to the depth of field. Finally, the plate is used to fix a threshold value on the maximum gray-level gradient in order to reject out-of-focus droplets. The low rejection rate in the tracking allows having a limited bias.

Measurements are performed with water and ethanol droplets. The main results concern droplet velocities and sizes before and after the impact and their distributions, as well as the maximum spreading diameter and the residence time in the case of droplet rebounds. Extensive comparisons were performed with models existing in the literature [3-5] and some of them have been extended. Contrary to the available literature, the restitution coefficient (ratio between the velocities after and before the impact) of the velocity component tangential to the wall decreases continuously with the Weber number in both bouncing and splashing regimes. The restitution coefficient for the normal velocity differs significantly from unity even for low values of the Weber number in the bouncing regime. It also decreases with the Weber number before it attains quickly an asymptotic value in the splashing regime. The maximum spreading diameter during the rebound is in agreement with existing models and a new correlation is proposed for the residence time as deviations with the modeling become apparent for Weber numbers greater than 30. In the splashing regime, the Sauter Mean Diameter of secondary droplets appears to be function of a combination of Weber and Reynolds numbers [6].

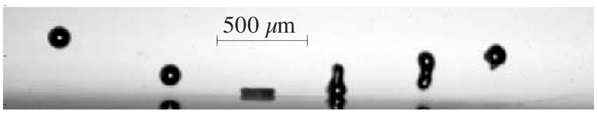


Image of a rebound (f_{inj}=18 200 Hz, D_d=125 μm, We_n=19.26, V=12.37 m/s)

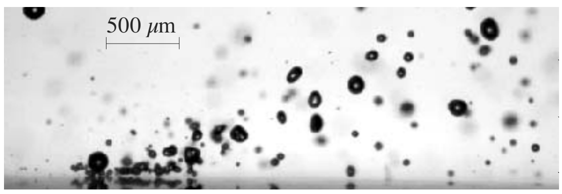


Image of a splashing (f_{inj}=11 000 Hz, D_d=138 μm, We_n=127.55, V=8.9 m/s)

References

[1] P. Dunand, G. Castanet, and F. Lemoine, *Experiments in Fluids 52(4):*843-856 (2012).
[2] D. B. Reid, *IEEE Trans. on Automatic Control* AC:843-854 (1979).
[3] A. Karl and A. Frohn, *Physics of Fluids* 12:785-796 (2000).
[4] K. Park and A. P. Watkins, *International Journal of Heat and Fluid Flow* 17:424-438, (1996).
[5] L. H. J. Wachters and N. A. J. Westerling, *Chemical Engineering Science* 21: 1047-1056 (1966).
[6] A. Moita and A. Moreira, *Experiments in Fluids* 47:755-768 (2009).

Analysis of Combustion Processes in HCCI Engine using LES

D. Goryntsev, A. Sadiki
Institute of Energy and Powerplant Technology (EKT), TU-Darmstadt, Germany
digor@ekt.tu-darmstadt.de and sadiki@ekt.tu-darmstadt.de

Abstract

The Homogeneous Charge Compression Ignition (HCCI) is still a promising concept to optimize internal combustion (IC) engines with respect to emissions and particulate matter. Having the potential of both providing high efficiency that is similar to diesel engine combustion and also producing ultra-low emission characteristics of NOx and particulate matters the HCCI features combustion properties different from the two well-known, namely the premixed combustion mode in SI gasoline engines and diffusion flames in diesel engines. However in HCCI combustion processes the cycle-to-cycle variations of in-cylinder flow play an important role and can lead to combustion instability due to the lean as well as knock combustion conditions. In this paper, multi-cycle Large-Eddy Simulation (LES) based analysis is carried out on a single cylinder, four-stroke IC-engine with two vertical valves in order to characterize the unsteady effects of HCCI combustion processes. For this purpose, LES simulation of HCCI combustion based on fully premixed iso-octane and early-direct spray injection has been carried out for 40 and 25 engine cycles obtained on coarse and fine grid, respectively. In order to reach the sufficient number of statistically independent samples a parallelization strategy has been used allowing perform LES of cyclic fluctuations in HCCI IC-engine with reasonable statistical accuracy. The effects of cycle-to-cycle velocity fluctuations on the resulting HCCI combustion processes are pointed out. In particular, a qualitative analysis of the intensity of cyclic fluctuations of in-cylinder velocity, temperature and pressure is provided in terms of mean and standard deviation. A verification based on mesh refinement study has been done in the present paper.

The effects of cycle-to-cycle velocity fluctuations on HCCI combustion processes are discussed. As an example, Figure 1 shows the cycle-to-cycle variations of velocity and temperature in the cross section of the combustion chamber for 25 consecutive engine cycles at 15 crank angle degree (CA) before top dead centre (TDC). Figure 2 demonstrates the cycle-to-cycle variations of the temperature obtained in the perpendicular sections of the combustion chamber for 4 consecutive engine cycles at CA = 5° before TDC for fully premixed HCCI combustion. There is only effect of velocity cyclic fluctuations which are initialized during the intake engine stroke taking into account. The initial and boundary conditions were kept identical for all considered cycles.

a) b) c)

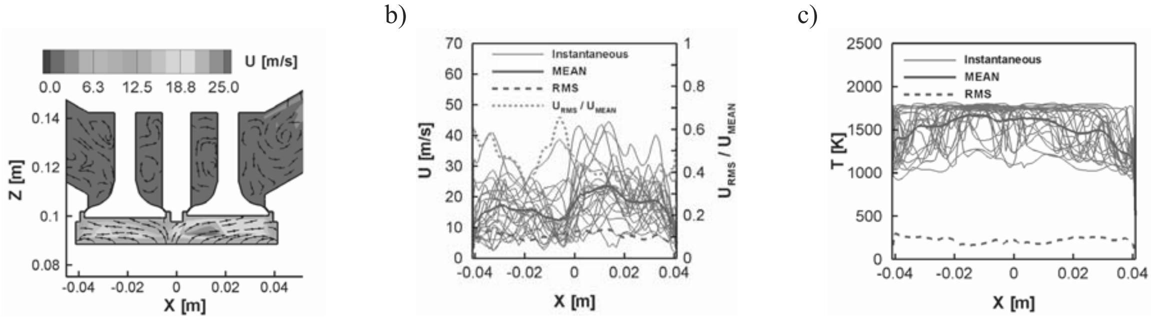


Fig. 1. Contour plots (a), instantaneous profiles and standard deviation of velocity (b) and temperature (c) in the cross section of the combustion chamber. Data obtained for 25 engine cycles at top dead centre for homogeneous autoignition at CA = 15° BTDC, fine grid.

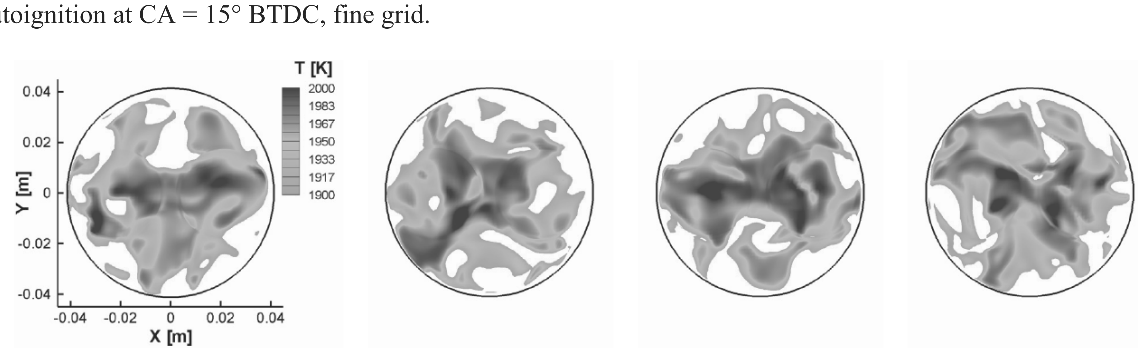


Fig. 2. Cyclic variations of temperature field for premixed HCCI combustion, 4 consecutive engine cycles. XY plane (z = 0.095 m) of the combustion chamber, CA = 5° before TDC, fine grid.

Application of the FGM Method to Spray A Conditions of the ECN database

S. Ayyapureddi*, U. Egüz, C. Bekdemir, L. M. T. Somers, L. P. H. de Goey
Eindhoven University of Technology, the Netherlands
s.ayyapureddi@tue.nl, u.eguz@tue.nl, c.bekdemir@tue.nl, l.m.t.somers@tue.nl and l.p.h.d.goey@tue.nl

Abstract

Modeling turbulent diesel spray combustion which combines complex flow and transport phenomena with combustion event including a vast amount of species and reactions is a major challenge. The Flamelet Generated Manifold (FGM) method is a promising technique to model reacting flows using tabulated chemistry approach. The method is adopted for diesel spray combustion by tabulating chemistry as a function of the mixture fraction (Z) and a reaction progress variable ($\mathcal{Y}$). In previous work, the method has been successfully applied to simulate Spray H cases as defined by the engine combustion network (ECN). Two different tabulation approaches (igniting counterflow diffusion flames (ICDF) and homogeneous reactors (HR)) were investigated and compared to the available experimental data of the ECN.

In this paper, the FGM method is applied to simulate Spray A conditions of the ECN. First, the sensitivity of the spray sub-models (atomization and breakup models) is studied for the non-reacting case of the Spray A setup. Later, the FGM approach is applied on the reacting case for FGM's generated with two different n-Dodecane reaction mechanisms, using two tabulation approaches, and with and without inclusion of a turbulent closure (PDF approach based on variance of Z). The 3D-RANS (Reynolds Averaged Navier-Stokes) simulations are performed with the commercial CFD code STAR-CD. The combustion results are analyzed by comparing the simulated and measured ignition delays and lift-off lengths. One mechanism results in ignition for all simulations, whereas the other mechanism does not. It was found that this can be attributed to the different sensitivity of the mechanisms to the strain rate. In general, HR tabulation predicts shorter ignition delay and lift-off length (LOL) than the ICDF in line with the observations from previous work. The atomization model does not show major effect on ignition delay however it affects the LOL significantly in both tabulation approaches. Inclusion of the turbulent closure does not affect ignition delay or LOL predictions. In general compared to the experiments, the ICDFs slightly over predict whereas the HRs systematically under-predicts.

* Corresponding author: s.ayyapureddi@tue.nl

Simulation of laminar flame propagation in a multicomponent droplet stream

A. Bruyat, C. Laurent, O. Rouzaud, G. Lavergne
ONERA-The French Aerospace Lab, F31055 Toulouse
anne.bruyat@onera.fr and claire.laurent@onera.fr

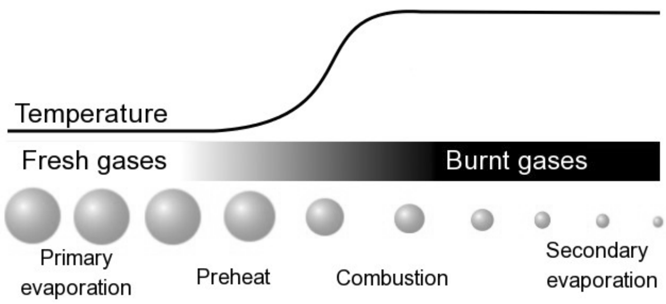
Abstract

To predict alternative fuels performances at ignition, a multicomponent evaporation model [1] was implemented into the lagrangian solver of ONERA's CFD code CEDRE. Several chemical components are now considered in the droplets, and the vaporisation process is sequential. The present study aims at testing the droplet composition effect on spray combustion.

The academic case of a 1D laminar flame propagating along a droplet stream is investigated. The first parametric study deals with monocomponent n-decane droplets and the results are compared to the equivalent gaseous flame configuration in order to understand vaporisation effects [2][3][4]. A reduced chemical scheme is considered for which Arrhenius preexponential factor is calculated from the equivalence ratio [6]. The evolution of the flame speed versus the total equivalence ratio is well reproduced, even if the flame speed value is slightly overestimated due to the simple chemistry used in the present study. The droplet diameter strongly affects the flame speed for a fixed total equivalence ratio (the flame speed is slower when the diameter is larger) [5].

Then 1D laminar flame computations are performed for multicomponent droplets. Droplets are composed with n-decane and n-hexadecane in order to underline how component volatilities alter flame behaviour. As both components are n-alkanes, n-decane and n-hexadecane are supposed to have the same chemistry mechanism, in order to only study the sequential vaporisation effect on flame propagation. Two reduced chemical schemes adapted to multicomponent combustion are compared [7][8].

The modified parameters are the droplets composition, the droplets initial diameter, and the total equivalence ratio. Results on the flame velocity, its temperature and its thickness are analysed. The flame structure depends on the droplet composition and the global behaviour of the 1D laminar flame for bicomponent droplet is halfway between results obtained for each monocomponent droplet (pure n-decane, and pure-n-hexadecane). The composition of burnt gases in rich combustion strongly depends on the total equivalence ratio, the diameter and composition of the droplets. The next step will be to consider different chemical kinetics. The final objective is to caracterize separately each effect of the droplet composition.



References

[1] B. Abramzon, W.A. Sirignano, Int. J. Heat Mass Transfer 32, 16051618, 1989
[2] M. Boileau, Universite de Toulouse, PhD Thesis, 2007
[3] J.M. Senoner, Universite de Toulouse, PhD Thesis, 2009
[4] A. Neophytou, E. Mastorakos, Combustion and Flame, Volume 156, 2009
[5] Ballal DR, Lefebvre AH., Proceedings of the Combustion Institute 18, 3218, 1981
[6] B. Franzelli, E. Riber, M. Sanjose, T. Poinsot, Combustion and Flame, Volume 157, Issue 7, July 2010
[7] Doue, ISAE, PhD Thesis, 2005
[8] S.K Aggarwal, Combustion and Flame 76, 1989

Modeling the chemical structure of spray flames using tabulated chemistry method

B. Franzelli[a,b,*], B. Fiorina[a,b], N. Darabiha[a,b]
[a] Ecole Centrale Paris, Grande Voie des Vignes, 92290 Châtenay-Malabry, France
[b] CNRS, UPR 288, Laboratoire d'Energétique Moléculaire et Macroscopique, Combustion (EM2C), Grande Voie des Vignes, 92290 Châtenay-Malabry, France
benedetta.franzelli@ecp.fr and benoit.fiorina@ecp.fr and nasser.darabiha@ecp.fr

Abstract

A complete comprehension of the spray combustion is necessary to guarantee security and to reduce pollutant emissions in aircraft combustions and internal combustion engines. Spray combustion presents a complex nature, comprising liquid dispersion into droplets and evaporation, and is an active topic of research. The fuel droplets evaporation process causes strong inhomogeneities of equivalence ratio in the fresh gases. The chemical structure of the spray flame is therefore highly complex presenting premixed-like and non-premixed-like reaction zones. Numerical prediction of the combustion phenomena such as the flame propagation or the pollutant predictions, which are particularly sensitive to detailed chemical effects, is very challenging and requires a precise and reliable chemical description. Flamelet-based tabulated chemistry methods have been initially developed to introduce detailed chemistry in gaseous flames with a reasonable computational cost. They assume that the local turbulent flame structure is similar to those of single flamelet elements. For instance, PFT methods are based on fully-premixed flamelets whereas the DFT tables are constructed using diffusion flamelets. These approaches seem to be unable to predict the complex chemical structure of spray flames since they are based on a single combustion regime. Multi-regime flamelet methods have recently been proposed to model the structure of complex flames where both premixed-like and diffusion-like reactive layers are present and seem better adapted for spray flames. The performance of tabulated chemistry method for spray combustion is attractive but it has never been rigorously identified. The present work analyses the capability of tabulated chemistry methods to model the chemical structure of spray flames assuming that the chemical subspace accessed by a spray flame can be mapped by a collection of adiabatic gaseous flamelets. Different tabulated chemistry strategies are considered: the PFT method, based on premixed flamelets, the DFT approach, based on diffusion flame elements, and a new technique called Partially-Premixed Tabulated Flamelet (2PFT). In the 2PFT method information from partially-premixed flames are stored into a 3-D look-up table parametrized as function of the progress variable Y_c, evolving monotonically between fresh and burnt gases, of the mixture fraction Y_z, denoting the local equivalence ratio, and the scalar dissipation rate of the mixture fraction χ^*, which identifies the combustion regime.

The three techniques have been tested on a 1-D laminar axisymmetric counterflow kerosene/air spray flame. Since the spray flame structure strongly depends on the gas and liquid phase properties at injection, an exhaustive collection of spray flames has been calculated using a detailed kinetic mechanism and complex thermodynamic properties, for different values of the liquid and gas velocity, the droplet diameter, the liquid volume fraction and the injection temperature at injection. The capability of the PFT, DFT and 2PFT techniques to reproduce the structure for all the considered spray flames has been assessed and the impact of mapping spray flames with a chemical subspace on adiabatic and gaseous flamelets has been evaluated.

---

*Corresponding author: benedetta.franzelli@ecp.fr

Formation and Breakup of Ligaments from a Rotary Bell Cup Atomizer

M. Shirota[1*], Y. Hatayama[1], T. Haneda[1], T. Inamura[1], M. Daikoku[2], Y. Saito[3], H. Aoki[3]

1. Course of Intelligent Machines and Engineering, Graduate School of Science and Technology, Hirosaki University, Hirosaki, Aomori, Japan
2. Department of Mechanical Engineering, Faculty of Engineering, Hachinohe Institute of Technology, Hachinohe, Aomori, Japan
3. Department of Chemical Engineering, Graduate School of Engineering, Tohoku University, Sendai, Miyagi, Japan

Abstract

Rotary bell cup atomizer is commonly used in industrial spray painting for automobiles. In a rotary bell cup atomizer, liquid is spread on an inner bell-shaped surface of the rotating cup due to centrifugal force caused by a high rotational speed. The thinned liquid film is then split over the grooves at the peripheral edge of bell cup. Over a certain rotational speed of cup, ligaments are formed from the peripheral edge. The ligaments are further elongated and finally collapsed into droplets.

In the present study, we experimentally investigated the formation and breakup of ligaments ejected from a high-speed rotary bell cup atomizer. The rotational speed of bell cup was varied in the range between 5000 to 30000 rpm, while volumetric liquid supply rate was kept constant at 300 mL/min. We used glycerol aqueous solutions with viscosity of 30, 70 and 130 mPa·s. The elongated shape and breakup behavior of the ligaments were observed by using short exposure photography with a 180 nsec flash light. By using image processing based on edge detection and curve fitting, we quantitatively evaluate the following geometric properties of ligaments: trajectory of centerline, local curvature corresponding to the diameter of circular cross section and length of ligaments. Figure 1 and 2 show a typical examples of the local curvature and change in ligament diameter along its axis obtained through the image processing. The droplet size distributions were measured by using a particle size analyzer based on Fraunhofer diffraction theory. The uncertainty of photographic measurements was found to be 5 μm, the main factor of which arose from the optical resolution of the photography. In addition, a theoretical model on the droplet formation from a rotary atomizer with grooves at the edge is proposed and evaluated with experimental results. This theoretical model includes film thinning due to centrifugal force, film split over grooves at bell cup edge, ligament root formation at bell cup edge, and stretch and breakup of ligament.

The conclusions are summarized as follows: (1)The film spread on the inner surface of the bell cup is split over the groove at the edge of the cup when the thickness of the film is smaller than the depth of the groove. (2) The diameter of ligament at bell cup edge is determined by the width of the split film. (3)Simple momentum balance equating the momentum fluxes at the bell cup edge and some position downstream where the velocity profile flattens provide good estimation on the thinning rate of ligament diameter along its axis. In the analysis of momentum balance, we examined the effect of axial surface tension, Young-Laplace over pressure due to curved interface of ligament, centrifugal force, and viscous dissipation due to stretching. (4) Weber's theory on the breakup of viscous ligament gives good estimation on the diameter of droplet of liquids having Ohnesorge number ranging from about 1 to 5.

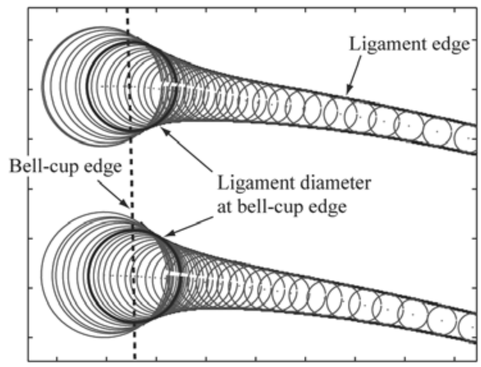


Fig. 1 Local ligament curvature obtained via image processing.

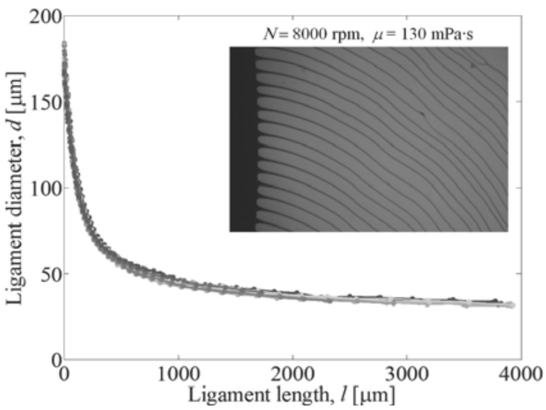


Fig. 2 Change in ligament diameter along its axis.

* Corresponding author: mshirota@cc.hirosaki-u.ac.jp

Air assisted atomisation of a liquid film investigated by way of improved phase detection probes

Sylvain Marty*, Jean-Philippe Matas, Alain Cartellier, Stephane Gluck
LEGI, University of Grenoble, France
A2 Photonic Sensors, Grenoble, France
sylvain.marty@hmg.inpg.fr, jean-philippe.matas@hmg.inpg.fr, alain.cartellier@hmg.inpg.fr and sgluck@a2photonicsensors.com

Abstract

Gas assisted atomization, i.e. the atomization of a slow liquid film, sheet or jet by a rapid gas stream, is commonly exploited in turboreactors as well as in cryotechnic rocket engines. As combustion strongly depends on the characteristics of the spray, to decrease the amount of pollutants emitted.

Previous investigations on droplets stripped off the interface (Figure 1) have shown that three successive instabilities drive the atomization. First, longitudinal waves are formed by a Kelvin-Helmholtz type instability those most amplified wavelength is controlled by the gas vorticity thickness δ_G at the injector exit (Villermaux & Marmottant 2004). The axial frequency prediction was recently improved by accounting for the presence of the splitter plate between gas and liquid (Matas et al. 2011). Second, ligaments arise on the wave crest by a wind induced Rayleigh-Taylor instability (Hong et al. 2002, Varga et al. 2003). These ligaments break then into droplets (Villermaux 2007).

Based on this scenario, a phenomenological model has been proposed for the mean drop size <D> (Hong et al. 2002): <D> evolves as $\delta_G\ We^{-1/2}$ where the Weber number = $\rho_G\ (U_G - U_C)^2\ \delta_G\ /\sigma$, U_C being the convective velocity of the axial instability. That proposal proved valid both for planar and axisymetric configurations (Ben Rayana et al. 2006). Yet that model was only tested in the limit of large dynamic pressure ratio $M=\rho_G\ U_G^2/\rho_L U_L^2$, namely for M about 10 and above that correspond to conditions encountered in cryotechnic engines and in turboreactors during take off. M values down to unity or below arise during cruise or relight. Thus, our recent investigations were aimed at testing the model validity at low M and to check if there is any change in the instability mechanisms.

To access drop size and flux, we exploited a new version of the phase detection optical probe (Hong et al. 2004) with a sensitive length as small as 10μm to be compared with 20μm for the previous sensors. The performances of such probes were thoroughly checked. In particular, drops chord distributions from these two sensors proved nearly identical (Figure 2) while the small sensitive length probe detects more droplets below 10-15μm.

The influence of the M parameter on the drop size will be discussed. Measurements indicate a weak sensitivity of the mean drop size and of the chord distributions on the M parameter in the range 4 to 16.

Figure 1

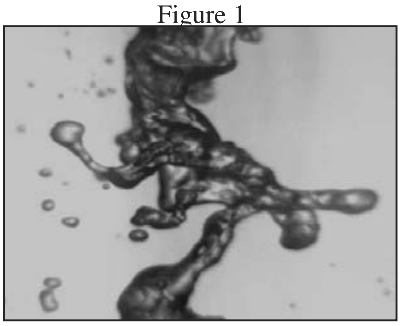


Figure 2

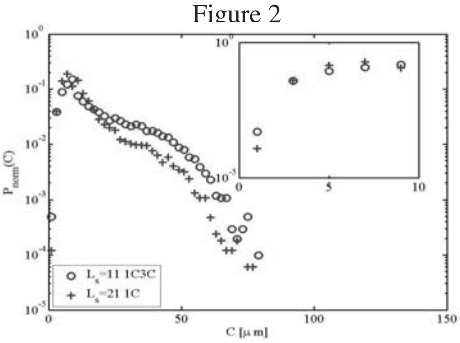
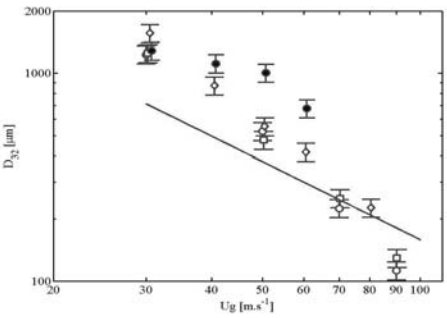


Figure 3



1. Ben Rayana F., Cartellier A., Hopfinger E., Assisted atomization of a liquid layer: investigation of the parameters affecting the mean drop size prediction, (paper ICLASS06-190), CD Proc. ICLASS 2006, Aug. 27 – Sept. 1, Kyoto, Japan, ISBN 4-9902774-1-4, Publ. Academic Publication and Printings Co., (2006).
2. Hong, M., Cartellier, A. and Hopfinger, E. J. Atomization and mixing in coaxial injection. (2002). Proc. 4th Int. Conf. On Launcher Technology. Liege, Belgium (3- 6 Dec. 2002).
3. Hong M., Cartellier A., Hopfinger E., Characterization of phase detection optical probes for the measurement of the dispersed phase parameters in sprays. *Int. J. Multiphase Flow* **30**, 615-648 (2004).
4. Marmottant, P., and Villermaux, E. On spray formation. J. Fluid Mech., 498 (2004), 73-111.
5. Matas J.-Ph., Marty S., Cartellier A., Experimental and analytical study of the shear instability of a gas-liquid mixing layer, *Phys. Fluids* **23**, 094112 (2011); doi:10.1063/1.3642640
6. Varga, C. M., Lasheras, J. C., and Hopfinger, E. J. Initial breakup of a small-diameter liquid jet by a high-speed gas stream. J. Fluid Mech., 497 (2003), 405-434.
7. Villermaux E., Fragmentation, Ann. Rev. Fluid Mech., 2007. 39:419–46

* Corresponding author: sylvain.marty@hmg.inpg.fr

An Analysis of the Surface Breakup Mechanism of a Liquid Jet in Cross-flow

M. Behzad, N. Ashgriz*
Department of Mechanical and Industrial Engineering, University of Toronto, Canada
m.behzad.jazi@mail.utoronto.ca and ashgriz@mie.utoronto.ca

Abstract

The surface breakup of a liquid jet injected into a gaseous cross-flow is observed in moderate to high gas Weber number conditions which normally occur in the combustion applications. Although many experimental studies on the breakup regimes, the mechanism of jet surface breakup has not been fully understood because of difficulties to capture the near nozzle breakup phenomena. This study aims at providing useful observations regarding the fundamental physics involved in the surface breakup mechanism of a liquid jet in cross-flow in relatively high Weber number, using detailed numerical simulations. The results show that infinitesimal disturbances are excited immediately after the jet is exposed to the gas flow. As the disturbances are transported along the jet trajectory, they start to grow due to the shear instability. Subsequently a two-stage mechanism causes the jet surface to break up. In the first stage, the cross-flow drags the crests of waves in the downstream direction, which results in formation of sheet-like structures protruding to the leeward of the jet. In the second stage, the sheet surrounded by a thick rim, disintegrates into ligaments and finally droplets due to the propagation of span-wise waves.

*Corresponding author: ashgriz@mie.utoronto.ca

Instability of a Planar Liquid Sheet Formed with Surfactant Aqueous Solution

T. Wakimoto*, R. Fujii, K. Katoh
Osaka City University, 3-3-138 Sugimoto, Sumiyoshi-ku, Osaka, Japan
wakimoto@mech.eng.osaka-cu.ac.jp
m11taF0333@ex.media.osaka-cu.ac.jp
katoh@mech.eng.osaka-cu.ac.jp

Abstract

In this study, we clarified the growth characteristics of the instability waves of a planer liquid sheet formed with a surfactant solution. It has been widely reported that a liquid sheet jet oscillates in a sinuous shape because of Kelvin–Helmholtz instability, and that the growth rate of the instability wave depends on the surface tension. In addition, a surfactant solution has time-dependent surface tension, which is referred to as "dynamic surface tension," because the adsorption of surfactant molecules into a newly formed surface requires a certain time. For this reason, the surface tension on a new surface decreases from the surface tension of a solvent to the equilibrium surface tension of the solution for a specific relaxation time. This suggests that the surface tension of the liquid sheet formed with the surfactant solution decreases downstream. In this study, wave growth as affected by dynamic surface tension was clarified using the planar liquid sheet. The planar liquid sheet was ejected from a slit nozzle with a 0.5-mm gap and 150-mm width at a velocity of 8 m/s, and an artificial perturbation of 250 Hz was introduced at the nozzle to induce precise periodical instability waves. Polyoxyethylene-(10)octylphenyl ether was selected as a surfactant. Fluorescence dye was dissolved in the surfactant solution, and the cross section of the wavy liquid sheet was visualized with a laser sheet to measure the amplitudes of the waves. This study consists of four stages. First, the dependency of the spatial growth rate of the wave on surface tension was determined experimentally using ethanol aqueous solutions, which have a time-invariant surface tension. The measured growth rates of the ethanol solutions at different concentrations indicated that the growth rate decreases linearly with an increase in surface tension. Second, the dynamic surface tensions of the surfactant solutions were measured at concentrations of 250, 1500 and 5000 ppm, and the locally varied surface tension of the liquid sheet was estimated. The relaxation times t_r of the dynamic surface tensions at 250, 1500 and 5000 ppm were 99, 9.6 and 0.13 ms, respectively. Because the liquid sheet produced at the nozzle required time t_f = 5–15 ms to reach the measurement zone of the wave amplitude, the magnitude relations between t_r and t_f at 250, 1500 and 5000 ppm were $t_r >> t_f$, $t_r \approx t_f$ and $t_r << t_f$, respectively. Third, the wave amplitudes of the surfactant solutions were predicted using the results of these two steps. The dependency of the spatial growth rate on surface tension and the measured dynamic surface tension gave a spatially varied growth rate. The wave amplitude was calculated from the integration of the growth rate. Finally, the calculated wave amplitude was compared with the measured amplitude of the surfactant solution. The good agreement between the calculated amplitude and the experimental one proved that the growth characteristic of a surfactant solution depends on dynamic surface tension. In addition, the measured amplitudes at 250 and 5000 ppm were almost the same as the amplitudes that were calculated under the assumption of a constant equilibrium surface tension of the solvent and solution, respectively, because these solutions have a short/long relaxation time t_r in comparison with time t_f = 5–15 ms required for the flow.

---

* Corresponding author: wakimoto@mech.eng.osaka-cu.ac.jp

Atomization of liquid film in annular flow: Studying tracks of droplets at the initial stages using high-speed LIF system.

S.V. Alekseenko1,2, A.V. Cherdantsev1,2*, D.M. Markovich1,2 and A.V. Rabusov1,2
1 Kutateladze Institute of Thermophysics, Novosibirsk, Russian Federation
2 Novosibirsk State University, Novosibirsk, Russian Federation
aleks@itp.nsc.ru, cherdantsev@itp.nsc.ru, dmark@itp.nsc.ru, arabusov@gmail.com

Abstract

Atomization of liquid film in annular flow essentially changes the integral characteristics of flow. The most part of atomization mechanisms, available in literature, are related to the wavy structure of liquid film. In our recent works (e.g., [1]) we applied high-speed laser-induced fluorescence technique to studying the wavy structure of liquid film in annular flow. Local intensity of fluorescent light was recalculated into local film thickness. The technique allowed us to study the temporal evolution of instantaneous distribution of film thickness over one longitudinal section of the pipe with high spatial (0.2 mm) and temporal (0.1 ms) resolution. Analyzing such data, one can easily distinguish disturbance waves; slow ripples, covering the base film; fast ripples, covering crests of the disturbance waves.

One more type of structures is visible in the experimental data: very short and abnormally high 'waves', moving with velocity several times greater than that of disturbance waves. Normally these trajectories begin at the disturbance waves. No such trajectories were observed in regimes without liquid entrainment. The most reasonable explanation is that such objects represent tracks of droplets, which travel along the investigated section of the pipe not far from the film surface. The Figure shows evolution of film thickness in space (abscissa) and time (ordinate). Wide bright band represents disturbance wave, narrow bright lines – to the droplets. Thus, though the technique was originally aimed for film thickness measurements, it can be also used for investigation of droplets dynamics. Advantage of the method is the possibility to study the droplets at the initial stage of their evolution in both space and time. Disadvantages of current realization of the method are related to the use of two-dimensional (longitudinal distance and time) measuring system and relatively low spatial and temporal resolution.

Processing of tracks of droplets was performed in view of three mechanisms of entrainment. The first one is the disruption of fast ripples, moving over crests of disturbance waves [2]. Velocity and longitudinal size of just-entrained droplets were estimated. The second mechanism of entrainment is the coalescence of disturbance waves (see Figure) [3]. Comparison of the number of droplets, observed right after the coalescence event, to the average number of droplets, generated by single disturbance wave, didn't show any essential difference. The third mechanism is the secondary entrainment after the impingement of a depositing droplet into the surface of the base film. It was observed that the depositing droplet normally impinges into a slow ripple, and, in part of cases, the new secondary droplet appears. It rapidly accelerates, reaching nearly the same value of velocity as that of 'primary' droplets.

References:

1. Alekseenko SV, Antipin VA, Cherdantsev AV, Kharlamov SM, Markovich DM (2008) Investigation of waves interaction in annular gas-liquid flow using high-speed fluorescent visualization technique. Microgravity Sci Technol 20:271-275.
2. Woodmansee DE, Hanratty TJ (1969) Mechanism for the removal of droplets from a liquid surface by a parallel air flow. Chem Engng Sci 24:299-307.
3. Wilkes NS, Azzopardi BJ, Thompson CP (1983) Wave coalescence and entrainment in vertical annular two-phase flow. Int J Multiphase Flow 9:383-398.

*Corresponding author: cherdantsev@itp.nsc.ru

Atomization Performance of Algae-Derived Renewable Diesel in a Swirl-Stabilized Research Combustor

J. M. Legg, A. A. Narvaez, V. G. McDonell*
UCI Combustion Laboratory, University of California, Irvine, USA
jml@ucicl.uci.edu, aan@ucicl.uci.edu, and mcdonell@ucicl.uci.edu

Abstract

This study considers the atomization performance of an Algae-derived renewable diesel fuel in a twin-fluid airblast atomizer. Two other alternative liquid fuels, biodiesel and ethanol, are considered in addition to conventional petroleum-based fuels (#2 Diesel and F-76 Military Distillate). As the Algae-derived fuel is produced through hydrotreatment, its chemical composition and physical properties are very similar to the conventional fuels; in this study, this fuel was blended with the conventional F-76. To characterize the atomization performance, the breakup process was investigated using high-speed cinematography, and droplet sizes were measured using ensemble laser diffraction. The high-speed video of the spray also provided a basis for a Particle-Image Velocimetry (PIV) analysis, resulting in a velocity-field measurement of the atomizer. All of the fuels exhibited very similar atomization characteristics. Due to the atomizer geometry and flow conditions, prompt atomization was exhibited for all fuels, resulting in a relative insensitivity of liquid properties on the droplet sizes. The traditional jet-breakup effects of wavy-deformation were found to be insignificant, as the low liquid flow led the process to be dominated by aerodynamic effects. A relevant correlation for Sauter Mean Diameter (SMD) was selected, showing excellent agreement over a range of flow conditions for all fuels except biodiesel, due to its high viscosity. Despite this, at the best conditions all fuels exhibited very small drop sizes (10-25 microns). The velocity fields were very similar for all fuels, suggesting that for the prompt atomization process, the evaporation rates may be primarily controlled by the fuel volatility.

---

* Corresponding author: mcdonell@ucicl.uci.edu

The influence of non-ideal vapor-liquid-equilibrium on vaporization of multicomponent hydrocarbon fuels

A. Bader*, P. Keller, C. Hasse, B. Meyer
Department of Energy Process Engineering and Chemical Engineering,
Technical University of Freiberg, Germany
Andre.Bader@iec.tu-freiberg.de, Peter.Keller@iec.tu-freiberg.de,
Christian.Hasse@iec.tu-freiberg.de, Bernd.Meyer@iec.tu-freiberg.de

Abstract

In this work, the differences between non-ideal vapor-liquid-equilibria (VLE) and the effect on vaporization for multicomponent hydrocarbon fuels, which are representative for engines or gasifiers, are investigated. The VLE for a general multicomponent system is given by

$$\widetilde{y}_i \phi_i^v p = \gamma_i \widetilde{x}_i f_i^+$$

where $\widetilde{x}_i$ is the molar fraction of the liquid, $\widetilde{y}_i$ the molar fraction of the vapor, p the pressure of the system, ϕ_i^v the fugacity coefficients of the vapor, γ_i the activation coefficients of the liquid, and f_i^+ the fugacity of a selected reference state. For an ideal VLE the fugacity and the activation coefficients are equal to unity one ($\phi_i^v = \gamma_i = 1$) and fugacity is equal to the saturation pressure of the pure component. The equation for the ideal VLE is also called Raoult's law and most standard models, e.g. in CFD codes, for droplet vaporization use this relation. Here, the fugacity coefficients are determined using the binary Non-Random-Two-Liquids (NRTL) approach.

The VLE differences at 1.0 atm between Raoult and non-ideal approaches are illustrated for the binary mixture iso-octane/ethanol in Fig. 1. The non-ideal VLEs show an azeotrope point at around 0.4 kg/kg ethanol and lower boiling and condensation temperatures compared to Raoult's law. The software Aspen Plus is used to determined the non-ideal approaches. Figure 1 shows furthermore the in this work used NRTL-model and experimental data from Wen [1]. Iso-octane/ethanol mixtures are suitable to descripe engine fuels like E10. Significant differences between the Raoult and all the non-ideal approaches can be seen both for the condensation and the boiling curve. The VLE results are then applied for single droplet vaporization using the formulation of Law [2] modified for convective environments, which can be considered as base model for most CFD applications. The binary droplet mixture of iso-octane/ethanol results in a ternary VLE within e.g. the surrounding nitrogen gas atmosphere. The droplet life time and the vaporization rates of the single components and their mixtures, which corresponds among other to E10 and E90, are compared between ideal and non-ideal behavior. Based on the deviations in the VLE approach, a parametric study of non-ideal VLE-behavior for a pressure range between 0.5-20.0 bar, a gas temperature range of 0-$600\,^{\circ}$C and particle Reynolds number below break-up and atomization, which are typical vaporization conditions in IC engines and gasifiers, is performed. The observed differences suggest that non-ideal VLE have a significant influence on droplet vaporization of complex hydrocarbon mixtures under realistic conditions for technical systems.

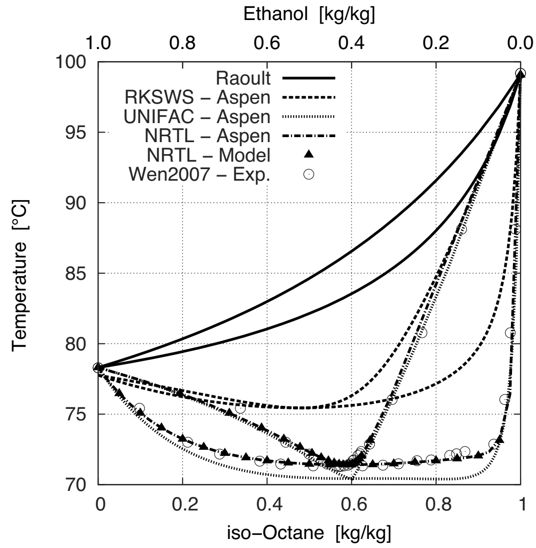


Figure 1. Ideal and non-ideal (real) VLE of iso-octane and ethanol at 1.0 atm

[1] Wen, C.C. and Tu. C. H., *Fluid Phase Equilibrium* Vol. 258, p. 131-139 (2007).
[2] Law, C.K., *Combustion and Flame* Vol. 26, p. 219-233 (1976).

*Corresponding author: Andre.Bader@iec.tu-freiberg.de

A Comparison of Jatropha Methyl Ester and Diesel Non-evaporating Sprays

Prasad Boggavarapu*, R V Ravikrishna
Department of Mechanical Engineering, Indian Institute of Science, Bangalore, India.
prasadb@mecheng.iisc.ernet.in and ravikris@mecheng.iisc.ernet.in

Abstract

Recently, bio-fuels are being considered as important alternatives for petroleum-fuels because of their renewability. In particular, jatropha, pongamia and rapeseed methyl esters are being actively considered as petroleum-diesel alternatives. Many researchers have studied and reported various effects of using biodiesels on spray, combustion and emissions with respect to compression ignition (CI) engines ranging from very broad engine level tests to fundamental studies on liquid properties, spray structure and its implications.

The present experimental study compares various spray characteristics of diesel and jatropha methyl ester (JME/ jatropha biodiesel) non-evaporating sprays. These sprays are studied in a specially fabricated high pressure chamber with optical access. The fuels were injected at injection pressures of 500, 1000 and 1500 bar into a nitrogen environment maintained at 20, 25 and 35 bar, respectively. First, the spray structure in terms of spray tip penetration and spray plume angle are compared. It is observed that the spray tip penetration is around 2 to 5% higher and the spray plume angle is around 7 to 17% smaller for JME as compared to diesel. This indicates a slower breakup for the jatropha biodiesel sprays. Droplet diameters are measured using the particle/droplet image analysis (PDIA) technique. For JME, around 5% higher droplet diameters are observed. Detailed probability distribution of droplets showed that jatropha biodiesel has more probability for larger droplet diameters (>18μm) and lesser probability for smaller droplet diameters (<18μm) which explains the overall SMD trend. The main reason for larger droplet diameters is the higher viscosity and surface tension of JME compared to diesel. The effect of fuel properties on the near nozzle structure is studied. A longer unbroken liquid length and narrower spray plume is observed for JME as compared to diesel indicating slower breakup.

* Corresponding author: prasadb@mecheng.iisc.ernet.in

Simulation of Biodiesel Jet in Cross Flow

E. Farvardin, A. Dolatabadi[1*]
Mechanical Engineering Department, Concordia University, Canada
e_farva@encs.concordia.ca and dolat@encs.concordia.ca

Abstract

Liquid jet in cross flow has many applications such as in gas turbine combustion chamber, ramjets, scramjets and agriculture industry. Furthermore, it can be served as a fundamental study of spray quality for different fuels as it illustrates jet disintegration, surface waves, primary and secondary breakups, effect of viscosity, effect of surface tension and the effect of density of the fuel on the final droplet size distribution, mixing, evaporation and combustion processes. In the last decade, scientists and industries have been attracted to the use of renewable energy such as biofuels. Biodiesel as an alternative for diesel, which is already in use by many industries such as transportation and energy, played a pioneer role in this direction. Moreover biodiesel has different renewable resources including Canola oil, cottonseed oil, animal fat, soybean oil, yellow grease and brown grease. Contrary to these advantages, the ignition, atomization and evaporation of biodiesel have been critiqued. In a recent comparative study by the authors, it has been shown experimentally that biodiesel has lower penetration depth comparing to diesel. The lower penetration depth is thought to be as a reason of primary breakup and jet column bending before the disintegration.

In this study, near field behaviour of a liquid jet in crossflow, i.e. deformation, primary breakup and penetration is investigated. Biodiesel, diesel and their blend are used as the liquid jets entering the cross flow of air. The primary breakup of biodiesel jet is simulated using the Volume of Fluid (VOF) numerical method. In order to, accurately, capture vortices including the horseshoe vortex in the gas and the liquid internal vortices, Large Eddy Simulation (LES) turbulence model is coupled with the VOF. The operating condition of this study is gas Weber number of 48 and 80 with a liquid to gas momentum ratio of 50 and 100. This work serves as a comparative study of biodiesel and diesel spray characteristics in cross flow.

The results show lower penetration depth of biodiesel jet comparing to the diesel jet. This behaviour is assumed to be due to the larger drag acting on biodiesel liquid columns which bends the jet more towards the downstream. This is in a manner that diesel jet owing to smaller drag penetrates more inside the crossflow of air before complete bending towards the cross flowing air. Several bag shapes are observed after the primary breakup of biodiesel at large Weber numbers, i.e. atomization Weber numbers, which argues the common classification of primary breakup regimes. This is in a manner that at the same Weber numbers, diesel show atomization mode of breakup. On the other hand, biodiesel droplets at downstream of the domain have lower velocities which can be as a result of having wider wakes surrounding the biodiesel column. Ultimately, the windward trajectory of VOF-LES simulation is compared with the DPM simulation results. Apparently, the results of VOF-LES seem to be in better agreement with the experimental shadowgraph results in comparison with the DPM trajectories.

* Corresponding author: dolat@encs.concordia.ca

Spray Characterization of Palm Olein/Diesel Blends under Various Injection Pressures

M.H.A.R Mantari*[1], Y.A. Eldrainy[1], M.N.M Jaafar[1], M.S.A.Ishak[1]
[1]Department of Aeronautical Engineering
Faculty of Mechanical Engineering
Universiti Teknologi Malaysia
81310 UTM Skudai, Johor

Abstract

The Sauter Mean Diameter (SMD) and spray cone angle are two important parameters that characterize spray performance. The objective of this study is to characterize palm olein/diesel blends spray in terms of spray angle and SMD under different injection pressures using a hollow cone pressure swirl atomizer. The physical properties of five diesel/palm olein blends, namely B5, B10, B15, B20 and B25 were measured and their spray characteristics were tested at injection pressures of 0.8 MPa, 1 MPa, and 1.2 MPa under ambient atmospheric condition. The results were compared to spray established using petroleum diesel fuel. The SMD was measured using a phase Doppler analyzer (PDA). The spray cone angle was visualized using a digital single-lens reflex (DSLR) camera. The results indicated that petroleum diesel fuel had the widest cone angle followed by B5, B10, B15, B20 and B25 under the same injection pressure. Additionally, when the injection pressure increases from 0.8 MPa to 12 MPa, the spray cone angle widen accordingly. It is concluded that high content of palm olein in the palm biofuel blends increases viscosity and surface tension and hence higher value of SMD and narrower spray cone angle was generated. An increase in injection pressure resulted in smaller droplet SMD and wider spray cone angle.

* Corresponding author: mhanafi22@live.utm.my

Evaluation and Validation of ELSA Model in Diesel Sprays: 3D Cavitating Nozzles Case

S. Hoyas[†], A. Gil[†], P. Fajardo[†], D. Khuong-Anh[†*], F. Ravet[‡]
[†] CMT-Motores Térmicos, Universitat Politècnica de València, Valencia 46022, Spain
serhocal@mot.upv.es, angime@mot.upv.es, pabfape@mot.upv.es, and ankh2@mot.upv.es
[‡] Renault, 1 Avenue du Golf 78288, Guyancourt, France
frederic.ravet@renault.com

Abstract

Computational Fluid Dynamic (CFD) techniques have become one of the main tools in the design and development of diesel engines. There exist, however, some drawbacks and problems that must be overcome in the next years. One of the challenges is to predict accurately the couple between the flow inside the nozzle and the spray, including the primary break-up and the secondary atomization. In the last years, several authors have been developed the Eulerian-Lagrangian Spray Atomization (ELSA) model. ELSA model combines an Eulerian and Lagrangian descriptions by coupling these two methods properly. ELSA model also accounts for the modeling of droplets and their formation process, particularly in the dense spray region. The ELSA model for diesel spray modeling has been recently implemented and developed into Star-CD CFD commercial code. Author's effort was focused on a detailed validation and evaluation of the fuel injection in diesel engines using this last implementation. Spray atomization, spray formation and macroscopic characteristics of diesel spray behavior were investigated. The overall work has been conducted in non-evaporative conditions. As cavitation greatly affects to spray behavior and it is though that cavitating nozzles will be present in most of close future engines, this sort of configuration has been chosen for validation. Velocity profiles at the section area of the nozzle exit obtained from trusted and experimentally validated RANS internal flow simulation were used. Results have been validated again experimental data, mostly coming from CMT-Motores Térmicos institute. It was found that the ELSA model reproduces accurately the experimental results.

---

* Corresponding author: ankh2@mot.upv.es

Investigation on Spray Characteristics of Water Emulsified Diesel with Different Injection Pressure and Ambient Temperature

Ming Huo1, Shenlun Lin2, Haifeng Liu3, Chia-fon F. Lee1.4*
1 Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, IL, 61801, USA
2 Department of Environmental Engineering, National Cheng Kung University, Tainan 70101, Taiwan
3 State Key Laboratory of Engines, Tianjin University, Tianjin, 300072, China
4 Center for Combustion Energy and State Key Laboratory of Automotive Safety and Energy, Tsinghua University, Beijing 100084, China
minghuo2@illinois.edu and cflee@illinois.edu

Abstract

The application of emulsified fuel in diesel engines has been proved to be beneficial as it reduced the exhaust emission of both nitrogen oxides and particulate matter. The enhanced atomization, often associated with the "micro explosion" phenomena, will lead to better fuel air mixing. In this study, the spray characteristics of water emulsified diesel fuel with different water content were experimentally investigated in a constant volume combustion chamber with different injection pressures and various ambient temperatures. The bubbles' size of the water phase in the fuel was first measured using microscope for all the prepared fuels and stability tests have been conducted to ensure no phase separation before measurement. The fuels were later injected and combusted in a constant volume chamber with optical access. The evolution of the entire injection was record by a high speed camera using Mie scattering. The images were processed to acquire the spray characteristics such as liquid penetration and cone angle, as such, the impacts of the ambient temperature and injection pressure on the spray performance were evaluated. It is shown that both W10 (10% water by volume) and W20 were featured with longer liquid penetration, especially under low ambient temperatures, which was attributed by the low volatility of the water. Notable increased cone angles and "fattened" main jet body were observed for emulsified fuel at the beginning stage of injection indicating the possible occurrence of micro-explosion.

* Corresponding author: cflee@illinois.edu

Visualization of Internal Flow and Spray Formation with Real Size Diesel Nozzle

T. Hayashi*[1], M. Suzuki[2], M. Ikemoto[3]
1: NIPPON SOKEN, INC., Nishio, Aichi Japan
2: DENSO CORPORATION, Kariya, Aichi Japan
3: TOYOTA MOTOR CORPORATION, Susono, Shizuoka Japan

Abstract
The internal flow of a diesel nozzle and its spray formation have been investigated by flow visualization with real-size transparent nozzle. The string-type and film-type cavitations are separately observed in the nozzle hole during the injection stage. Through analyzing the frequency of the cavitation generation and spray fluctuation, it is evident that with the increase of string-type cavitation, spray cone angle tends to be wide.

Introduction

Recently, many studies have focused on the internal flow in the injector nozzle and its spray formation. They indicate that the spray characteristics are markedly influenced by cavitation generated in the nozzle hole. This study established the flow visualization method in the real-size nozzle and investigated the relationship between the transient nozzle internal flow, cavitation, and spray formation. A mini-sac (MS) nozzle and valve-covered orifice (VCO) nozzle were surveyed to analyze the influence of the nozzle design toward the internal flow and spray characteristics.

Transparent nozzle

The tip of an actual injector nozzle was modified to provide optical access. The transparent nozzle is made of acrylic resin, which possesses a refractive index that is similar to that of the diesel fuel (Figure 1). In order to evaluate the internal flow and cavitation generated in the actual nozzle, the geometry of the sac and hole of the transparent portion were made to be the same as the original nozzle.

Results and Discussion

The string-type and film-type cavitations are separately observed in the nozzle hole. String-type and film-type cavitations both generated in the MS nozzle hole (Figure 2-a). Regarding the VCO nozzle, film-type cavitation on the nozzle hole inlet plays a more dominant role rather than the string-type cavitation, and the spray cone angle is narrower than that of the MS nozzle (Figure 2-b).

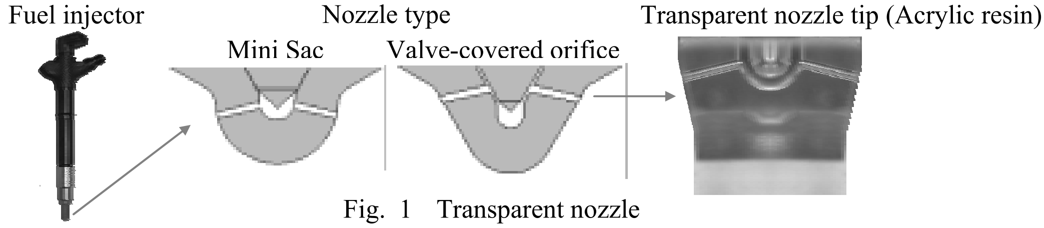
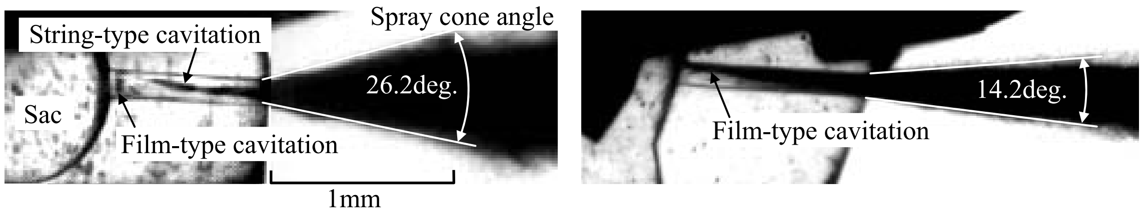


Fig. 1 Transparent nozzle



(a) 3-hole MS nozzle (b) 3-hole VCO nozzle

Fig. 2 Transient characteristics of cavitation and spray close to the nozzle hole outlet
(t_a=293K, P_a=1MPa, P_{inj}=50MPa, t_{inj}=1.0ms, 0.88ms ASOI)

* TOMOHIRO_HAYASHI@soken1.denso.co.jp

A visual experimental research on spray characteristics of blended fuel with a large proportion of soybean oil methyl ester (SME)

Qian Wang, Chengcheng ZHANG*, Hengsong JI, Aiping Shi
School of energy and power engineering, Jiangsu University, China
zcc_cool@126.com and qwang@ujs.edu.cn

Abstract

As a biodegradable and oxygen-bearing fuel, biodiesel is valued extensively for its similar physical and chemical properties to fossil fuel. In recent years, biodiesel alternative application in diesel engine has been the great concern of researchers at home and abroad. Nevertheless, some key technologies have not been solved at present. Basing on synthetic consideration of economic and environment protection, alternative research of biodiesel in diesel engine focus on blended fuel with a certain proportion of biodiesel (volume ratio≤25%). Aiming at SME, transient injection process of blended fuel with different proportions of SME and diesel was investigated in this paper. High speed photography was adopted to test the transient evolution of the spray cone angle and growth process of spray penetration. The axial velocity, radial velocity and the distribution of particle size were studied using a phase-Doppler particle anemometer (PDPA) as well. Based on the visualization methods, the leading factors of blended fuel atomization were deeply investigated, especially for the heating blended fuel with a large proportion of SME.

The study shows that axial velocity is the main factor to affect the spray performances, which remarkably influenced the spray penetration and was also the vital factor of radial velocity. And the sauter mean diameter (SMD) of SME was 25% larger than the one of 0# diesel. It indicates that spray quality will get deteriorative obviously when mixing ratio of SME goes beyond 50%, and thermal enhancement before injection can make B75 take on good atomization performances and B75 exhibited a similar atomization quality to 0# diesel with the fuel temperature of 473K, but the influence of axial velocity on spray would be weakened simultaneously.

---

*Corresponding author: zcc_cool@126.com

Effects of Pressure on the Fundamental Physics of Fuel Injection in Diesel Engines

J. C. Oefelein*, R. N. Dahms, G. Lacaze, J. L. Manin, L. M. Pickett
Combustion Research Facility, Sandia National Laboratories, Livermore CA, USA
oefelei@sandia.gov, rndahms@sandia.gov, gnlacaz@sandia.gov, jmanin@sandia.gov, lmpicke@sandia.gov

Abstract

Past works have suggested that two extremes exist with regard to liquid injection in high-pressure systems. At lower pressures, the classical situation exists where a well defined interface separates the injected liquid from ambient gases due to the presence of surface tension. Under these conditions, surface tension forces form a discontinuous non-continuum interface that promotes primary atomization, secondary breakup and the resultant spray phenomena that has been the well recognized and is widely assumed. At high-pressure conditions, however, the situation can become quite different. Under these conditions, a distinct gas-liquid interface may not exist. Effects of surface tension become diminished and the lack of these inter-molecular forces minimizes or eliminates the formation of drops and promote diffusion dominated mixing processes prior to atomization. In this paper, we present some of the first evidence that diffusion dominated mixing, not atomization, occurs at certain Diesel engine conditions. In addition, we derive a theoretical model that explains why and quantifies the change in the interfacial dynamics that leads to the transition between the classical non-continuum "jump" conditions associated with two-phase flows and the continuous gas-liquid interfacial diffusion layers. To facilitate the analysis, we apply the Large Eddy Simulation (LES) technique using a detailed real-fluid model that is designed to account for key high-pressure phenomena. We perform a series of calculations of key target experiments associated with the Engine Combustion Network (see www.sandia.gov/ECN): namely the “Baseline n-heptane” and “Spray-A (n-dodecane)” cases, which are designed to emulate conditions typically observed in Diesel engines. Calculations are performed by rigorously treating the experimental geometry, operating conditions and thermo-physical gas-liquid mixture properties. To further augment the analysis, the real-fluid framework used in the LES is then combined with vapor-liquid equilibrium and linear gradient theory to facilitate calculations of the detailed vapor-liquid interfacial structure associated with the multicompnent mixtures of interest. In applying this integrated theoretical framework, we focus on two particularly relevant reference conditions where the ambient gas is $p_1 = 29\ bar$, $T_1 = 440\ K$ and $p_2 = 60\ bar$, $T_2 = 900\ K$. We then compare the results with corresponding observations from high-speed imaging data. The high-temperature interface showed a substantially reduced surface tension and a wider interface thickness compared to the low-temperature interface. An applied Knudsen-number criterion then revealed a major finding. Contrary to conventional wisdom, gas-liquid interfacial diffusion layers develop not necessarily because of vanishing surface tension forces, but because of broadening vapor-liquid interfaces. These interfaces become so thick that they enter the continuum length scale regime. Thus, independent of any residual surface tension forces that might be present, the Navier-Stokes equations apply across the high-temperature vapor-liquid interface if the viscous stress term is modified appropriately. Similarly, continuum based diffusion laws apply across the vapor-liquid interface, producing a continuous phase transition not a “jump” condition. Our analysis suggests that at certain conditions in Diesel engines, the classical view of spray atomization as an appropriate model is questionable. Instead, nonideal real-fluid behavior must be taken into account using a multicomponent formulation that applies to arbitrary hydrocarbon mixtures at high-pressure supercritical conditions.

*Corresponding author: oefelei@sandia.gov

Tuesday

September 4, 2012

Contents

Twin Fluid Atomization

Alternative Atomizers I

Diagnostics: Imaging Techniques I

On Using Detailed Simulations to Study Primary Atomization

M. Herrmann*
School for Engineering of Matter, Transport and Energy
Arizona State University, Tempe, AZ 85287, USA
marcus.herrmann@asu.edu.edu

Abstract

Commonly, atomization is thought to occur in two consecutive steps: the initial primary atomization of liquid streams into large and small scale structures, followed by secondary atomization of these structures into ever smaller drops. While a number of established models exist for the latter process, the details of the former process are as of this day not fully understood. This is in part due to the fact that experimental access to the primary atomization region is at best extremely difficult under most conditions of relevance to technical applications. Detailed numerical simulations may help study the fundamental mechanisms of the initial breakup in regions and under operating conditions, where experimental access and analysis is too challenging. However, simulating atomization accurately is a tremendous numerical challenge since time and length scales vary over several orders of magnitude, the phase interface is a material discontinuity, and surface tension forces are singular.

In this talk some recently developed numerical techniques to simulate atomization will be discussed briefly. These include the refined level set grid method to describe the motion of immiscible interfaces, the finite volume balanced force method for unstructured polyhedra meshes to account for surface tension forces in a stable and accurate manner in complex geometries, the multi-scale Eulerian interface tracking/Lagrangian point particle coupling procedure to couple near injector detailed primary atomization simulations to far field secondary atomization Lagrangian spray simulations, numerical techniques for level set based formulations to solve the governing equations in the presence of large density ratios and high shear in a stable manner, and consistent level set based filter functions to derive novel subgrid models for phase interface dynamics.

However, to achieve a simulation tool that is predictive, special focus must be placed on both code and solution verification. A novel method of applying the Method of Manufactured Solutions to one-fluid formulations will be presented, demonstrating that even in the presence of discontinuous immiscible interfaces, finite volume methods for scalar equations are at least first-order accurate in the infinity norm. Several remaining challenges to achieve a truly predictive simulation tool for atomization will then be discussed. Finally, simulation results for the atomization of turbulent liquid jets injected into gaseous crossflows, a turbulent liquid jet injected into compressed still air, and a single drop subjected to a turbulent crossflow will be presented.

---

*Corresponding author: marcus.herrmann@asu.edu

Off Center Impact of Water Droplets on a Thin Horizontal Wire

L. Haim[1], I. Sher[2] and E. Sher[1,3 *]

[1]Department of Mechanical Engineering, Ben-Gurion University, Beer-Sheva, Israel
[2]School of Engineering, Cranfield University, Cranfield, Bedfordshire, United Kingdom
[3]Visiting Professor, Faculty of Aerospace Engineering, Technion - Israel Institute of Technology, Haifa, Israel

Abstract

The current work focuses on the initial stage of filtering droplets, and explores the influential factors of drops impacting on a thin horizontal dry wire. We investigate the effect of the impact velocity, wire thickness, initial drop size and liquid surface tension along with the impact eccentricity, i.e. the distance between the trajectory of the drop and the axis of the wire, on the amount of liquid trapped on a wire. A large number of experimental sets were performed in order to determine the amount of the liquid remains on a wire after a drop impact. In all cases, the initial velocity was higher than the capture velocity for the centered impact [2]. For each set, 100 droplets were allowed to impact the wire, while measurements were taken for the un-captured droplet fraction. The results were analyzed statistically.

It was found that for high impact velocities (1.36, 1.42 m/s), the amount of liquid that remains on the wire is minimal and is fairly constant. For low and medium velocities (0.46-1.25 m/s), the amount of liquid that remains on the wire increases at a critical eccentricity value (from 0.2 to 2 mg), and from there it decreases. As the velocity increases, the maximum amount of liquid captured on the wire decreases while the corresponding critical eccentricity increases. This behavior was first observed and explained by Lorenceau et al. [5]. For a centered impact, the droplet is divided into two independent fragments, where each volume is bigger than the critical capture volume [5]. These fragments do not remain on the wire, and are detached under the effect of gravity and inertia. Only a small fraction of liquid remains on the wire in this case, following coating theorems. As the eccentricity increases, less equal the two lobes become, till the volume of one of them get smaller than the critical volume, consequently, it remains on the wire under the effects of the capillarity and friction that keep it from falling. An additional increment of the eccentricity above the critical value results a decrease of the captured fragment, till it extinct at e~R.

Thickening the wire was found to increase the max amount of captured liquid because of the larger surface area. The max amount of captured liquid is also increased when increasing the initial drop radius, but the relative amount in this case is significantly decreases. At low surface tension, the max amount of captured liquid is dramatically decreases, though the small amount of coating liquid that remains on the wire before the critical eccentricity is actually increases in agreement with the coating features of surfactant solution.

These findings led to development of a criterion that characterizes the amount of liquid that can be captured by the wire. The criterion is based on a force balance and includes 4 non-dimensional (Re, We, Fr and the wire-drop radii ratio). The critical eccentricity and the max amount of captured liquid have been calculated and compared with the experimental observation, and good agreement has been obtained. Therefore, our equation provides a reliable criterion for capturing liquid by the wire.

References:

[1] E. Lorenceau, T. Senden and D. Quéré, *Molecular Gels part 2*, R. G. Weiss, P. Terech, 2006, p. 223-237.
[2] E. Lorenceau, C. Clanet and D. Quéré, *Journal of Colloid and Interface* Science 279: 192–197 (2004).
[3] L.S Hung, S.C. Yao, *International Journal of Multiphase Flow* 25: 1545-1559 (1999)
[4] M. Pasandideh-Fard, M. Bussmann, S. Chandra, J. Mostaghimi, *Atomization and Sprays* 11:397-414 (2001).
[5] E. Lorenceau, C. Clanet, D, Quéré and M. Vignes-Adler,*The European Physical Journal Special Topics*, 166: 3-6 (2009).
[6] Zar, Jerrold H. *Biostatistical Analysis,*T. Rym, 1999, p. 99, 208-222.

* Corresponding author: Sher@bgu.ac.il

Energy conversion during the splash

D. Kalantari*, C.Tropea**
*Dep. of Mech. Eng., University of Sari (SANRU), P.O.Box 578, Sari, Iran
**Institute of Fluid Mechanics and Aerodynamics, Center of Smart Interfaces, Technische Universität Darmstadt, Germany
dkalantari2000@gmail.com and ctropea@sla.tu-darmstadt.de

Abstract

In an overall effort to model the impact of liquid sprays onto rigid walls, the splashing phenomena plays an important role in determining the velocity and size distribution of ejected droplets from the wall as well as the ejected mass fraction. In practice, increasing the number of splashing droplets in spray impact phenomena can decrease the quality of coated or painted surfaces. During the splash, an uprising crown-like thin liquid sheet develops at the kinematics discontinuity position (a point between the spreading droplet and unperturbed wall film with very high velocity and film thickness gradient). This crown-like sheet is bounded with a free end rim due to the surface tension effect, which generates finger-like jets disintegrating into the secondary droplets. In this study an energy conservation approach is considered for estimating the maximum crown height during the crown development. The energy conservation links the total energy of the impacting droplet and splashing crown. Energy balance ($E_0 = E_{diss} + E_g + E_\sigma$) at the maximum crown height (H^*_{Cmax}) for $h_0{}^* \geq 0.25$ yields

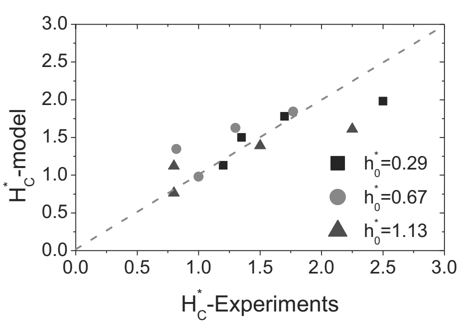
$$\sum_{n=0}^{6} A_n \left(H^*_{C_{max}}\right)^{\frac{n}{2}} = 0$$

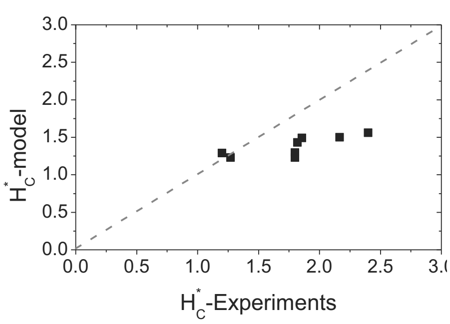
$$A_0 = -\left(\frac{We}{12}+1\right) \qquad A_1 = \sqrt{30}D_f^* \qquad A_2 = \frac{15}{4}\left(\frac{We}{Fr}\right)D_f^{*2} + \frac{3\pi}{2} \qquad A_3 = 11 \qquad A_4 = \frac{3}{4}\left(\frac{We}{\sqrt{\text{Re}}}\right)$$

$$A_5 = 0.55\left(\frac{We}{Fr}\right) \qquad A_6 = \frac{3\pi}{80}\left(\frac{We}{Fr}\right)$$

Which We, Re, and Fr are dimensionless impact parameters defined as: $We = \rho u^2 d_0/\sigma$; $Re = \rho u d_0/\mu$; $Fr = u^2/g d_0$, also $D_f = h_0(1 - h_1/h_0)$.

Numerical solution of the obtained theoretical results in the case of single isolated drops, indicate that the non-dimensional crown height increases nonlinearly with increasing the impact velocity. On the other hand, the non-dimensional crown height decreases slightly with the non-dimensional film thickness, corresponding to the wall film thickness varied in the range 0.25 $<h_0{}^* < 1$. Theoretical predictions properly estimate the maximum crown height in the case of a splash in isolation, i.e. single drop impact, whereas slightly underestimate in the case of a splash in spray impact conditions. Perhaps in the case of a spray impact, velocity fluctuations inside the wall film cases such differences. Results obtained in this study indicate that the maximum non-dimensional crown height increases linearly with the Weber number before the impact in spray impact phenomena.





Prediction of the maximum non-dimensional crown height estimated from theory as a function of the experimental measurements: splash in isolation (left), and splash in a spray(wright).

* Corresponding author: dkalantari2000@yahoo.com

Influence of Elevated Pressure on Impingement of a Droplet Upon a Hot Surface

Ilja Büchmüller*, Ilia V. Roisman§, Cameron Tropea§
Institute of Fluid Mechanics and Aerodynamics,
§ Center of Smart Interfaces,
Technische Universität Darmstadt, Germany
buchmueller@sla.tu-darmstadt.de, roisman@sla.tu-darmstadt.de, tropea@sla.tu-darmstadt.de

Abstract

Introduction

The conditions of drop impact at high wall temperatures and high ambient pressures are relevant for spray impact onto a wall inside internal combustion engines or in air-blast atomizers for gas turbines. The ambient pressure has a significant influence on the evaporation rate of an impacting drop because it changes the saturated vapor temperature of the fluid. Therefore, it affects the boiling temperature, the Leidenfrost temperature, the thickness of the vapor layer, and thus the dynamics, the outcome, the heat transfer rates and the parameters of the secondary spray at certain temperatures.

In the present investigation we consider a specialized and simplified case of a single spherical droplet of water descending a short distance in air and impinging onto a hot flat metal target. Evaporation during flight and subsequent condensation during evaporation of the droplet upon the target are neglected. Although simplified, this case is suitable for applications such as high power cooling.

Experimental method

This study focuses on the qualitative effect of elevated ambient pressure on water droplets with a diameter of 2.4 mm, which accelerate 30 mm, under the influence of gravity, and impact onto a heated surface. A flat aluminum target was heated up to 400 °C (673.15 K) in order to cover nucleate boiling, transition and rebound impact regimes at 1 bar. The experiment is placed in a pressure chamber filled with air pressures upto 24 bar. This is done to simulate conditions comparable to those of combustion chambers in modern engines. Observations of impact dynamics are made with a high-speed camera at 4000 fps and a high-speed LED stroboscope with flash duration of 300 to 400 ns.. Figure 1 shows the visualization examples of impacting droplets.

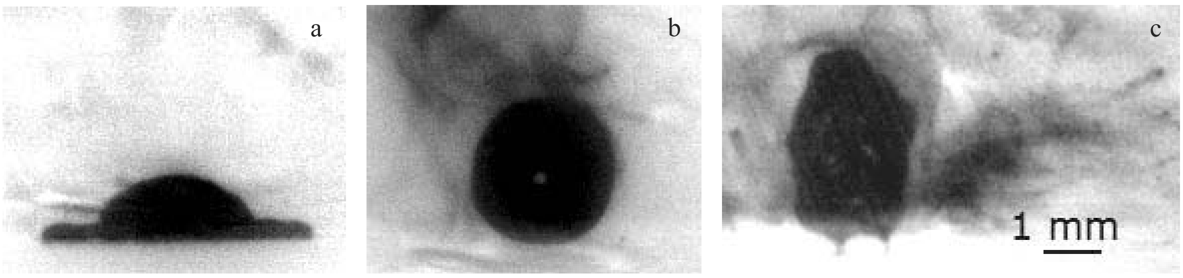


Figure 1. Impact of water droplets onto a heated plate: a: wetting; b: in-flight evaporation; c: satellite droplets.

Results and Discussion

The lifetime of a droplet changes according to both the surface temperature and the pressure in the impact regimes. The regime limits shift according to the saturated vapor pressure. The breakup is observed to generate accelerated secondary droplets in the transition boiling regime. A clear bouncing is observed, when the temperature of the target is above the Leidenfrost point. State-of-the-art boiling theory with nucleation site density for bulk liquids projected at the corresponding contact area explains the initiation of boiling in the droplet. Departure from nucleate boiling is not yet understood, a significant difference in the target superheat at different pressures, compared to a nucleation model, is observed.

Conclusions

The observed behavior of the evaporating droplets is dependent on the conditions upon the surface of the hot plate, especially in the transition region between the boiling and the Leidenfrost temperatures. Understanding of these phenomena provides the opportunity to design a heat exchanger interface equipped with coatings or structures to achieve enhanced performance.

* Corresponding author: buchmueller@sla.tu-darmstadt.de

Effect of injector nozzle flow number on injection evolution in a transparent diesel engine operating with pure RME fuel

E. Mancaruso[1*], B. M. Vaglieco
Istituto Motori - CNR, Italy
e.mancaruso@im.cnr.it and b.m.vaglieco@im.cnr.it

Abstract

The present paper describes an experimental investigation over the impact of diesel injector nozzle flow number on both injection and combustion evolution in transparent compression ignition engine equipped with modern Euro5 light-duty diesel engine head and the production common rail injection system. The research activity is devoted to understanding the basic operating behavior of low flow number nozzles, which are showing promising improvements in diesel engine behavior at partial load. In fact, because of the compelling need to push further emission, efficiency, combustion noise and power density capabilities of the last-generation diesel engines, the combination of high injection pressure fuel pumps and low flow number nozzles is popular among major OEMs. Therefore, aim of this paper is to provide a deeper understanding about the link between the nozzle flow number, the spray and mixture formation and the consequent combustion behavior for nozzle geometries and engine operating conditions that are typical of last-generation diesel engines operating with pure RME fuel. This will generate guidelines for the balanced nozzle flow number selection based on engine targets as well as will generate reference spray for upgrading 3D-CFD simulations models. Spray opening angle, break-up length and tip penetration are evaluated for three different nozzle flow numbers for a 2.0L diesel engine in various operating conditions. The results confirm that by reducing the flow number, for low injected quantities typical of low load and speed engine operating conditions, better fuel/air mixing improves the emissions/fuel economy trade-off.

* Corresponding author: Ezio Mancaruso, Istituto Motori-CNR, Via G. Marconi, 8 – 80125 Napoli
Tel-fax: +390817177187 - +390812396097 – email: e.mancaruso@im.cnr.it

Quantifying Statistical Measures of Diesel Spray Soot Characteristics using Laser-Induced Incandescence

Jaclyn E. Johnson*, Seung Hyun Yoon, Seong-Young Lee, and Jeffrey D. Naber
Michigan Technological University
Mechanical Engineering – Engineering Mechanics Department
Houghton, Michigan, USA

Abstract

Increased understanding is needed with respect to soot formation from high pressure diesel sprays as emissions standards become increasingly stringent and require complex methods for its reduction. Understanding soot formation and its spatial distributions is necessary to advance fundamental spray combustion knowledge. Given that the underlying nature of high pressure combusting diesel sprays results in turbulent mixing and combustion, significant variation in the location of soot and its structure is observed even when the conditions of the test are closely controlled. In this work, diesel spray soot characteristics are studied in a constant volume optically accessible combustion vessel through the application of laser-induced incandescence (LII). Studies are performed using a piezoelectric high pressure common rail injector with a high cetane (CN=56.5) diesel fuel at an injection pressure of 620 bar. Combusting sprays are examined at a part load charge-gas condition of 11.6 kg/m^3 density, 1300 K temperature, and in 21% oxygen and 15% oxygen (simulating 38% exhaust gas recirculation) environments. Images are acquired at 1.0 ms after start of injection in 21% oxygen and 1.5 ms after start of injection in 15% oxygen. Tests are repeated 14 times at each condition to provide a statistically significant sample. Images are compared, with average images and images of the ratio of the local standard deviation to the average, or local coefficient of variation, also considered to understand structure variations test to test. The results are quantified for total soot intensity and location of first soot. From this the required number of samples for a 95% confidence interval and 5% allowed error are determined.

*Corresponding author: jenesbit@mtu.edu

Detailed Laser based investigations of the dynamic of spray combustion inside a multipoint injection system

L. Zimmer*, T. Providakis, P. Scouflaire, S. Ducruix

CNRS, UPR 288 "Laboratoire d'Energetique moleculaire et macroscopique, combustion"
Grande Voie des Vignes, 92295 Chatenay-Malabry
Ecole Centrale Paris, Grande Voie des Vignes, 92295 Chatenay-Malabry

laurent.zimmer@ecp.fr, theodore.providakis@ecp.fr, philippe.scouflaire@ecp.fr and sebastien.ducruix@ecp.fr

Abstract

In order to reduce pollutant emissions from aircraft engines, enhanced mixing and reduced combustion temperature using partially premixed flames is a promising technology. However, this may lead to combustion instabilities, such as blow-off or strong coupling with pressure oscillations. To study the strategies of injecting fuels through different stages, a laboratory scaled combustor has been developed and runs at atmospheric pressure with dodecane as fuel. The fuel is injected through two, co-swirling stages. The first, using full cone, pressurized nozzle with 20% of air massflow rate is called the pilot stage. The second stage consists of a swirler in which 80% of the air flows and fuel is injected through 10 equally spaced holes of 0.3 mm diameter. Air is preheated at 473K and a typical air massflow rate of 53 g/s is used for the experiments. A staging parameter α is defined as being the ratio of the mass of fuel injected through the pilot stage to the overall mass of fuel. In this paper, only one value of the staging parameter (α =60%) is deeply analyzed, even tough a different values of α (20%) is used to illustrate the different behavior observed. Different laser diagnostics are applied in both non-reacting and reacting conditions to understand the way the spray and the flame interact. A time resolved PIV measurement system is used to retrieve time resolved planar droplet velocities at 10 kHz. Furthermore, flame front position through OH-PLIF, at a lower frequency is used. Analysis of the results show appearance of a strong aerodynamic structure around frequencies of the order of 2500 Hz that is linked to the precessing vortex core (PVC). This structure still exists in reacting conditions and in some cases is strengthened when the flame is stabilized by the pilot. Reacting cases show also a peak at lower frequency (300Hz) associated to quarter wavelength of the combustion chamber. Two different flame structures are found : pilot or rim stabilized. This leads to the existence of a hysteresis phenomena with two different flame structure possible, despite having identical injection condition. Detailed analysis shows that for the rim stabilized flame, the main reaction front lies in the inner part of the chamber whereas the spray is in the outer part. For the flame stabilized by the inner recirculation zone, the main reaction front lies in the outer part of the spray in droplets stay in the inner side. This leads to different delays for evaporation and therefore different coupling with the acoustic mode of the combustion chamber. Furthermore, for one case, the PVC is strengthened, leading to a very stable flame whereas it is weakened in the second case, letting the flame be driven then by acoustic coupling.

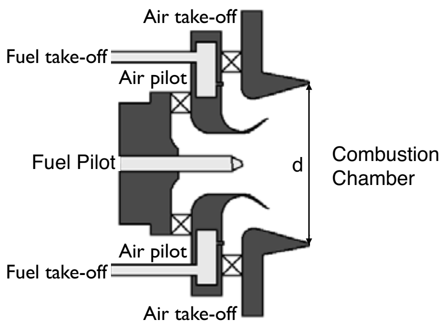


Figure 1. Schematic view of the injection device.

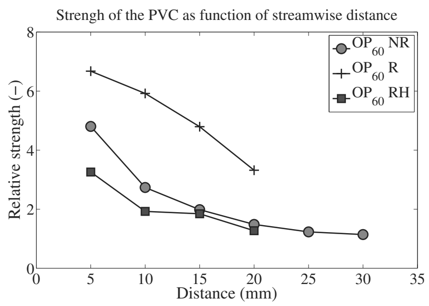


Figure 2. Evolution of the PVC as function of experimental condition.

*Corresponding author: laurent.zimmer@ecp.fr

Modeling, simulation and experimental verification of the trajectory and break-up of a particle laden spiraling liquid jet

S. Gramlich*, M. Piesche
Institute of Mechanical Process Engineering, University of Stuttgart, Germany
gramlich@imvt.uni-stuttgart.de and piesche@imvt.uni-stutgart.de

Abstract

A physical-mathematical model is presented which is able to predict the drop size, that results from the break-up of a particle laden rotating liquid jet with laminar flow. The model is based on the perturbation theory. Thus the flow is decomposed into a time steady motion and the motion of small time dependent disturbances.

The equations of motion for the time steady flow are derived by means of a force balance on an infinitesimal jet segment and result in the contour and the spiraling trajectory of the jet. The contour contracts due to the centrifugal force and thus the time averaged velocity increases along the jet. The suspension is treated as a liquid with shear thinning fluid flow properties with two plateau viscosities, which can be expressed by the Carreau-Yasuda model. The drag force due to the relative motion of the surrounding gas phase is considered. The necessary drag coefficient depends on both the Reynolds number of the relative gas flow and its angle of incidence and is determined by computational fluid dynamics. The resulting equation of motion in axial direction results in an ordinary differential equation of second order with only one boundary condition known. The solution method for this type of equation is developed using a "backward shooting"-method with an approximate solution serving as the second boundary condition.

The small transient perturbations of the jet contour are treated with a linear stability analysis, which results in a critical wave number that leads to break-up of the jet into droplets. The description of the solid and liquid motion is carried out in Eulerian formulation. Hence each phase is treated as a continuum with separated balances for mass and impulse. The interaction between the particulate solid and the liquid phase are included by additional terms in the impulse balances.

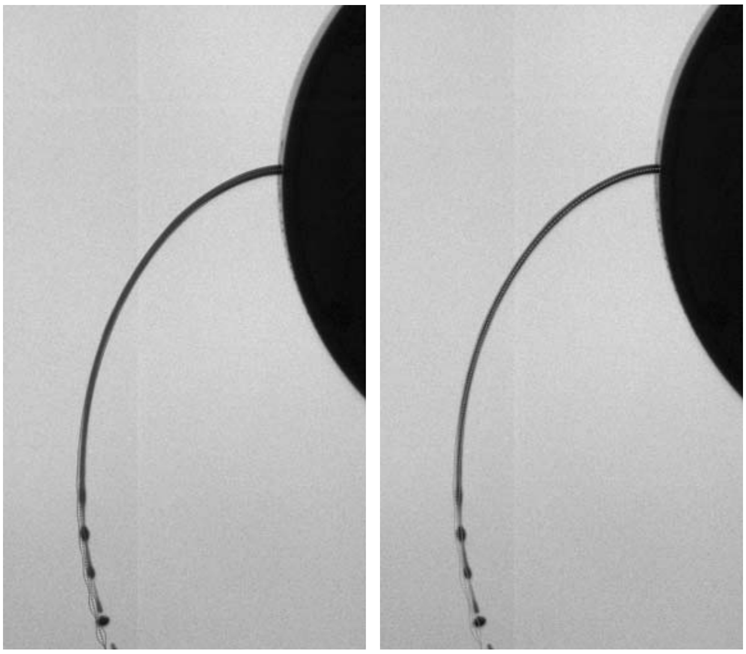


Figure 1. Comparison of calulated (coloured) and experimental jet contours from temporal (left) and spatial (right) stability analysis.

The perturbation method leads to the dispersion relation, which can be solved with regard to a temporal absolute or a spatial convective instability. Both solutions are compared to the experimental results gained by shadow imaging technology. The shadow images are evaluated with regard to the drop size and the break-up length of the jet, which serves the model as a boundary condition. Figure 1 shows a comparison of the calculated jet contour of the temporal stability analysis (left) and the spatial stability analysis (right) as an overlay over the experimental result. The spatial stability analysis describes the perturbations of the jet surface more accurate than the temporal stability analysis.

Based on an extensive parametric study the centrifugal acceleration, the surface tension, the viscosity as well as inertia were found to have a major effect on the drop size resulting from the break-up of a rotating liquid jet with laminar flow. The solid phase does not show an influence on the drop size. Neither in the time steady motion nor in the motion of the small disturbances significant changes due to different particle concentrations, particle sizes or particle densities are observed. Experimental findings support this result.

*Corresponding author: gramlich@imvt.uni-stuttgart.de

Experimental Investigation of Dynamics and Atomization of a Liquid Film Flowing over a Spinning Disk

M. Freystein[1], T. Gambaryan-Roisman[1,2,3] , P. Stephan[1,2]
[1]Institute of Technical Thermodynamics, Technische Universität Darmstadt, Darmstadt, Germany
[2]Center of Smart Interfaces, Technische Universität Darmstadt, Darmstadt, Germany
freystein@ttd.tu-darmstadt.de, gtatiana@ttd.tu-darmstadt.de, pstephan@ttd.tu-darmstadt.de

Abstract

Many intermediate and final products in chemical, pharmaceutical and food industry as well as in material processing are in the form of powder. Rotary atomizers are very common in the production of powder due to the possibility to create a spray with a narrow drop size distribution. In this technique a thin liquid film is formed and spreads radially over a spinning disk. In most of the cases the film is wavy. As the film flows over the disk edge, it disintegrates into droplets forming a spray. The influence of the liquid film hydrodynamics on the spray characteristics is not yet understood.

We have developed an experimental method for high-speed measurements of film dynamics and drop size distribution on a spinning stainless steel disk with a diameter of $d = 0.15$ m. Experiments are carried out in a wide range of mass flow rates and rotational speeds ($\dot{m} = 10 - 250$ kg/h, $f = 0 - 25$ Hz). The liquid jet is impinging at the center of the rotating disk. The local and temporal film thickness has been measured at different radial positions using a confocal chromatic sensoring (CHR) technique, which is based on chromatic longitudinal aberration of special optical probe. The sensor has been positioned above the disk and the radial position has been varied from $r = 2.5 \cdot 10^{-2}$m to the edge of the disk. The film thickness has been measured with a frequency of 4 kHz. The wave formation and the film development have been additionally captured using high speed video imaging. The drop formation at the disk edge and the drop size diameter have been observed by the shadowgraphy technique using an additional high speed camera.

The film flow on the rotating disk has been investigated in a wide range of parameters. The strongly wavy structure of the film flow has been observed for all sets of parameters. The development of waves depends on the nozzle-to-disk distance. At low mass flow rates the film thickness continuously decreases with increasing distance from the disk center, which indicates that the flow is dominated by viscosity and centrifugal force. At higher mass flow rates a maximum of the film thickness can be observed on the film thickness profile.

The comparison with correlations from literature shows a quantitatively good agreement for large dimensionless radii but only a qualitative good agreement for moderate values. First experimental results on the drop size distributions show the development of a bimodal distribution for small mass flow rates of the impinging liquid jet within the experimental range of parameters.

---

[3]Corresponding Author: gtatiana@ttd.tu-darmstadt.de

Laser Sheet Dropsizing of DI Sprays under Various Conditions Calibrated Using PDI

Zhenkan Wang, Min Xu*, Wei Zeng, Yuyin Zhang
School of Mechanical Engineering, Shanghai Jiao Tong University, National Engineering Laboratory for Automotive Electronic Control Technology, Shanghai 200240, China
David L. S. Hung
University of Michigan - Shanghai Jiao Tong University Joint Institute, Shanghai Jiao Tong University, Shanghai 200240, China
k_k@sjtu.edu.cn; mxu@sjtu.edu.cn; weizeng@sjtu.edu.cn; yuyinzhang@sjtu.edu.cn; dhung@sjtu.edu.cn

Abstract

It is time consuming for phase Doppler interferometry (PDI) to get the diameter distribution of a plane of the spray though it is quite precise. Laser sheet dropsizing (LSD) is a new laser technique which is based on a planar laser-induced fluorescence (PLIF) and Mie-scattering images originated from a cloud of droplets in a spray. LSD technique could obtain the Sauter mean diameter (SMD) across the spray region simultaneously and quickly by the ratio of the LIF signal to the Mie scattering signal. However, the precision of LSD technique is highly dependent on the calibration. The objective of this paper is to combine these two techniques so as to verify the accuracy of the calibration coefficient K of LSD technique in a wide range of injection pressures and fuel temperatures. Since the LSD technique produces a spatial distribution result while the PDI generates a single-point measurement result with temporal resolution, two comparison methods for the calibration of the coefficient K were conducted. Data conversion between the drop size results of these two measurement techniques were implemented in this paper. The drop size results of PDI and LSD techniques were in a good agreement. After the calibration, the spray SMD distribution can be thoroughly investigated by LSD technique with good temporal and spatial resolutions.

* Corresponding author: Prof. Min Xu, mxu@sjtu.edu.cn

Experimental Investigation on Spray Characteristics of Pressure-Swirl Atomizers for a Small-Sized Jet Engine

L. Durdina*, J. Jedelsky and M. Jicha
Faculty of Mechanical Engineering, Brno University of Technology, Czech Republic
y107165@stud.fme.vutbr.cz, jedelsky@fme.vutbr.cz and jicha@fme.vutbr.cz

Abstract

Within a design update of a small-sized jet engine combustor the original spill-return pressure-swirl atomizer is intended to be replaced with a newly designed low flow rate simplex atomizer. Objective of this study was to investigate the differences in spray morphology and atomization quality of both atomizers and the possible impact on the combustion process. Both atomizers were tested on a recently designed cold flow test rig including mobile fuel supply system with PC-based acquisition of flow rate, gauge pressure, liquid density and temperature readings in several regimes over the gauge pressure range from 150 kPa up to 1 MPa based on the typical operating conditions of the engine, from start and idle to maximum power regime.

Spray characteristics were studied by means of two established laser measurement techniques in fluid mechanics – Phase/Doppler Particle Analyzer (P/DPA) and Particle Image Velocimetry (PIV). These techniques complement each other; PIV gives instantaneous velocity distribution in the whole flow field and P/DPA gives both the droplet velocity and droplet size simultaneously in selected measurement points.

A 1-D P/DPA system was used for characterization of the atomizers in terms of their liquid distribution and droplet velocity. Data were acquired in two different planes normal to the atomizer axis in 25 mm and 50 mm distance downstream of the discharge orifice.

As for PIV, measurements were carried out in an axial section of the spray yielding 2-C and 3-C velocity vector fields. In case of the SPIV measurements, cameras were placed on the same side of the light sheet with 80° stereoscopic viewing angle between the lens axes.

Our measurements revealed that the new atomizer generates spray that differs from the original atomizer in terms of atomization quality and spray morphology. The new nozzle also showed different response to changes in gauge pressure, which may mean different demands on power regulation of the jet engine. Spray pulsations of the old atomizer have been observed in all operating regimes by fluctuating cone angle. The new atomizer was much more stable without any visible fluctuations in cone angle. The spray droplets generated by the new nozzle have lower Sauter mean diameter and lower droplet velocities at higher pressure regimes (Figure 2). This difference results from the different internal geometry of the new nozzle. Results of the SPIV measurements allowed determining the magnitude of the tangential velocity component.

Investigated differences will influence the behaviour of the nozzle in the combustion chamber, affecting mainly the evaporation, heat release and flue gas emissions. This study provides an extensive database for validation of numerical models of the tested nozzles.

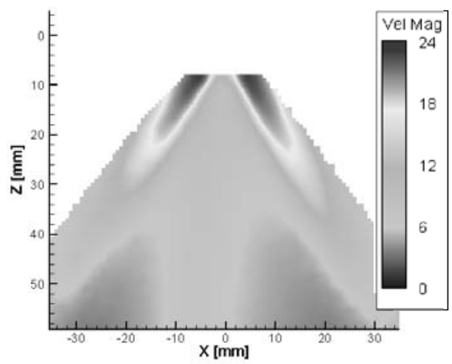


Figure 1 Velocity magnitude of droplets generated by the original spill-return nozzle at 690 kPa inlet pressure.

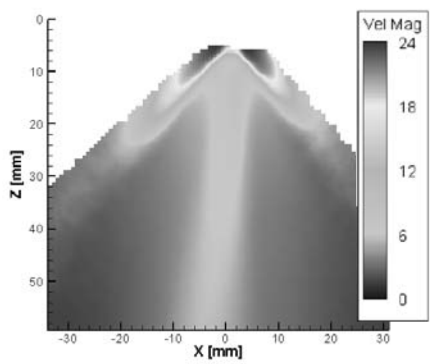


Figure 2 Velocity magnitude of droplets generated by the newly designed simplex nozzle at 690 kPa inlet pressure.

* Corresponding author: y107165@stud.fme.vutbr.cz

Experimental and Numerical Investigation of a Pressure Swirl Atomizer

B. Sumer*, N. Erkan*, O. Uzol+, I.H.Tuncer+
* Defense Industries Research and Development Institute, Scientific and Technological Research Council of Turkey (TÜBİTAK-SAGE), Ankara, Turkey.
+Department of Aerospace Engineering, Middle East Technical University, Ankara, Turkey.
bulent.sumer@tubitak.gov.tr, nejdeterkan@gmail.com, uzol@metu.edu.tr, tuncer@ae.metu.edu.tr

Abstract

The flow structure inside a pressure swirl atomizer is investigated using high-speed shadowgraphy techniques and computational fluid dynamics tools. The hollow cone spray properties are detected using Phase Doppler Particle Analyzer. The experimental and numerical results are analyzed and compared. The aircore inside the pressure swirl atomizer is visualized at high temporal and spatial resolutions with the high-speed shadowgraphy system. The images captured are analyzed quantitatively with a developed image processing tool. The analyses reveal strong fluctuations of the aircore diameter. Three dimensional and two dimensional axisymmetric-swirl models are used for the numerical study performed with ANSYS-FLUENT software. Unsteady, two phase laminar flows are computed using the volume of fluid method. Three-dimensional and two-dimensional numerical simulations also predict strong oscillations of aircore diameter near the base of swirl chamber. In addition, a two-component Phase Doppler Particle Analyzer is employed to investigate the spray properties.

* Corresponding author: bulent.sumer@tubitak.gov.tr

Improving Swirl Atomizer Performance with Coanda-Deflection Outlets

E. Musemic*, P. Walzel
Mechanical Process engineering, Technische Universität Dortmund, Germany
emir.musemic@bci.tu-dortmund.de and peter.walzel@bci.tu-dortmund.de

Abstract

In the present work, the application of the Coanda-effect on the sheet atomization at swirl atomizers is described. It can be used to reduce the mean drop size by increasing the spray angle above its limitation given by the swirl paramter. With swirl atomizers liquids can be sprayed with fairly low pressures. Commonly, they operate in the regime of sheet formation, leading to relatively small drop sizes compared to the nozzle orifice diameter. Reasonably narrow drop size distribution can be achieved. The drop sizes strongly depend on the initial liquid sheet thickness at the nozzle outlet, the sheet velocity and the sheet thinning effect due to the divergent spray propagation. It generally known, see e.g. [1], that a reduction of the sheet thickness and a spray angle increase can be achieved through an intensified swirl flow by modifying the nozzle geometry. Unfortunately the higher tangential velocities within the nozzle lead to higher friction losses, especially for viscous liquids, reducing the sheet velocity and thus increasing the mean drop size. It can be shown, that for each liquid viscosity and pressure, an optimum for the swirl intensity exists where the mean drop size reaches its lowest value.

A deflection outlet mounted below the orifice allows for enlargement of the spray angle up to $\Theta = 180°$ by the Coanda-effect as explained in [2]. The Coanda-outlet is a trumpet shaped outlet, like illustrated in the figure below, deflecting the sheet on a smaller radius R_C compared to the natural hyperboloid osculation radius R_H. The sheet then detaches from the trumpet at a sharp detachment edge. The advantage of the Coanda-design over the conventional sharp edged orifice is the higher energy efficiency, which allows for generating the same mean drop size with noticeable smaller liquid pressures. Also, the swirl atomizer application area is extended toward higher liquid viscosities, which otherwise could only be sprayed with disproportional high atomization pressures or using alternative atomizer designs. When Coanda-deflection outlets are considered the sheet velocity slowing down due to the additional contact area to the nozzle wall has to be analyzed. Choosing smaller deflection radii can reduce the contact area. However, reducing the ratio of the deflection radius to the sheet thickness $\delta_{0,C}$ below a certain value may lead to a premature film detachment and disintegration, which is usually unwanted.

The aim of the present work is to identify optimized Coanda-deflection geometries depending on of the swirl flow intensity, the liquid properties and the atomization pressure. In addition the influence of the ambient gas pressure on the sheet detaching behavior, which is quite unexplored so far, is analyzed.

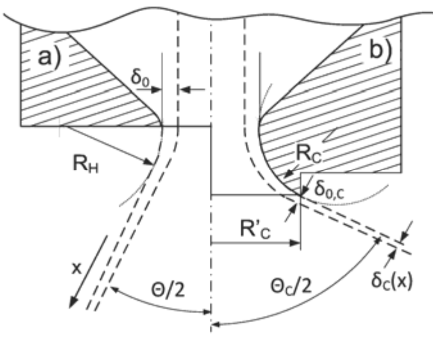


Figure:

Sheet detachment at the swirl atomizer orifice a) without - and b) with Coanda-deflection, R_H = hyperboloid osculation radius, R_C = Coanda-deflection radius, R'_C = Coanda detachment radius, Θ = Spray angle, Θ_C = Spray angle with Coanda-deflection, δ_0 = initial sheet thickness, $\delta_{0,C}$ = initial sheet thickness with Coanda-deflection, x = coordinate along the main sheet trajectory

[1] Lefebvre, A.H., *Atomization and Sprays*, Taylor & Francis, 1989

[2] Kistler, S.F., L.E. Scriven, The Teapot Effect: Sheet forming Flows with Deflection, Wetting and Hysteresis, *J. Fluid Mech.*, 263:19-62, 1994

* Corresponding author: emir.musemic@bci.tu-dortmund.de

Basic Preexaminations of Inline Measurements of Droplet Size Distributions by Statistical Extinction Method

F. Dannigkeit*, L. Steinke, S. Ripperger
Chair of Mechanical Process Engineering, University of Kaiserslautern, Germany
florian.dannigkeit@mv.uni-kl.de

There is a need for inline sensors for the monitoring of process sprays, which are able to measure the size of droplets or solids, their concentration, and their spatial distribution inline in process plants. The knowledge of these characteristic spray values are of particular importance in many spray processes, e.g. drying processes. The mass transport and thermodynamic characteristics of the droplet phase are depend to a large extend on the droplet size. These characteristic spray values define the proceeding processes in process sprays. Optical methods for particle analysis work contactless and almost do not influence the running processes.

Measurement techniques which are able to monitor process sprays inline in process plants provide a better inline control of drying processes and may improve the understanding of spray drying.
An inline process control allows improving the quality of drying products and facilitates a reduction of degraded material and costs. Therefore, only temporal modifications of the process sprays have to be detected. Thus, there is a need for easy measurement devices which can measure a mean droplet size and a droplet concentration reliably for a long term of time.
A better understanding of drying processes may simplify the design of new process plants for drying processes. Therefore, especially the knowledge of the relationship between characteristic spray parameters (droplet size distribution, droplet concentration, propagation speed) and the generated particles (size distribution, structure) is necessary. Thus, a measurement device which provides information about the characteristic spray parameters and the generated particles at different positions in drying processes is necessary.

Based on statistical extinction method two different measurement devices have been developed. A basic system determines the mean droplet size and the droplet concentration of sprays. Therefore a collimated laser beam with a diameter of about 3 mm crosses the spray. A pinhole with a diameter of 600 µm positioned behind the measuring volume limits the measurement beam cross section to a well defined value. A photodiode detects the fluctuating luminous intensity signal, which is extincted by the droplets of the spray. The transmission of light and its root mean square deviation can be calculated from these values. The statistical extinction method uses the transmission and their the root mean square deviation to calculate the mean droplet size. Bigger droplets which are entering or leaving the measurement volume cause larger root mean square deviations than smaller droplets. The droplet concentration is calculated from the mean value of the transmission and the mean droplet size.
A big advantage of this measuring principle is the simple optical construction which allows a lance-shaped realization of the sensor for the insertion in spray towers. For the measurement in one plane, only one incision with a diameter of about 50 mm in the outer wall of the tower is necessary to insert the lance into the tower. Measurements at very high droplet concentrations and at different positions in the spray tower are possible, if axially displaceable sheaths are used, which shield the laser beam from spray influence for a variable length. This is neither possible with scattered light measuring systems, nor with imaging techniques.
A second measurement system has been constructed to examine if the statistical extinction method is qualified for measuring droplet size distributions of sprays. A collimated laser beam with a diameter of 20 mm crosses the measuring volume. A pinhole with nine holes with different diameters from 10 µm up to 1000 µm is positioned behind the measuring volume. It generates nine laser beams with nine different cross sections. Behind each pinhole a photodiode is positioned which measures the extincted luminous intensity signal. The additional information is caused by the fact that droplets whose diameter is larger than the diameter of one laser beams cannot extinct a cross section which is larger than the laser beam. Thus, these droplets are measured too small. In a larger laser beam these droplets can extinct a cross section which is larger.

In the paper the statistical extinction method and the extension of this measurement principle to measure droplet size distributions are discussed. The construction of both measurement devices and some measurement results are displayed. The ability of the statistical extinction method is discussed by comparing droplet size distributions measured with a laser diffraction system with measurement results of the new sensor.

* Corresponding author: florian.dannigkeit@mv.uni-kl.de

Simultaneous observation of the scattered light in the rainbow region of two falling droplets

N. Roth*, G. Gerhardinger, A. Mariani and B. Weigand
Institut für Thermodynamik der Luft- und Raumfahrt, University of Stuttgart, Germany
norbert.roth@uni-stuttgart.de

Abstract

The droplets are falling within a observation chamber and while they are falling they are illuminated by a vertical laser beam, as shown in Fig. 1. The light scattered in the forward hemisphere is used to determine the droplet size using the interference pattern. The light scattered in the backward hemisphere in the region of the first rainbow is used to obtain the refractive index. The two lenses in front of the two cameras are not used as Fourier lenses anymore. The droplets are first imaged by the lenses and then the image plane of the cameras is moved along the optical axis in order to obtain defocussed images of the droplet. Then interference fringes in the forward direction and the first rainbow in the backward direction become visible. For the forward direction this is a well known method used in the so called ILIDS or IMI technique, which allows to determine the droplet sizes and velocities in a plane of a multiphase flow field. Here, in addition to the scattered light in the forward direction, the position of the rainbows from the light scattered in the backward direction is determined for two droplets of slightly different terminal velocity falling simultaneously within the observation chamber. Examples for two droplets of different substances are shown. In one case the distance between the droplets increases and in the other case a collision of the two droplets is observed.

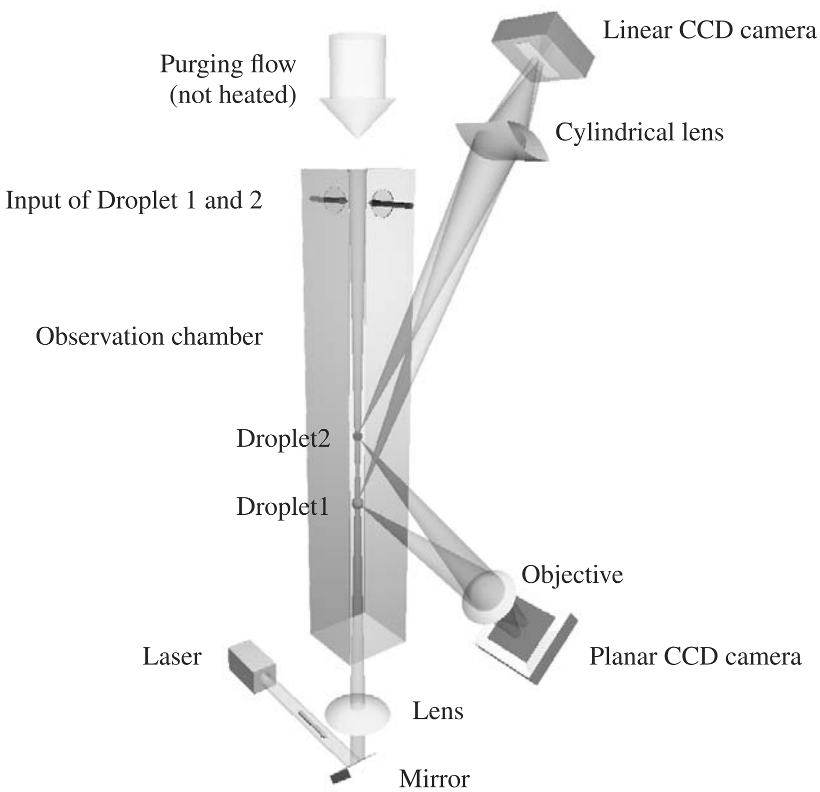


Figure 1: Schematic view of the experimental setup

*Corresponding author: norbert.roth@uni-stuttgart.de

Spray Droplet Size and Velocity Measurement using Light-field Velocimetry

T. Nonn, V. Jaunet, S. Hellman
Dantec Dynamics A/S, Skovlunde, Denmark
thomas.nonn@dantecdynamics.com

Abstract

The past decade has seen considerable activity regarding light-field (LF) cameras and their application to photography. The ability to refocus images over a finite depth range has consequently led to interest in measuring the distance of objects from the camera. While originally developed for imaging structured opaque surfaces, interest has led to successful application in volumetric flow-fields using a commercially developed light-field camera. This paper focuses on the measurement of droplets in a spray using a light-field camera. While previous measurement techniques utilized a limiting 2D shadowing technique, measuring the size and velocity of droplets silhouetted from behind with uniform illumination, light-field measurements open the possibility of accessing the depth position, and thus the 3D position and velocity of individual droplets in a well-defined volume.

The general principle behind light-field camera can be described as follows: a micro-lens array mounted directly onto an image sensor, with the same f-stop as the main objective, reconstructs at selected positions virtual images representing focus planes within the measurement volume. Particles appear in micro-images within several micro-lenses, albeit at slightly different perspectives, thus allowing for depth determination. Essentially, the microlenses act as local stereo systems. The downside of light-field cameras, however, is the loss of spatial resolution. The loss, however, can be mitigated by applying multiple-focus micro-lens techniques used by a commercially available plenoptics camera. In such cases roughly 25 percent of the spatial resolution is retained, thus the 11 MP camera used in this study had an effective resolution of 2.6 MP.

Measurements were conducted on a agricultural nozzle using water. Images were recorded for the entire spray and for a specific region-of-interest (ROI) within the fully developed region. Comparison measurements were made using a phase-Doppler (PDA). Time-averaged results were compared between PDA and LF, showing good agreement. In general, the light-field camera simplifies the measurement and calibration process for 3D flow analysis. Another added value of LF over the shadow measurement technique is in resolving out-of-focus particles, a persistent problem for conventional imaging. The investigation also focused on the uncertainties involved with LF measurement. This work is still considered preliminary. However, these early results show promise and point to the practical usefulness of light-field technology in 3D measurement.

Numerical Simulation of Evaporating Sprays in a Convective Flow Field

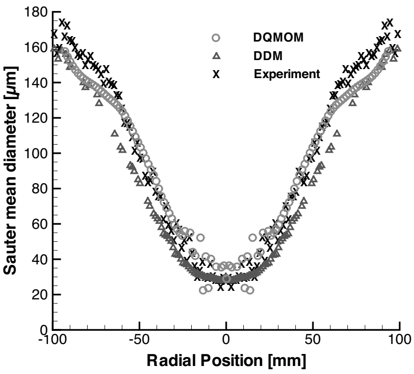
R. M. Humza*, S. R. Gopireddy and E. Gutheil
Interdisciplinary Center for Scientific Computing, University of Heidelberg, Germany
rana.humza@iwr.uni-heidelberg.de, srikanth.reddyg@iwr.uni-heidelberg.de and gutheil@iwr.uni-heidelberg.de

Abstract

In this paper, the direct quadrature method of moments (DQMOM) and the discrete droplet model (DDM) in an axisymmetric, two-dimensional configuration are used to model an evaporating water spray carried by nitrogen, the spray is injected into a vertical spray chamber. The models include the Abramzon and Sirignano model for convective droplet evaporation, and droplet motion is included by considering droplet drag and gravity. In DDM, the effects of the two-phase flow are captured by solving the gas phase conservation equations considering the droplets as point sources. DQMOM considers the inlet gas flow properties to compute the drag force exerted on droplet velocity. The phenomena of droplet – droplet interactions are currently neglected as the liquid volume in the present case is small. Appropriate initial and boundary conditions as well as the starting values for simulations are generated from experimental data, which have been carried out by the group of Prof. G. Brenn at TU Graz, Austria. The measurements were performed with phase Doppler anemometry (PDA). The experiment gives the spray characteristics (droplet size and velocity) at different cross sections away from nozzle exit. The DQMOM and DDM simulation results are compared with experimental data at these cross sections, and very good agreement with experiment is observed.

Figure 1 shows the computed and experimental profiles of the Sauter mean diameter at cross sections 0.12 m (left) and 0.16 m (right) away from nozzle exit. The DDM simulation result matches quite well the experiment at the center of the spray at 0.12 m from nozzle exit, but slightly under-predicts towards the periphery of the spray. Good agreement is observed at 0.16 m cross section between DDM and experiment. The DQMOM simulation results are in nice agreement with experiment at 0.12 m downstream the nozzle exit, and it is closer to the experimental data at higher radial distance. Further downstream, at 0.16 m from the nozzle exit, the DQMOM simulations reveal some scattering near the centerline, and at higher radial distances, they under-predict the experimental results. This discrepancy may result from the numerical scheme which employs an explicit finite difference method to solve the DQMOM transport equations; the results may be improved by implementing an implicit method.

The overall shape of a hollow cone spray is captured quite nicely by both methods, although some deviations are observed, particularly in DQMOM as compared to experimental profile, that might have resulted from the present DQMOM formulation, which is not yet fully coupled with the gas phase equations. Moreover, the discrepancies in the DQMOM results may stem from post-processing of the experimental data which is done to correct the number frequency at every measuring position to rule out the fluctuations in the effective cross section area of the measuring volume for the larger droplet sizes.



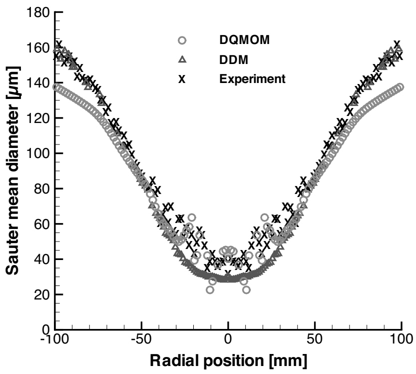


Figure 1. Radial profiles of Sauter mean diameter at cross sections of 0.12 m and 0.16 m away from nozzle exit.

*Corresponding author: rana.humza@iwr.uni-heidelberg.de

Numerical and Experimental Investigation of Fuel Spray Behaviour in Very High Density Environment Using LES

O. Kaario, T. Hulkkonen, V. Vuorinen, A. Wehrfritz, K. Keskinen, and M. Larmi
Department of Energy Technology, Aalto University, Finland
ossi.kaario@aalto.fi

Abstract

An effective way to increase the specific power from engines is to increase the maximum cylinder pressure. However, this also increases the gas density which has significant effect to the fuel spray evolution. In this study, experimental data from room temperature conditions at gas density of 39 kg/m^3 and at very high gas density of 115 kg/m^3 representing cylinder density at 300bar pressure conditions is compared to fuel spray large eddy simulations (LES). Special attention is put to the analysis of turbulence on local basis but also on a spray-induced volume basis. It is observed that several realistic features that can be seen in experimental sprays could be identified from the simulated fuel sprays. These include local turbulent structures, droplet clustering, and non-symmetric features. The computational model was also able to produce a range of frequencies and small scale structures which are needed from a properly working LES simulation. The spray opening angles as well as the droplet sizes were well captured with the LES model whereas the trend for the spray penetration was somewhat too long. The main reason for this is believed to be the mesh resolution that did not produce quite enough momentum spreading close to the nozzle.

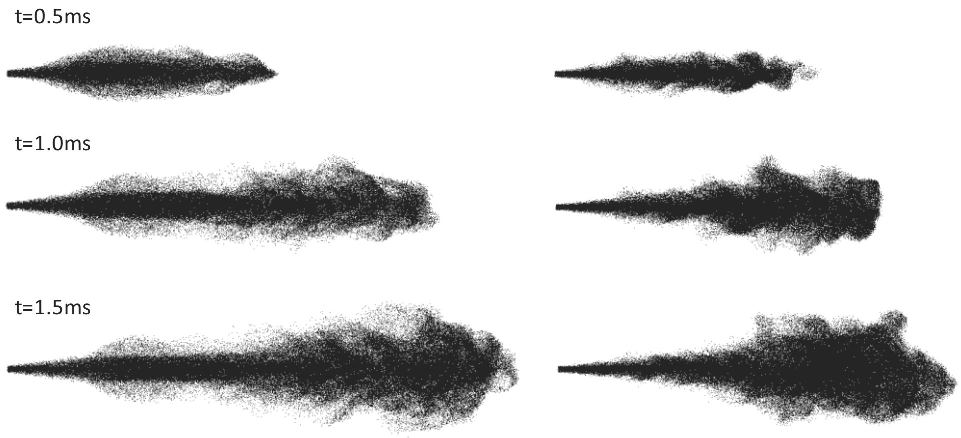


Figure. Fuel sprays at different times. Left: gas density is 39 kg/m^3. Right: gas density is 115 kg/m^3.

Conclusions

LES simulations and experiments have been carried out at two gas densities, 39 kg/m^3, and in very high gas density of 115 kg/m^3. It is observed that:

- The computed sprays in both normal 39 kg/m^3 gas density and in very high 115 kg/m^3 gas density have many realistic features that resemble those observed in experimental sprays. These include local turbulent structures, droplet clusters (preferential concentrations), voids, and non-symmetric features.
- LES was able to resolve large spectrum of frequencies and small scale structures which is necessary for a LES model to work properly.
- The production of turbulence was analyzed on cell basis and on a spray-induced volume basis. It was observed that the portion of the resolved scales was relatively high indicating good quality LES.
- The global parameters such as droplet sizes and opening angles were relatively well captured but the trend in the predicted spray penetration is too long. The main reason for this is in the momentum coupling between the gas phase and the dispersed phase. The mesh used in the current study is not fine enough leading to somewhat insufficient momentum spreading and turbulence production close to the nozzle. The calculated velocity boundary condition at the nozzle hole (z=0) is given as such at $z=6d_n$ assuming negligible velocity decrease during the distance of $6d_n$. This is because up to $z=6d_n$ the liquid volume fraction is high leading to reduced momentum transfer between the phases.

Effect of Parcel Models on particles' dispersion and gas-particle two-phase interaction in a particle-laden turbulent mixing layer

Masaya MUTO*, Hiroaki WATANABE
Central Research Institute of Electric Power Industry, 2-6-1 Nagasaka, Yokosuka, Kanagawa, Japan
masayam@criepi.denken.or.jp and whiroaki@criepi.denken.or.jp

Abstract

Particle-laden turbulent flows are encountered in a number of engineering applications such as energy conversion and propulsive devices using solid or liquid fuel. Therefore it is great importance to understand the effects of laden particles on turbulence and the diffusivity of particles in turbulent flow fields. In most practical particle-laden turbulent flows, the volume fraction of the dispersed particle is small, so that inter-particle collisions are often negligible. On the other hand, the particle mass fraction often becomes large owing to a large density ratio of solid or liquid particle to ambient gas, hence the modulation of flow field by particles cannot be neglected. Consequently, momentum exchange between two phases needs to be considered. In the traditional approach for particle-laden flow computation, the Eulerian equations for gaseous phase are solved along with a Lagrangean model for particle transport. However, tracing every particle in Lagrangean manner is still difficult for practical purpose because the number of particles in most engineering applications is huge. Therefore a parcel model is generally employed in the Eulerian-Lagrangean numerical procedures to reduce computational cost.

In the parcel model, a certain number of particles are represented by one parcel and the parcels are traced in flow field instead of each particle. Here we can consider two types of parcel model. The first model is that each parcel has the same volume, which is generally used in commercial software on gas-particle two-phase flow. In this model the smaller particle diameter becomes, the larger the number of particles is represented by one parcel. Another model is that each parcel represents the same number of particles. In this model the smaller the particle diameter becomes, the more the number of parcels increases. Both models have been used in existing studies but few reports on a difference between two models exist. So in this study, effect of parcel models on particles' dispersion and two-phase interaction is investigated. To validate both models, a base case in which parcel model is not applied is also implemented as a reference.

A three-dimensional DNS is performed in a particle-laden turbulent flow to investigate particles spatial dispersion, exchange of momentum, and interphase mass transfer which is represented by nonreactive scalar distribution yield from dispersed particles, are investigated in a turbulent mixing layer. In this study, neither inter-particle collision nor particle breakup is considered. The statistical results show that in former stage of the mixing layer parcel models do not affect particle dispersion so much compared to the reference case. However, the latter stage of the mixing layer the former parcel model causes a spatially skewed distribution of particles and low dispersivity compared to the reference case. This is because that a number of particles with small diameter, which are considered to be diffusive, are represented by small number of parcels in this model. Therefore a diffusion of the small number of parcels causes the skewed distribution of particles. Subsequently scalar distribution is also skewed. In the same stage of the mixing layer it is found in comparing with reference case that the latter parcel model shows better reproducibility than the former model. However, particle, momentum and scalar distributions are not smoothly distributed locally because of the discrete parcel appearance spatially. Finally, characteristics of different two parcel models are clarified.

* Corresponding author: masayam@criepi.denken.or.jp

Finite Volume Simulations of the Collision of Viscoelastic Droplets using Adaptive Re-meshing and Explicit Interface Tracking

Kyle G. Mooney, David P. Schmidt*
Dept. of Mechanical and Industrial Engineering, University of Massachusetts Amherst, U.S.
kmooney@ecs.umass.edu and schmidt@acad.umass.edu

Abstract

The effectiveness of gelled fuel jet impingement, food production, and polymer–based spray coatings can depend on the atomization and droplet collision behavior of fluids with complex rheology. Non-Newtonian effects such as shear-thinning, shear thickening, and viscoelastic extensional hardening (usually a consequence of macromolecular interactions) could alter collision outcomes as well as drop-to-drop mixing rates. Extensional hardening specifically has been shown to promote the stability of liquid ligaments, a structure often formed during the transients of a droplet collision. In this study, direct numerical simulations of single phase viscoelastic droplet collisions are performed within a finite volume framework. The free surface is modeled with an explicit moving mesh interface tracking method, allowing highly accurate calculation of surface curvature, which is increasingly important with decreasing Weber number. In this method, the boundary of the computational domain acts as the free surface with pressure and velocity field boundary conditions applied directly. Control volume distortion due to large boundary deformation is minimized using quality–driven node smoothing and a localized edge reconnection algorithm specific to tetrahedral meshes. Mesh-to-mesh vector and scalar field mapping error is reduced using a recently developed second-order accurate conservative interpolation scheme. The free surface and viscoelastic implementations are validated against analytical solutions and experimental data. Simulation results capture rapid growth in viscoelastic stress during ligament drainage in areas surrounding pinch-off points. In addition, fluid ligaments have shown to be increasingly stable with higher Deborah numbers.

---

*Corresponding author: schmidt@acad.umass.edu

High-speed rotary bell atomization of Newtonian and non-Newtonian fluids

J. Domnick
University of Applied Sciences Esslingen, Germany
joachim.domnick@hs-esslingen.de

Abstract

The atomization of liquid paints with high-speed rotary bells represents the most important application technique in the automotive industry, delivering paint films with highest quality in terms of color, effect and gloss. At this type of atomizer, a very thin film (thickness between 15 and 25 µm) is formed on the inner surface of rotating bell under the effect of centrifugal forces, which finally disintegrates at the bell edge in the form of a free film or liquid jets. In practice, a large variety of paint qualities is atomized. Hence, the present contribution deals with the effect of different flow behaviors. Namely, a solvent based, almost Newtonian paint (automotive clear coat) is compared with a waterborne, non-Newtonian paint (automotive base paint).

A 20 ns Nanolight system and a SPRAYTEC Fraunhofer diffraction system were used to investigate the disintegration process and the final size distribution as a function of the application parameters, i. e. paint flow rate and bell speed. As expected, the bell speed is the most important parameter influencing the atomization for both systems, Newtonian and non-Newtonian paints. In Fig. 1, measured size distributions of clear coat and base coat at two different bell speeds are compared. Obviously, the Newtonian clear coat with a constant viscosity around 85 mPas delivers a finer spray, although the shear-thinning base coat system has a significantly smaller apparent viscosity. However, for both systems Sauter mean diameters around 10 µm are obtained at practically relevant application parameters, confirming the efficiency of the atomization process.

Unfortunately, the results in terms of mean diameters are only partly confirmed by former investigations using water-alcohol-sucrose mixtures up to viscosities around 40 mPas [1]. In Eq. 1, the Sauter mean diameter is expressed in terms of the *Re* and *We* number and the flow number *q*. Bell diameter and bell speed are used as characteristic length and velocity, respectively.

$$D_{32} = 0.60 \cdot Re^{0.056} \cdot We^{-0.58} \cdot q^{0.41} \tag{1}$$

Although the general dependencies might be very similar, the absolute Sauter mean diameters are theoretically underestimated by a factor of 2. More investigations are necessary to consider the effect of polymeric components and solid content.

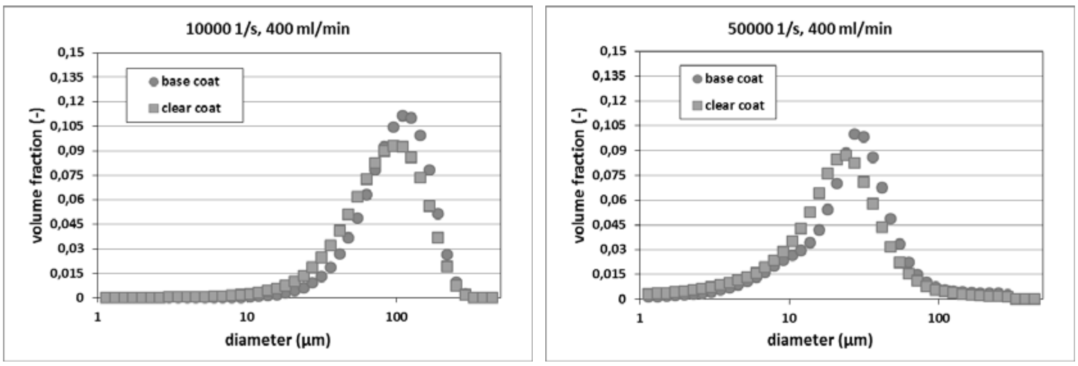


Fig. 1: Comparison of measured size distributions for 2 different bell speeds at a paint flow rate of 400 ml/min

[1] J. Domnick, M. Thieme: Atomization characteristics of high-speed rotary bell atomizers, Atomization and Sprays, Vol. 16, No.8, 2006

Experimental Study of Oil-Water Emulsions Injected into a Subsonic Crossflow

C.D. Bolszo, G.A. Gomez and V.G. McDonell[1*]
Department of Mechanical and Aerospace Engineering
University of California, Irvine
Irvine, CA 92697-3550 USA
cdb@ucicl.uci.edu, gag@ucicl.uci.edu and mcdonell@ucicl.uci.edu

Abstract

The current study investigates the influence of introducing water and diesel fuel oil as an emulsion on the penetration of a liquid jet into a gaseous crossflow. Tests are conducted at atmospheric pressure, with momentum flux ratios spanning 30 – 120 with water addition of up to 40 percent. Liquid and gas velocities up to 20 and 80 m/s, respectively are considered. Nozzle Reynolds numbers ranged from 3,000 - 11,000 and aerodynamic Weber numbers spanned from 200 – 1,400. The spray morphology, in conjunction with edge filtering and intensity thresholding, was utilized to establish the spray plume edge. Existing liquid jet trajectory equation framework successfully correlates the penetration of the spray plume without modification to account for characteristics of emulsions. It is also observed that, for the conditions studied which span between column and shear mode breakup, the breakup mode itself also influences the ability of the correlations to describe penetration.

* Corresponding author: mcdonell@ucicl.uci.edu

Experiments and Direct Numerical Simulations of binary collisions of miscible liquid droplets with different viscosities

C. Focke[1], D. Bothe[1*], M. Kuschel[2], M. Sommerfeld[2 †]
[1]Center of Smart Interfaces, University of Darmstadt, Germany
focke@csi.tu-darmstadt.de and bothe@csi.tu-darmstadt.de
[2]Mechanische Verfahrenstechnik, Martin-Luther-Universität Halle-Wittenberg
matthias.kuschel@uni-halle.de and martin.sommerfeld@uni-halle.de

Abstract

Binary droplet collisions are of importance in a variety of practical applications comprising dispersed two-phase flows. The background of our research is the prediction of properties of particulate products formed in spray processes. To gain a more thorough understanding of the elementary sub-processes inside a spray, experiments and direct numerical simulations of binary droplet collisions are used. The aim of these investigations is to develop semi-analytical descriptions for the outcome of droplet collisions. Such collision models can then be employed as closure terms for scale-reduced simulations.

In the present work we focus on the collision of droplets of different liquids. During spray drying the residence time of droplets will vary and their viscosity will be increased as a result of the drying process. In recirculation zones droplets of different drying state (i.e. different solids content and viscosity) may collide with each other. Due to the evaporation of the solvent, collisions of unlike viscous droplets will mostly exhibit also a diameter ratio less than unity.

In order to investigate these collisions, the experimental method as well as the numerical code are extended to analyze the collision of different liquids. A new experimental method has been developed in order to visualize the mixing and penetration process of two colliding droplets. Therefore, the fluorescence marker Rhodamine B is added to one liquid and droplets are excited by an Ar+ Laser. A combination of LED back light and fluorescence light is recorded with two synchronous cameras, whereas the second camera only records the fluorescence in order to have more details on the distribution of the fluorescence marker. Thus it is possible to study not only the dynamical behavior of the collision complex (outer surface) but also the internal mixing and even penetration can be analyzed quantitatively. Another advantage which comes with the new method is based on the fact that numerical researchers now have better validation data due to the high time resolution.

In the numerical part of the work we investigate the collision of droplets of different liquids and assume that surface tension and density are equal. The liquids are miscible, so no surface tension is acting at the liquid-liquid interface. The numerical method is extended to simulate the collision of different viscous droplets by solving an additional transport equation to simulate the mass fraction distribution inside the collision complex. The viscosity is coupled to the mass fraction and a careful averaging is employed at the liquid-liquid interface to capture the dynamics of the interface. The experimental data is used to validate the effect of non-constant viscosity. Subsequently, the local field data is used to gain a deep insight into the flow within the colliding droplets.

This study is devoted to the investigation of such collisions experimentally as well as numerically in order to analyze the effects of penetration and mixing. One primary is to discover elementary phenomena caused by the viscosity ratio of the droplets.

We gratefully acknowledge financial support provided by the Deutsche Forschungsgemeinschaft within the scope of the Priority Program 1423 ”Process Sprays”.

*Corresponding author numerical part: bothe@csi.tu-darmstadt.de
†Corresponding author experimental part: martin.sommerfeld@uni-halle.de

Effect of Charge on Aerodynamic Breakup of Charged Water Droplet

Yuan-Ping Huo, Jun-Feng Wang*, Zi-Wen Zuo, Zhen-Tao Wang
School of Energy and Power Engineering, Jiangsu University, 212013, Zhenjiang, China
*(*Email: wangjunfeng@ujs.edu.cn)*

Abstract. This study concerns experimentally the secondary breakup morphology of water charged droplets with the action of aerodynamic force in the dripping mode. A new measurement of charge-to-mass ratio has been designed and succeeds in calculating the tiny charge (nC) on a droplet. The high-speed camera combined with a microscopic zoom lens has been used to take images of the breakup morphology of charged and uncharged droplet with air assistance, and the differences which is calculated by the image processing technology is used to analyze the effect of charge on droplet deformation and breakup. The results show that two kinds of charged droplet of size 2.8mm are measured to have 32% and 50% of the rayleigh charge limit, respectively. Drop charging enhances drop deformation in the vibrational regime and makes droplet deform more easily because of the lower surface tension caused by electrostatic force. It seems no new breakup modes are observed for charged drop but large differences are observed from breakup morphology and breakup time compared to uncharged drops. And it indicates that charged drop can gain faster breakup equal to uncharged drop of higher *We* as a result of the electrostatic Weber number.

Keywords: Charged droplet; Air assistance; Secondary Breakup; Charge-to-mass ratio; High-speed camera; Breakup time; Breakup morphology.

The onset of fragmentation in binary liquid drop collisions

C. Planchette[1], E. Lorenceau[2], G. Brenn[3*]
[1]Research Center Pharmaceutical Engineering, Graz, Austria
[2]Lab. de Physique des Matériaux Divisés et des Interfaces (LPMDI), Univ. Paris-Est, France
[3]Institute of Fluid Mechanics and Heat Transfer, Graz University of Technology, Austria
brenn@fluidmech.tu-graz.ac.at and carole.planchette@tugraz.at

Abstract

Binary collisions of liquid drops have been of scientific interest since many decades, starting with investigations in meteorology to explain rates of rainfall accelerated by drop collisions and merging. Subsequent investigations of binary drop collisions treated primarily drops consisting of the same liquid. It has been only since a few years that interest in collisions of drops of different (in part immiscible) liquids has arisen [1]-[4]. The application behind these processes may be the encapsulation of, e.g., an aqueous liquid inside an oil shell. The latter could carry a dissolved oligomer which could be polymerized by UV light to solidify the surface layer. For the mentioned applications, the stability of the collision complex formed by the binary drop collisions plays a major role. Encapsulation, for example, requires the formation of an oil layer on the surface of an aqueous drop, and, therefore, the stability of the collision complex in a sense that at least a part of the oil remains on the aqueous drop after the end of the deformation and break-up processes induced by the collision. It is thus of natural interest to look at the conditions for the onset of fragmentation in binary liquid drop collisions (Fig. 1). We present experiments on collisions of glycerol-water and silicon oil drops of equal sizes, varying the relative impact velocity and the impact parameter. The drops are produced by controlled Rayleigh-Plateau-type break-up of laminar liquid jets. The trajectories of the drops are controlled accurately by placement of the drop generators on high-precision traverses. All measurements (drop size, velocity, impact parameter, geometrical properties of the drops after the collision) are based on image processing. The images are taken with a high-resolution video camera under flash-light illumination [4]. From the experiments we deduce the energy budget of the drops for two phases after the collision: a first phase from the instant of drop contact to the state of maximum deformation, where the collision complex assumes the shape of a disk with a rim at its edge, and a second phase of relaxation of the disk into an approximately cylindrical shape with an aspect ratio measured on the images. The results show that a considerable part of the initial drop energy is invested in the rim formation and dissipated there, so that only a small part of it remains available in the relaxation phase. The Rayleigh criterion turns out suitable for characterizing the stability of the relaxed cylindrical state.

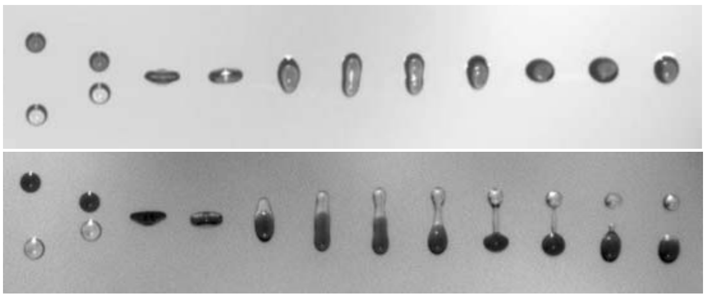


Figure 1. Top: Full encapsulation of a 50 % glycerol solution drop by SO M20 at relative velocity $U = 3.88$ m/s. Bottom: Crossing separation of a 50 % glycerol solution drop with SO M5 at $U = 3.97$ m/s.

References

[1] Gao, T.-C., Chen, R.-H., Pu, J.-Y, Lin, T.-H., *Experiments in Fluids* 38:731-738 (2005)
[2] Chen, R.-H., Chen, C.-T., *Experiments in Fluids* 41:453-461 (2006)
[3] Planchette, C., Lorenceau, E., Brenn, G., *Colloids and Surfaces A - Physicochemical and Engineering Aspects* 365:89-94 (2010)
[4] Planchette, C., Lorenceau, E., Brenn, G., *Journal of Fluid Mechanics* (2012) accepted.
[5] Jiang, Y. J., Umemura, A., Law, C. K., *Journal of Fluid Mechanics* 234:171-190 (1992)

*Corresponding author: brenn@fluidmech.tu-graz.ac.at

Electrospray characteristics of aqueous KCl solutions with various electrical conductivities

S. Faraji *, B. Sadri, B. Vajdi Hokmabad, E. Esmaeilzadeh, N. Jadidoleslam
Heat and Fluid flow research labratory, University of Tabriz, Iran
Sahandfaraji@yahoo.com and b.sadri.k@gmail.com and babak.vajdi@gmail.com and esmzadeh@tabrizu.ac.ir and info@navid.org

Abstract

In the present experimental study, the effects of electrical conductivity on electrospraying procedure are investigated. A metallic nozzle with 0.6 mm ID as high voltage electrode and a stainless steel ring as a ground electrode were employed in this work. Experiments were carried out in still room temperature. Four different aqueous KCl solutions were sprayed in various high voltages and flow rates.

For better manipulation of the disruption of a liquid into a spray of charged droplets, when subjected to an intense electric field, it is needed to study the electrospraying procedure exactly. The aim will be available when there is enough information about effects of surrounding medium and physical properties on electrospraying phenomenon. For accurate investigation, the path of sprayed liquid can be divided into 3 areas: Before liquid ejection from nozzle, at the tip of nozzle and after emission from tip of nozzle. In the literature, quality of liquid emission and types of droplet detachment from disrupted jet or cone/menisci, formed at the emitter tip, by varying physical properties such as constant dielectric, viscosity and surface tension were investigated. Present work investigates the spraying modes of water and aqueous KCl solutions with the aid of recognizing conductivity effects on electrospray phenomenon in air medium. The cone length of each case is reported and finally the behaviors of produced droplets discharging from these well-known cone shape modes after detachment are studied.

Results revealed that conductivity of dispersed solution acts a main role on forming and elongation of the cones in electrospraying procedure. The size and velocity of emanated droplets are also investigated in order to present a deep insight to the electrospraying phenomenon.

Results also demonstrate that:

- Modes of spraying change for different dispersed solutions and the effect of voltage and flow rate on the modes of spraying are presented.
- The produced cone on the nozzle elongates with conductivity increasing due to more charge on the cone surface and subsequently more EHD body force exerted on it.
- The onset voltage for appearing a cone shape on the nozzle decreases with conductivity due to the fact that the conductivity augmentation leads to more free charges on the liquid surface and results in forming of cone shape in lower applied voltage.
- Droplets bigger sizes and increasing of external applied electric field strength leads to discharging velocity diminishing with conductivity rising.
- The size of droplets produced in cone appearing modes(e.g. cone jet mode, unstable cone jet mode, ...) decreases when conductivity of dispersed solutions increase due to more charge on the emanated liquid and consequently more EHD force acting on it. It is noteworthy that the rate of augmentation of droplet sizes in this range of flow rate increases with conductivity rising.

* Corresponding author: sahandfaraji@yahoo.com

Vaporization and Collision Modeling of Liquid Fuel Sprays in a Co-axial Fuel and Air Pre-mixer

Xin Gu[1*], Saptashi Basu[2], Ranganathan Kumar[3]
[1,3]Department of Mechanical Materials and Aerospace Engineering
University of Central Florida, Orlando FL 32816
[2]Department of Mechanical Engineering, Indian Institute of Science, Bangalore 560012, India
xgu@knights.ucf.edu, sbasu@mecheng.iisc.ernet.in and Ranganathan.Kumar@ucf.edu

Abstract

Droplet collision occurs frequently in regions where the droplet number density is high. Even for lean premixed pre-vaporized (LPP) liquid sprays, the collision effects can be very high on the droplet size distributions, which will in turn affect the droplet vaporization process. Hence, in conjunction with vaporization modeling, collision modeling for such spray systems is also essential. The standard O'Rourke's collision model, usually implemented in CFD codes, tends to generate unphysical numerical artifact when simulations are performed on Cartesian grid and the results are not grid independent. Thus, a new collision modeling approach based on no-time-counter method (NTC) proposed by Schmidt and Rutland is implemented to replace O'Rourke's collision algorithm to solve a spray injection problem in a cylindrical coflow premixer. The so called "four-leaf clover" numerical artifacts are eliminated by the new collision algorithm. Next, the dispersion and vaporization processes for liquid fuel sprays are simulated in a coflow premixer. Two liquid fuels under investigation are jet-A and Rapeseed Methyl Esters (RME). Results show very good grid independence. At relatively low spray cone angle and injection velocity, we found that the collision effect on the average droplet size and the vaporization performance are very high due to relatively high coalescence rate induced by droplet collisions. We also found that the vaporization performance and the level of homogeneity of fuel-air mixture can be significantly improved when the dispersion level is high, which can be achieved by increasing the spray cone angle and injection velocity.

---

* Corresponding author: Ranganathan.Kumar@ucf.edu

Influence of Signal Properties on ElectroHydroDynamic Primary Break-up of Thin Sheets of Dielectric Liquid

C. Louste*, M. Daaboul, P.Traore and H. Romat
Laboratoire d'Etudes Aerodynamiques, University of Poitiers, Futuroscope, France
christophe.louste@univ-poitiers.fr

Abstract

In industrial air blast atomizers, the fuel is injected at low pressure through an annular slot. This produces a tubular sheet of fuel which is disintegrated into droplets by two coflowing airstreams. Unfortunately, a high air velocity is needed in order to obtain a fine atomization. This is the main drawback of air blast atomizer. With an air velocity below 30m.s^{-1} the atomization is too weak for fuel ignition. This problem could be encountered in relight ignition at high altitude, where the pressure and the temperature are very low. When the air velocity is below 10m/s the sheet of fuel is not sprayed anymore.

This paper is an experimental study of the primary break-up induced by an ElectroHydroDynamic actuator. Experiments have been performed on thin sheet of commercial diesel oil without active surface agent. The flow rate and the liquid sheet thickness are similar to the ones used in turbo engines. In the present work, the sheet of fuel is directly destabilized and disintegrated by an electrohydrodynamic actuator. Atomization is only due to electrical forces and the air velocity is equal to zero. As there is no coflowing airstreams and as the fuel is injected at low pressure, the liquid sheet is stable when the EHD actuator is turned off. Investigations on the primary break-up characteristics have been made with a high speed camera on the break-up length and on the spray cone angle, for various signal frequencies and amplitudes. The liquid velocity ranges from 0.6 to 2 m.s^{-1}

In industrial atomizers, the fuel is injected through an annular slit. Such device produces a thin turbular sheet of fuel. This shape is very efficient for atomization but difficult to investigate. More than a decade of studies in atomization process has proved that mechanism of disintegration are similar on cylindrical and planar thin sheet of liquid. Fig.1 shows a schematic diagram of the experimental setup. The fuel is pumped from a tank to the injector. The flow rate is controlled by the pump. Inside the injector a surge chamber smoothes out the turbulence of the fuel. Then the liquid is pushed through the planar slit.

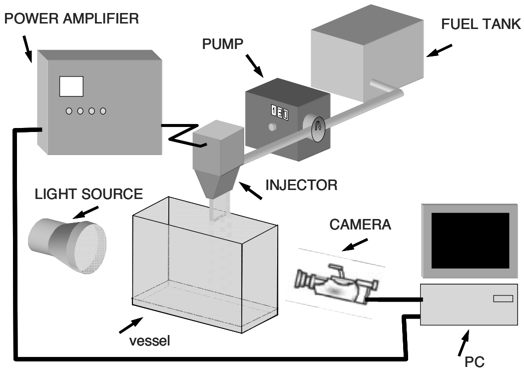
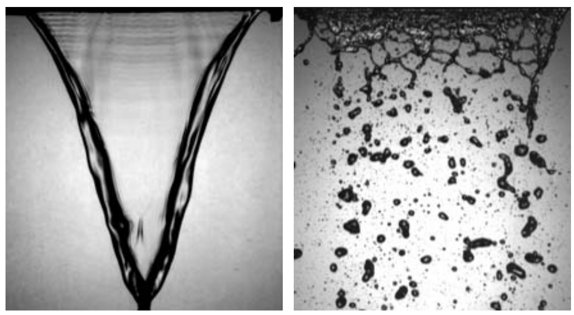


Figure 1: device



a) 0Hz b) 1kHz

Figure 2: front view of a sheet of diesel oil without electrical excitation (a) and with an AC signal frequencies (b)

The sheet is stable, when the EHD actuator is off (Photo a). When an AC signal is applied to the actuator, the sheet of fuel is shacked by electric forces. Four disintegrating modes depending on liquid velocity and on signal amplitude are analysed and described. In the flapping mode, the sheet waves as a function of both the liquid velocity and the signal frequency. The sheet wave amplitude can reach more than a 1cm. At relatively high liquid velocity, the sheet is greatly affected by charge injection. High frequency disturbances in the sheet become more predominant until perpendicular ligaments and then holes appear, giving rise to a perforated sheet. In the disintegrating mode, the sheet is fully atomized. The higher the frequency is, the more numerous the droplets, and the smaller the droplets size. A comparison between air blast and EHD atomization is made in order to underline similarities and differences of the two processes. It is demonstrated that the primary atomisation obtained with a 2kHz signal frequency is equivalent to the one observed with a 25m.s^{-1} coflowing air stream, which is an important advance in EHD technology.

Combining Global rainbow refractometry and PDA to extract the refractive index value by class of size: Application to CO2 capture by MEA spray.

S. Saengkaew*[1], M. Ouboukhil[2], L. Estel[2] and G. Gréhan[1]
[1] UMR CNRS 6614/CORIA, CNRS/INSA et Université de Rouen, FRANCE
[2] LSPC, INSA de Rouen, FRANCE
sawitree_s@coria.fr, maria.ouboukhil@insa-rouen.fr,
lionel.estel@insa-rouen.fr and grehan@coria.fr

Abstract

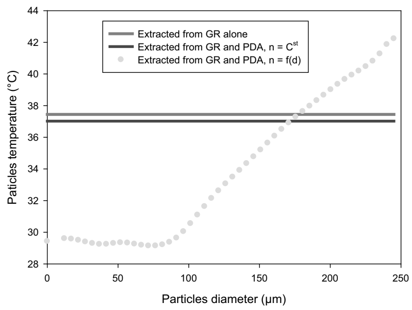
In the study of spray, it is now a challenge to know the geometrical parameters which characterize the spray (droplets velocity and size) as well as thermo-physical properties (temperature and composition). To extract the velocity and size information several techniques can be used but the most popular is the PDA which measure the size and the velocity of individual droplets. To measure the temperature and composition, the rainbow refractometry is attractive as the refractive index value is a function of both of them. Rainbow refractometry exists under two main configurations:

- Standard refractometry where the rainbow issues from one droplet is recorded
- Global refractometry where the collective rainbow created by a large number of droplets is recorded.

The standard rainbow is very sensitive to the droplet shape, and then it is essentially used on lines of mono-dispersed droplets. Global rainbow has been successfully used in complex spray but gives only access to an average value of the refractive index.

In our presentation to ICLASS 2012, by using the size distribution measured by PDA to process global rainbow signal, the value of the refractive index by class of size will be extracted. The approach will be detailed, and then proved on numerical simulations and experiments on sprays of n-octane.

As an example, the next figure displays the measured behavior of n-octane droplets injected at 46°C in air at 25°C. The spray has been created by an ultrasonic nozzle (SONIC USVC 130 AT) working at 20 kHz. The measurements have been carried out at 75 mm after the injection.



Measured refractive index versus particle diameter.
A) From global rainbow signal alone (the refractive index and size distribution are extracted from the signal). B) From global rainbow signal by using the size distribution measured by PDA but assuming the refractive index constant. C) From global rainbow signal by using the size distribution measured by PDA but assuming that the refractive index is a function of the particle diameter.

Afterward, the dual device will be applied to characterize the thermo-chemical behavior of Monoethanolamine drops (MEA drops) during CO_2 capture in a close tank. During the capture of CO2, the temperature and composition of the droplets change. These modifications affect the value of the refractive index which increases of typically 0.02. Then, the measurement of the refractive index value is a measurement of the chemical extent.

* Corresponding author: sawitree_s@coria.fr

High-speed ILIDS measurement of evaporating droplet

T. Kawaguchi*, I. Satoh. and T. Saito
Dept. of mechanical and control engineering, Tokyo institute of technology, Japan
kawat@mep.titech.ac.jp

Abstract

In order to investigate the dynamic interaction between liquid and gas phases in such multiphase flows as spray and bubbly flows, the size and momentum of individual particle are the significant quantities. Moreover the improvement of multi-dimensional and spatio-temporal spray measurement technique leads us to investigate the detailed heat and mass transport phenomena, and to construct the practical computation models for numerical simulations. Interferometric laser imaging for droplet sizing (ILIDS) technique was firstly developed for the investigation of the instantaneous spatial distribution of droplet size by means of laser light source and film imaging methodology. Sizing equation was derived by considering the optical pass difference of the scattered lights from a droplet by the geometric optical approximation. Although the difficulty in discriminating the overlapped parallel fringes in the captured image was overcome by optical squeezing technique, temporal resolution was still low due to the limitation of the imaging facility. In the last decade, the drastic improvement of optical matrix sensor such as CCD or CMOS camera enables the dynamic measurement of flow field whose sampling frequency is over several kHz with simultaneous exposure of each pixel. Another contribution is due to the light source. Nd:YLF laser could emit the coherent light with high repetition frequency. In the present study, the single-cavity double-pulsed laser system was employed in conjunction with the synchronized high-speed camera. The relation between droplet size and fringe frequency was numerically investigated to estimate the measurement error due to the non-linear relation between them. Moreover the effect of laser sheet thickness, and of refractive index fluctuation was analyzed. The size transition of the droplets within a few milliseconds was experimentally investigated. The results led us to evaluate the mass transfer rate at the gas-liquid interface of the individual droplet.

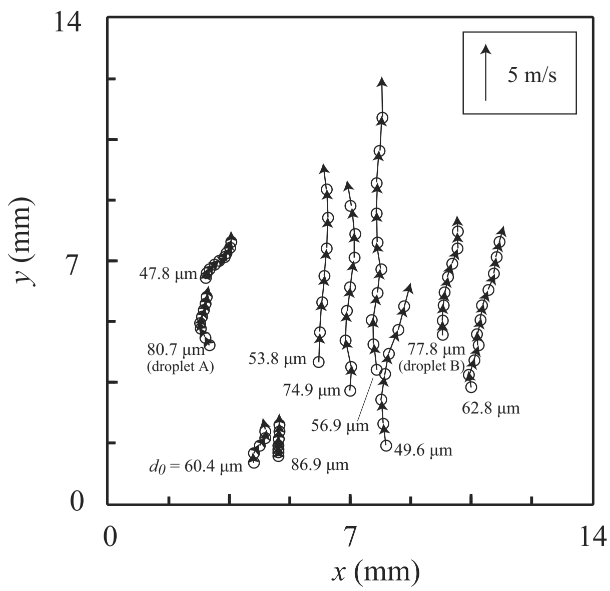


Figure 1. Measured particle trajectories by particle tracking. Hollow circles and corresponding arrows represent the instantaneous droplet locations and the individual velocity vector respectively. Frame rate of the image acquisition is 4000 Hz, consequently the temporal resolution of the droplet sizing and velocimetry is 0.25 ms. Initial droplet diameters are depicted in the vicinity of the initial droplet locations.

*Corresponding author: kawat@mep.titech.ac.jp

Spray velocity field analysis by Optical Flow Method – An alternative to Particle Image Velocimetry

S. Lorenz*[1], J. Goldlücke[2], D. Straub[1], D. Brüggemann[1]
1. Lehrstuhl für Technische Thermodynamik und Transportprozesse, Universität Bayreuth, Germany
2. Goldlücke Ingenieurleistungen, Germany
lttt@uni-bayreuth.de and juergen.goldluecke@giib.de

Abstract

There are various optical measurement methods to determine the flow field of sprays. In this work the Optical Flow Method (OFM) is applied to high speed recording of a spray and is compared to the approved method of Particle Image Velocimetry (PIV). Since sophisticated results can be achieved even by simple configuration, OFM demonstrates a promising alternative to PIV. For example quantities of transient and turbulent flows can be obtained only by using a high speed camera in connection with an intense white light source.

The determination of the flow field of a gasoline injector serves as an example of a typical measurement task. It is observed in an optically accessible spray chamber in which pressure and temperature can be varied. Spray wall interactions are possible with different bottom plates. In a first step OFM and PIV were compared by two simultaneously taken light sheet recordings. This was realized through a superposition of the light from a continuous Ar^+-laser for OFM high speed recordings and a Nd:YAG double pulse laser for PIV in the center of the spray. The Mie scattering is selected by appropriate optical filters to the high speed camera and to a two frame PIV camera. In a second step an integral OFM was compared to PIV. For this experiment the spray was back-illuminated with a diffuse white light source and the taken images were analyzed by OFM. For PIV evaluation the two frames were split into a large number of windows. For each window a velocity vector is calculated by cross correlation method.

The chosen OFM analysis is founded on a method of Horn and Schnuck. The essential point of this method is the assumption that a brightness pattern does not change during motion. The resulting data term and a weighted smoothness term based on a global image is combined in an energy functional. The minimisation of this function provides the motion vectors. To analyze a large number of frames in short time the resulting minimisation problem was simplified.

The OFM is compared to PIV at fixed times during injection. Lightsheets compared with each other show comparable results. For correct optical flow analysis the choice of a suitable light power is crucial. On the one hand the entire spray field has to be illuminated, on the other hand undesirable intensity gradients have to be avoided.

In the second comparison of OFM applied to shadowgraphy with simultaneous PIV the results show a very good agreement of the velocity vectors with respect to their magnitude and direction. Due to a homogeneous back illumination of the spray there is no sensitivity to intensity gradients from the light. In the investigation of spray wall interactions, both methods show good results for analyzing vertebral structures. Single droplets bouncing off the wall can only be analyzed by PIV due to general restrictions of OFM. With increasing pressure in the chamber and consequently higher density of the spray, the OFM shows better results because of greater intensity gradients inside the spray. In addition, the increase of chamber temperature shows the same effect.

In the presented work it is shown that OFM and PIV provide comparable results. Concerning the OFM, light-sheet imaging and shadowgraphy were investigated. It turns out that OFM is especially suitable for sprays at higher densities and at increased chamber temperature levels. Furthermore OFM allows the simple investigation of transient flows, due to the possibility to analyze an entire injection process with just one single measurement. Since this method can be applied in connection with Schlieren measurement technique, flow analysis of a spray will be extended to vapor phase in near future.

* Corresponding author: lttt@uni-bayreuth.de

Proper Orthogonal Decomposition Analysis of Cross Sectional Fuel Spray Data

Santosh Tirunagari*, Tuomo Hulkkonen, Ville Vuorinen, Ossi Kaario, Martti Larmi
Aalto University, School of Engineering, Espoo, Finland
(santosh.tirunagari, tuomo.hulkkonen, ville.vuorinen, ossi.kaario, martti.larmi)@aalto.fi

Abstract

In the present study, we analyse laser sheet spray imaging data using Proper Orthogonal Decomposition (POD). The analysis sheds light into the spray structures including spray-vortex interaction. The analysis of these structures is essential in understanding the fundamentals of spray characteristics. We study the influence of injection pressure on spray formation. Two different cases are considered corresponding to 450 bar, and 1000 bar injection pressures. The POD analysis reveals that the spray images can be decomposed into a set of orthogonal basis functions which provide more insight to the structures seen by bare eye. The effect of injection pressure is coherently explained with the POD modes. In particular, with the increase in pressure, might lead to better mixing in sprays. Furthermore, the sensitivity analysis of the POD modes reveals that the present conclusions are independent of the number of spray images for considered ensembles.

Introduction

Mixing of fuel droplets and air in diesel engines determines the quality of mixture. Thereby spray formation has a strong impact on emissions and efficiency in a particular engine. Hence the study of the spray formation is of great importance. To gain a deeper understanding on combustion process and mixing of fuel droplets and air, machine-learning methods can be utilised. Such a field include, Proper Orthogonal Decomposition (POD), a linear data transformation method. The POD modes can be considered to be energetically the optimal decomposition for the flow. Only a few POD modes are required to capture the large scale structures in sprays. In order to study about the inter connections among POD modes, POD mode coefficients can be utilized. The relationship between the two modes is the scatter plot of the two POD mode coefficients. The circular distribution in the scatter plot reveals strong connection between the two POD modes [1].

Results and Discussions

The spray images for different injection pressures corresponding to 450 bar and 1000 bar of pressure is decomposed into set of orthogonal basis functions using POD, called POD modes. From Figure 1, 450 bar pressure clearly has more energy when compared to 1000 bar pressure. The presence of high energy is related to the presence of large scaled structures. These large scaled structures in the POD modes reveal poor mixing.

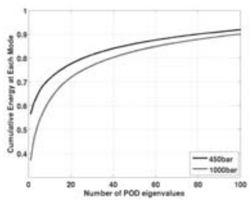


Figure 1. Cumulative energies.

Figure 2. pressure = 450 bar.

Figure 3. pressure = 1000 bar.

From Figure 2 and Figure 3, POD mode 2 reveals a coherent structure with greater intensity fluctuations near the tip region of the spray for both the injection pressures. This phenomenon is due to the spray to spray variations. A tilt in the tip penetration is observed for 1000 bar injection pressure. This possibly indicates a swirl caused in the spray. The Comparison of the two injection pressures is presented in the full paper.

References

[1] Meyer, K.E. and Pedersen, J.M. and Ozcan, O. A turbulent jet in crossflow analysed with proper orthogonal decomposition. Journal of Fluid Mechanics, 583 , pp 199-227 doi:10.1017/S0022112007006143

*Corresponding author: santosh.tirunagari@aalto.fi

Ballistics of Evaporating Spray in Wake of Shattering Drop

A. G. Girin
Odessa National Maritime University, Odessa, Ukraine
Club21@ukr.net

Abstract

Mathematical model of evaporation dynamics of a mist in a wake of a shattering in a speedy flow drop is elaborated on a base of obtained earlier distribution function for stripped droplets by sizes. Stripped droplets are considered as a multi-velocity evaporating continuum and a system of differential equations of two-phase polydisperse spray dynamics is composed. At one-dimensional spatial approximation of flowfield the mathematical problem is formulated and solved in a closed form in a dynamic 3D space. A detailed calculation of the ballistics of an evaporating spray, generated in the wake of a kerosene drop fragmented by air stream, is performed. The spray internal structure is investigated as related to the dynamic process of spray formation. Evolution of the dispersive characteristics of liquid-phase jet of the spray is studied. Analysis of the processes of liquid jet and vapor cloud formation is done and their structures are described.

System of spray dynamics equations consists of partial differential equations of torn-off droplets motion, evaporation and equation of evolution of density of their quantity distribution by sizes. Parent drop is regarded as a source of daughter droplets with given source distribution function and each daughter droplet – as moving point source of vapor. Dependencies of droplets drag coefficient on velocities and sizes, and intensification of evaporation due to their streamlining are taken into account.

The problem for system of spray dynamics equations is closed with obtained earlier source function. Non-stationary two-dimensional problem was formulated and solved numerically, and structure of two-phase spray was investigated. Calculations showed, that times of living of finest and largest droplets are the characteristics of liquid-phase jet formation in spray. Soon after moment $\tau_{\min}$ of entire evaporation of droplets of minimum in spray radius $r_{\min}$, that were stripped at $\tau = 0$, evaporation mass rate and current value of liquid-phase mass in spray exceed their maximum values because of beginning of droplets vanishing. After moment $\tau_{\max}$ of vanishing of droplets of maximum radius $r_{\max}$ the vanishing of droplets of new radii doesn't occur, and this is the necessary condition for stabilization of length of liquid-phase jet.

The proposed model allows to calculate spray mean diameters d_{ij}. Parameters of two kinds were considered: $\Omega_{ij}(x) = d_{ij}(x, \tau_c)$ are defined in any cross section of jet and characterize its spatial structure at fixed moment τ_c, while $D_{ij}(\tau) = \int d_{ij}(x,\tau)dx$ are calculated at any τ for the whole set of droplets and describe temporal changing of dispersity of jet in total. At the beginning bunch of curves $\Omega_{ij}(x)$ is narrow, that testifies to weak polydispersity of jet, but after droplets vanishing starts, the polydispersity increases. Stabilization proceeds gradually along jet from astern part to the tip. Soon after $\tau_{\max}$ all parts of $\Omega_{ij}(x)$ are stabilized. Dependencies $D_{ij}(\tau)$ confirm these conclusions. In final state jet polydispersity is much greater than that, produced by source.

Vapor cloud formation in spray was studied. At the beginning, intensification of evaporation due to rapid growth of liquid surface is so large, that vapor wave appears which has sharp front similar to blast wave. After losing contact with liquid phase this wave has convectional drift, keeping its form invariable. Gradual weakening of droplets source capacity generates rarefied wave in vapor mass distribution, so, far from source the distribution tends to "triangle" form.

Evaluations show that fuel – air mixture in wake of a shattering drop is substantially overreached in average, as vapor density several times exceeds the stoichiometric value. Vapor oversaturation leads to cooling of combustible mixture. Equation of heat balance for process of vapor – air mixing yields the temperature dropping in about 300°K, which means that delay of ignition may jump several orders high.

Presented mathematical model of evaporation ballistics of sprays allows to investigate the formation dynamics of liquid-phase jet and vapor cloud in the wake of a shattering drop in closed form, by the given parent drop radius and physical properties of gas – liquid system.

ICLASS 2012, 12th Triennial International Conference on Liquid Atomization and Spray Systems, Heidelberg, Germany, September 2-6, 2012

Explicitly Filtered Large Eddy Simulation of Two-Phase Flows with Evaporating Drops for Separating Numerical and Modelling Aspects

S. Radhakrishnan*, J. Bellan*,**
*Jet Propulsion Laboratory, California Institute of technology, Pasadena, CA 91109, USA
**Mechanical Engineering Department, California Institute of Technology, Pasadena, CA 91125, USA
senthil@engineering.ucsb.edu and josette.bellan@jpl.nasa.gov

Abstract

Numerical computations of sprays in turbulent flows provide a cheaper alternative to performing experiments in order to determine an optimal design envelope. The most accurate method for numerically simulating the flow is based on Direct Numerical Simulation (DNS) of the governing equations in which all scales of the flow relevant to dissipation are resolved. DNS, however, requires high computational cost and cannot be used in engineering design applications where iterations among several design conditions are necessary. Large Eddy Simulation (LES) provides a cheaper alternative to numerically simulate multiphase turbulent flows, although it has modeling requirements which do not exist in DNS. In LES only the energy-containing large scales, which are of engineering interest, are resolved and the more universal small scales are modeled thereby minimizing computational costs. The LES equations are obtained by filtering the Navier-Stokes equations. Thus, in the LES equations, the effect of the filtered small-scale motion on resolved large scale motion appears as Subgrid-Scale (SGS) terms and it represents the effect of the unresolved or "sub-grid" flow field which is unavailable; thus, these terms must be modeled. This modeling is typically done through representing the subgrid scale terms as functions of the large scale, i.e. LES, flow field. Despite great strides in LES, several unresolved problems remain. One of these problems plaguing model validation when comparing with a trusted template is the lack of grid independence of LES results. It is often assumed that the filter width is the same as the local grid spacing, an assumption which introduces considerable error in the smallest resolved scales that are most affected by numerical errors and considerable error in regions where the grid spacing varies drastically, as for instance in simulations where adaptive grid refinement is used in some regions to better capture the physics. To investigate whether predictions from conventional Large Eddy Simulation (LES), which are known to be grid-spacing and spatial discretization-order dependent, can be rendered grid-spacing and discretization-order independent, LES has been reformulated by explicitly filtering the non-linear terms in the governing equations. This reformulation leads to different gas phase equations, different gas-phase SGS terms and correspondingly, different gas-phase SGS models. Because in two-phase flow LES the effect of the drops on the gas mathematically manifests as source terms in the gas-phase equations, upon filtering the governing equations, the resulting filtered source terms can no longer be computed from the flow field solution, and instead must be modeled. Thus, SGS models must be developed for the drop field to compute these filtered source terms. The template for comparison with either LES or explicitly-filtered LES (EFLES) was obtained by creating a DNS database which is then filtered (FDNS). Conventional LES is conducted with the Smagorinsky model for the gas phase, and EFLES is also performed with Smagorinsky model; the drop-field SGS model is the same in all these simulations. The conventional LES method yields solutions which are both grid-spacing and spatial discretization-order dependent. The EFLES solutions are found to be grid-spacing independent for sufficiently large filter-width to grid-spacing ratio, although for the highest discretization order this ratio is larger in the two-phase flow compared to the findings in a similar study conducted for single-phase flows. For a sufficiently fine grid, the results are also discretization-order independent.

* Corresponding author: josette.bellan@jpl.nasa.gov

Influence of varied suspension properties on properties of spray-dried granules

S. Eckhard*, M. Fries, S. Höhn, C. Rödel
Fraunhofer Institute for Ceramic Technologies and Systems IKTS, Winterbergstrasse 28, 01277 Dresden, Germany

Abstract

To improve the poor flowability and handling behaviour of powdered materials an agglomeration step is necessary. Spray-drying is one of the most common processes to produce ceramic material mixtures with specific tailored properties from water or solvent based suspensions as feedstock. By varying the granule properties like granule density, size, shape, internal or external structure it is feasible, to conform the produced bulk to following processing steps. From literature and own investigations it is known, that the granule properties itself can be influenced by the suspension properties and spray-drying conditions. Especially in industrial applications it is essential, to fabricate constant granule product qualities or to be able to modify the product qualities in a well defined way. Therefore it is important to know, which influence the change of single suspension properties as well as single spray drying parameters has on the resulting product properties.

To study the effect of various suspension parameters on the suspension viscosity and resulting granule properties like internal granule structure and fracture behaviour, several suspensions of a particular Al_2O_3 material were produced. As the importance of suspension viscosity for the development of different internal granule structures is known, concentration lay on the selection of suspension parameters, which were able to modify the suspension viscosity:

- Solid content of the particular material
- Primary particle size
- Primary particle surface charge modification via pH value

The modification of suspension viscosity by different parameters enabled the investigation, if specific viscosity levels always result in comparable internal structures, independent of the parameter used for viscosity modification. During the drying process, the spray drying parameters were kept constant to exclude the influence of drying conditions. After spray drying, the resulting granules were characterized concerning internal granule structure and mechanical granule properties like granule strength and deformation behaviour.

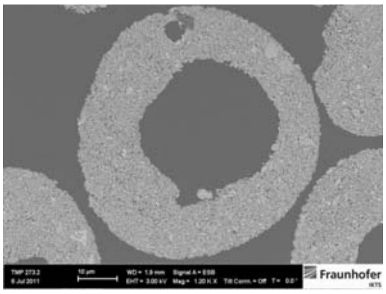


Figure 1: Solid content 40 wt%

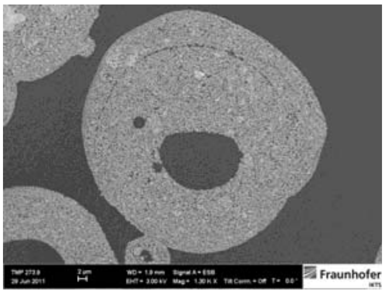


Figure 2: Solid content 68 wt%

Even if all suspension parameters changed the suspension viscosity, the resulting internal granule structures were not affected in comparable ways. E.g. an increase of solid content resulted in a viscosity increase and therewith a change in internal granule structure (see Figures 1 and 2). Higher solid contents resulted in smoother granules with thicker shells, which were responsible for an increase of fracture strength.

At the end of all investigations it should be possible to define, which suspension parameter influence the resulting suspension and granule properties the strongest to be able to influence the resulting granule properties systematically.

* Corresponding author: Susanna.Eckhard@ikts.fraunhofer.de

Movable and adjustable injection of air through concentrated sources of mass and momentum for the pneumatic atomization of polyurethane and the modeling of droplet-fiber-interaction for the manufacturing process of polyurethane-fiber-reinforced composites

P. Diffo*

PhD student, University of Applied Sciences in Hamburg, Germany
diffo@rzbt.haw-hamburg.de

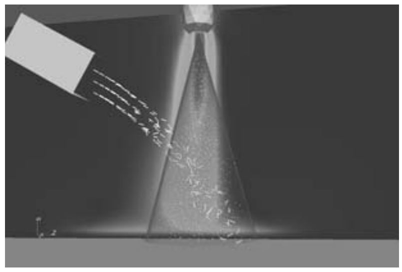


Figure 1: Spray-fiber-injection simulation

The polyurethane spray molding process is a manufacturing technology applied to produce composites by spraying the initially liquid polyurethane (PUR) matrix together with reinforcing fibers in a tool form or on a substrate. The fibers are laterally injected in the polyurethane-air spray cone for wetting before the entire composite is spread on the substrate, where it starts curing. In Fig. 1 a PUR mixing head nozzle is shown together with the inclined fiber glass chopper on the left side. A reliable prediction of the properties of the final composite is only possible if the average orientation and density distribution of the fibers in the matrix are known and translated into material properties. It is additionally of interest for the manufacturer to quantify the influence of some process parameters (e.g., fiber injection angle, fiber mass flow, polyurethane and air mass flow) on the orientation and density distribution of fibers for optimization purposes of the spray process itself. Therefore, the different interaction processes of the three phases (air, fibers and polyurethane droplets) within the spray jet are modeled and simulated in this project and their impact on the fiber distribution on the substrate are compared with experimental data from the facilities of our industrial partner (see Fig. 2).

The modeling of the two fluid phases, continuous air flow and particulate PUR droplets, including a four-way coupling (fluid-particle, particle-fluid and particle-particle-collision) is performed with a commercial CFD code (*ANSYS Fluent*) by using a RANS Euler-Lagrange approach with the built-in discrete model to represent the PUR droplets (discrete phase). A new approach has been developed for the air injection where a source term in the continuity equation (for the air mass) and in each of the momentum equations (for the regulation of the jet angle and velocity at the outlet) are applied to a definite number of adjacent cells of the computational domain. Those cells represent the nozzle outlet and can be activated and deactivated as the virtual injector moves through the domain. This method allows the implantation of an axial symmetrical air flow and avoids the modelling of a nozzle and the remeshing of domain after each motion of the nozzle. The impact of this new approach on the spray pattern will be compared with experimental data from the literature.

For the computation of the fiber dynamics, i.e., the one-way-interaction of the fibers influenced by the air and the one-way-interaction of the fibers affected by the PUR-droplets, the code *FIDYST* developed by the *Fraunhofer-Institute for Industrial Mathematics ITWM* (Kaiserslautern, Germany) is used. The fiber modeling in this tool is based on the Kirchhoff-Love-theory for large, non-linear deformations by making use of the Bernoulli-Euler material law and considering a fiber as a one-dimensional elastic rod, which depends on length and time. The fiber-droplet coupling is modeled using two approaches:

- Semi-Homogenization: an algorithm, which averages the time dependent dynamic data (mass, momentum) of discrete PUR-droplets present in a finite-volume cell over the cell volume to compute the effective droplet force acting on the fibers in that cell. This effective force is considered in *FIDYST* either as an independent drag force which together with the force from the air jet act on the fibers. in a straightforward manner by adjusting some specific flow field functions already used for the interaction of the fibers with the continuous air phase.
- Homogenization: an algorithm, which alternatively computes mixed/hybrid dynamic data out of the PUR-droplets and air present in a finite-volume cell. Thus a fictive ("homogenized") fluid is created and those data are used within the *FIDYST*-environment to compute the drag force acting on fibers.

Both methods are tested and compared to each other. The paper will present the entire methodology and results of the multiply coupled simulations.

* Corresponding author: diffo@rzbt.haw-hamburg.de

Polymerization in sprays – one step from a monomer to a powder polymer

M. Tewes*, U. A. Peuker
Institute for Mechanical Process Engineering and Mineral Processing, Technische Universität Bergakademie Freiberg, Germany
magnus.tewes@mvtat.tu-freiberg.de and urs.peuker@mvtat.tu-freiberg.de

Abstract

Spray drying of powder polymers is a common technique to generate well defined polymer particles in shape and size. To have a good solid handling spherical particles in a size range of 10 to 1000 µm are usually desired. The whole process seems to be standard processing but even the first step, the atomization, could be a bottle neck as we found out in our research. In particular the handling of the atomization is mostly dependent on the molecular structure of the polymer and length of polymer chain. The molecular weight of the polymer has a big influence on the rheological behavior of the solution, if it appears as non-Newtonian fluid with passing a critical concentration. Consequently the rheology of the fluid to be atomized has a big impact on the atomization behavior. Results on spray drying polyvinylpyrrolidone (PVP) solved in water show a change of morphology from particles to filaments with increasing molecular weight and concentrations of the polymer. This relation between molecular weight and atomizing behavior is also important for the current research where a polymerization will take place in sprays.

The spray drying of a reactive system is focus of this research. Here the former described spray drying process is expanded with an additional step, the polymerization or reactive step. Thus, a reactive monomer or partial linked monomer solution is atomized in a modified spray dryer. By decomposing a thermal starter in the fluid the monomer forms chains and consequently polymers. This starter is decomposed under certain kinetics by the drying gas or by the thermal input of a special 3-fluid nozzle (see following passage). The use of this nozzle is necessary because of the aim for optimal utilization of the small retention time within a spray dryer.

The coupling of starter, polymerization and drying kinetics is of special interest within our research. The model substances are water soluble monomers, salts of acrylic acid, and a radical thermal starter, an azo compound. The special tool for controlling starter and polymerization kinetics is the 3-fluid nozzle. This nozzle has three different feeds for the starter solution, monomer solution and the atomization gas, e.g. nitrogen. All feeds are mixed in- or outside the nozzle. In addition the nozzle has a double jacket for keeping a constant temperature inside the nozzle. The presented results show the influence of retention time of the fluids within the nozzle on the degree of polymerization at the nozzle tip. The optimal parameters for setting the start of the polymerization around the nozzle tip are presented. At this optimum a successful droplet formation is guaranteed, non-Newtonian effects are prevented and simultaneously completely polymerized particles with a certain molecular weight are generated in the process' downstream.

* Corresponding author: magnus.tewes@mvtat.tu-freiberg.de

Effects of superheat degree on flow field of multi-hole fuel sprays

Ming Zhang, Min Xu*, Yuyin Zhang, Wei Zeng
Shanghai Jiao Tong University, Shanghai, China, People's Republic of

ABSTRACT

Superheated spray has the potential to improve the fuel atomization and evaporation processes which is quite attractive for engineering application. However, the mechanism of superheated spray is still not clear yet. In this study, n-hexane sprays from an eight-hole injector in both vertical and cross-sectional planes under various superheat degrees were investigated by using high-speed Particle Image Velocimetry (HS-PIV) technique within the lower density region. In cross-sectional direction, the spray pattern changes from the eight plumes to a "donut" shape, a "pancake" shape and then an "octopus arms" shape as the superheat degree increases. In vertical direction, a lost momentum of spray particles with increased fuel temperature under non-superheated conditions can be observed through the flow field, resulting in a shorter penetration. As the fuel spray enters into the superheated region, significant flash-boiling induced plume expansion with increased superheat degree is observed; the vertical velocity increases while the radial velocity decreases with increasing superheat degree, which results in spray plumes collapsing to the injector axis (gas-jet shape under fully superheated conditions) and dramatically penetration increment under highly superheated conditions. The self-preserving velocity behavior of the gas-jet shape plume is captured and analyzed for fully-superheated sprays (SD>48°C). Obvious vortex at the spray plume outer boundary is observed. The vortex core of fuel sprays is identified. The vortex intensity and the vortex motion under various superheat degrees are also characterized, which are proved to be greatly dependent on the superheat degree. The results provide insight to the spray-collapsing processes under superheated conditions for multi-hole sprays. It's regarded that superheat degree is the predominant factor influencing the flow field, thus the structure of the superheated spray.

* Corresponding author: mxu@sjtu.edu.cn

An Investigation on the Spray Characteristics of DME with Variation of Nozzle Holes Diameter using the Common Rail Fuel Injection System

Ocktaeck Lim*, Sejun Lee, Soojin Jeong, Sedoo Oh
Department of Mechanical and Automotive Engineering, University of Ulsan, Korea
Graduate School of Mechanical and Automotive Engineering, University of Ulsan, Korea
Center of Green Powertrain, Korea Automotive Technology Institute, Korea
otlim@ulsan.ac.kr, sejun236@naver.com, sjjeong@katech.re.kr, and sdoh@katech.er.kr

Abstract

DME spray characteristics were investigated about varied ambient pressure and fuel injection pressure using the DME common rail fuel injection system when the nozzle holes diameter was varied. The common rail fuel injection system with DME cooling system was used since DME has properties of compressibility and vaporization in atmospheric temperature. The fuel injection quantity and spray characteristics were measured. The spray analysis parameters were spray shape, penetration length, and spray angle at each nozzle holes. Three types of injector were used, the nozzle holes diameter were 0.166 mm (Injector 1), 0.250 mm (Injector 2), and 0.250 mm with enlargement of orifice hole from 0.6 mm to 1.0 mm (Injector 3). The fuel injection pressure was varied by 5 MPa from 35 to 70 MPa when the ambient pressure was 2.5 and 5 MPa. When using Injector 3 compared with diesel injection quantity, the DME injection quantity was increased 1.69 ~ 2.02 times. Through this, it had the similar low heating value with diesel by Injector 1. In case of Injector 2 and 3, there were asymmetrical spray shapes at initial time. However as time goes by, the spray shape was symmetrical. Among three types of injectors, Injector 3 had the fastest development velocity of penetration length. In case of spray angle, Injector 2 and 3 got larger spray angle than Injector 1 and both injector had approximately same angle. Through these results, Injector 3 was optimized to solve the low heating value problem of DME.

* Corresponding author: otlim@ulsan.ac.kr

Injector Internal Geometry and Sub-Atmospheric Back Pressure Influence on Low Weber Number Liquid Flow

Nicolas Leboucher, Christophe Dumouchel*, Denis Lisiecki, Yves Wanner
CNRS UMR 6614 – CORIA Université et INSA de Rouen
Avenue de l'Université, BP 12, 76801 Saint Etienne du Rouvray, France

Abstract

The experimental work presented in this article addresses the question of the influence of the injector internal geometry on the liquid jet characteristics at the nozzle exit. As far as the spray formation is concerned, these characteristics are very much important since they constitute the atomization process initial conditions. High-rate shadowgraph imaging is used. Therefore, in order to have exploitable visualizations of the liquid jet structures, the injection pressure is kept low (not greater than 1 MPa). Furthermore, injections are performed under sub-atmospheric pressure to annihilate the aerodynamic force influence and to concentrate on the nozzle internal geometry effects. According to the literature, working at reduced ambient pressure might trigger internal flow cavitation without increasing the injection pressure. Thus cavitation might be triggered at low injection pressure allowing its effects to be visualized on shadowgraph images. This approach allows also addressing the question of the propensity of internal geometric characteristics to trigger cavitation.

A series of four injectors of the type of gasoline direct injection devices are used. They have three cylindrical orifices of the same diameter and regularly positioned at 23 degrees from the injector axis. One of the injectors is referred as standard and the others present geometrical differences concerning, 1 – the orifice entrance, 2 – the needle roughness and 3 – the sac roughness. The liquid used is Shellsol D40 with physical properties close to those of gasoline. The injection pressure P_i ranges from 0.2 MPa to 1 MPa and the ambient pressure from 0.004 MPa to 0.2 MPa. This corresponds to a gaseous Weber number of the issuing liquid jet ranging from 0.066 to 13.6. The experiments are limited to the fully open stage of the injection only. For each injector, the mass flow rate of this stage is measured as a function of the ambient pressure. Visualizations of the issuing liquid jet are performed with a high-speed camera Phantom V12.1 providing 66,667 frames/s. The snapshot resolution is 256x256 pixels and the spatial resolution is equal to 10.35 µm/pixel. The light source is a continuous 300 W Xenon arc source. The opening time is set to 0.3 µs which is sufficiently to freeze liquid structures.

At an ambient pressure P_a = 0.1 MPa, all injectors report the same mass flow rate (corresponding to a discharge coefficient equal to C_d = 0.59) except for the one with modified orifice entrance (C_d = 0.67). This behavior has been often reported in the literature. As P_a is decreased, the mass flow rate first increases with $\sqrt{P_i - P_a}$ and then decreases or remains constant according to the value of the injection pressure. This behavior is very similar to the one observed by many investigations of the literature and suggests the appearance of cavitation. The images show that the issuing liquid jet structures are affected by a reduction of P_a. The flow shows an intermittent behavior between two stages. The characteristics of this intermittency are quantified by measuring and analyzing the interface length per unit liquid surface area as a function of time. A Fast Fourier Transform of this signal reveals no specific frequency. However, contrary to what is often reported in the literature, it is observed that liquid cavitation does not always enhance atomization.

* Corresponding author: Christophe.Dumouchel@coria.fr

Finite Diffusion Multi-Components Fuel Droplet Vaporization Modeling Using Continuous Thermodynamics for Fuels with Distinct Composition Distributions

C. Shen[1], W.L. Cheng[1], C. F. Lee[1,2*]
[1]Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, USA
[2]Center for Combustion Energy and State Key Laboratory of Automotive Safety and Energy, Tsinghua University, China;
caishen1@illinois.edu, waycheng@illinois.edu, and cflee@illinois.edu

Abstract

Commercial fuels are composed of hundreds of chemically different hydrocarbons. An accurate and efficient way to model the vaporization process is required to represent the droplet evaporation under typical engine operation condition. In this study, a finite diffusion droplet evaporation model for complex liquid mixture composed of different homogeneous groups is presented in this paper. Separate distribution functions are used to describe the composition of each homogeneous group in the mixture. Only a few parameters are required to describe the mixture. Quasi-steady assumption is applied in the determination of evaporation rates and heat flux to the droplet, and the effects of surface regression, finite diffusion and preferential vaporization of the mixture are included in the liquid phase equations using an effective properties approach. A novel approach was used to reduce the transport equations for the liquid phase to a set of ordinary differential equations. The proposed model was compared against experimental measurements for single, isolated droplets of n-decane, kerosene, heptane-decane and diesel-butanol. The proposed model predicted the temperature and droplet size variations well. The present model was applied to simulate the evaporation of isolated droplets with composition of typical diesel. Computations showed that the model captured the main distillation characteristics of commercial fuels reasonably well. The proposed model is capable in capturing the vaporization characteristics of complex liquid mixtures.

---

* Corresponding author: cflee@illinois.edu

Modeling and Simulation of Water Evaporation from a Droplet of Polyvinylpyrrolidone (PVP) Aqueous Solution

S. R. Gopireddy*, E. Gutheil
Interdisciplinary Center for Scientific Computing, University of Heidelberg, Germany
srikanth.reddyg@iwr.uni-heidelberg.de and gutheil@iwr.uni-heidelberg.de

Abstract

This paper presents a model to describe the evaporation and drying behavior of a single spherical bi-component droplet containing evaporating liquid and dissolved solid substance. Although, many models are available in literature for droplet evaporation and drying, no appropriate model is available to study the liquid evaporation of a polymer aqueous solution. The present study concerns the evaporation and drying of a droplet of polymer (Polyvinylpyrrolidone - PVP) in the solvent water. The water evaporation rate from the droplet is computed using a modification of Abramzon and Sirignano's model [1], to account for resistance from solid layer at the droplet surface. The effect of PVP presence on vapor pressure of water is estimated by calculating the activity coefficient of water using the UNIFAC-vdW-FV method [2]. The system under consideration is governed by the continuity (diffusion) and energy equations, which are solved using a finite difference method.

Figure 1 shows the temporal evolution of PVP mass fraction inside the droplet of initial size 200 μm at 20°C when subjected to hot nitrogen gas flowing at 1 m/s with temperature of 75°C. The droplet size recedes with time and the solute concentration builds up at the surface, which becomes more significant at the higher gas temperature. The influence of drying conditions such as relative humidity in the gas and gas velocity on evaporation and drying rate are also studied. The effect of relative humidity of 5% in the surrounding gas on droplet evaporation and drying is shown in Fig. 2 at the same condition. The times of 4 s (black lines) and 8 s (gray lines) are displayed. The no humidity condition (solid lines) is also shown for purpose of comparison. The PVP mass fraction is lower for higher relative humidity, whereas the water mass fraction is higher due to the smaller driving force for the mass transfer, delaying water evaporation. The differences between no humidity and 5% humidity situations build up with time: at 4 s, the droplet interior is hardly affected, and humidity starts influencing the outer region inside the droplet. At a later time, $t = 8$ s, the differences are large in the droplet core, and they are reduced towards the droplet surface, where the saturation solubility is reached. The study shows the UNIFAC-vdW-FV method to greatly improve the results of the UNIFAC model.

Droplet humidity, and both gas velocity and temperature strongly affect PVP/water droplet drying characteristics. The present model effectively captures the initial stages of single droplet drying. A more detailed knowledge of saturation solubility, the vapor diffusivity into the solid layer and its thermal conductivity is desirable.

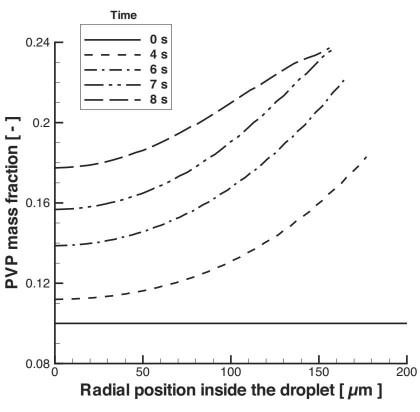


Figure 1. Time evolution of PVP mass fraction inside the droplet

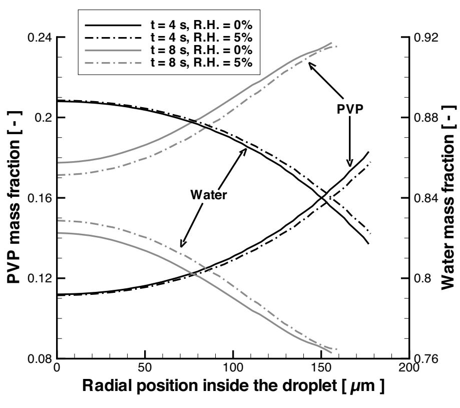


Figure 2. Effect of relative humidity (R.H.) on PVP mass fraction inside the droplet

[1] Abramzon, B., Sirignano, W. A., *International Journal of Heat and Mass Transfer* 32: 1605-1618 (1989).
[2] Kannan, D. C., Duda, J. L., Danner, R. P., *Fluid Phase Equilib*. 228-229: 321-328 (2005)

*Corresponding author: srikanth.reddyg@iwr.uni-heidelberg.de

Kinetic and Molecular Dynamic Modelling of n-dodecane Droplet Heating and Evaporation

S. S. Sazhin[1*], I. N. Shishkova[2], J.-F. Xie[1]
[1]Sir Harry Ricardo Laboratories, School of Computing, Engineering and Mathematics, University of Brighton, Brighton BN2 4GJ, U.K.
[2]Low Temperature Department, Moscow Power Engineering Institute, Moscow 111250, Russia

Abstract

A recently developed approach to taking into account the effects of inelastic collisions between two molecules on the solution of the Boltzmann equation is briefly described. This approach is based on presenting the change of state of molecules after collisions as a random movement along a surface of an N-dimensional sphere, the squared radius of which is equal to the total energy of the molecules before and after the collision in the reference system of the centre of mass. The projection of a point on the surface of this sphere in each of N-directions gives the root square of the kinetic energy in one of three directions in the physical space, or the internal energy of one of the degrees of freedom, of one of two molecules. The kinetic energies of two molecules are described by the first six dimensions of the system, and the remaining $(N-6)$-dimensions describe the internal energies. Recent results of molecular dynamics simulations to study the evaporation and condensation of n-dodecane ($C_{12}H_{26}$) at liquid-vapour phase equilibrium using the modified OPLS (Optimized Potential for Liquid Simulation) model are summarised. The predicted evaporation/condensation coefficient decreased from about 0.9 to about 0.3 when temperature increased from 400 K to 600 K.

Acknowledgements

The authors are grateful to EPSRC (Grant EP/H001603/1) of the UK for financial support of this project.

*Corresponding author: S.Sazhin@brighton.ac.uk

PIV Study on Flow Fields of Spray and Surrounding Gas under Non-Evaporating and Evaporating Conditions

J. Y. Zhu*, K. Nishida and O. A. Kuti
Department of Mechanical System Engineering, University of Hiroshima, Japan
zhujingyu-scl@hiroshima-u.ac.jp, nishida@mec.hiroshima-u.ac.jp, kuti@hiroshima-u.ac.jp

Abstract

The measurements of spray induced surrounding gas motion and spray flow were carried out by using LIF-PIV (Laser Induced Fluorescence-Particle Image Velocimetry) and PIV technique, respectively. The behaviors such as spray flow and gas flow velocity distributions, the ambient gas mass flow rate into the spray were compared quantitatively in both non-evaporating and evaporating conditions. Two injection pressures (100, 300MPa) and a micro-hole nozzle with diameter of 0.08mm were used. The results show the significant difference in the gas flow characteristics between non-evaporating and evaporating conditions. The total ambient gas flow mass is restricted in the evaporating condition. By comparing the droplets flow in the non-evaporating spray and the vapor phase flow in the evaporating spray, even though there is no much difference of the mean velocity distribution, the stronger vortex motion and faster spray momentum depletion can be observed in the evaporating condition, which results in a more homogeneous fuel/gas mixture and restricted penetration length. Higher injection pressure enhances the mean flow velocity and the small scale vortex distribution in the whole region, which implies the promotion of the fuel/gas mixing process. The vorticity distribution inside the spray corresponds to the resultant heterogeneity of the fuel concentration distribution. The fuel parcel with different mass concentration determines flow velocity distribution and the spray development.

* Corresponding author: zhujingyu-scl@hiroshima-u.ac.jp

Measurement of temperature and vapour distribution in an evaporating DISI-spray under engine relevant conditions using two-line excitation laser-induced fluorescence

Johannes Trost, Lars Zigan*, Alfred Leipertz
Dept. Engineering Thermodynamics (LTT) and Erlangen Graduate School in Advanced Optical Technologies (SAOT), FAU Erlangen-Nuremberg, Germany
Johannes.Trost@ltt.uni-erlangen.de and Lars.Zigan@cbi.uni-erlangen.de

Abstract

The main objective in the development of modern IC-engines is the reduction of fuel consumption and pollutant emission. Direct-Injection Spark-Ignition (DISI) strategies are commonly used and especially spray-guided concepts (SG-DISI) offer a high potential to reduce fuel consumption. Higher compression ratios are possible due to the evaporation cooling of the spray, which reduces the in-cylinder temperature so that engine knock is minimized. However, the mixture is not homogeneous through the combustion chamber, and therefore, local hot spots or low temperature regions can appear. The temperature distribution determines evaporation and ignition behavior. Furthermore, it is strongly affected by the fuel evaporation properties, which are different for modern fuels compared to gasoline. Thus, a spatial and temporal resolved measurement of the temperature and fuel concentration during injection is required to locally resolve vapor distribution and cooling in the spray.

Planar laser-induced fluorescence (PLIF) is an approved method to measure fuel vapor distribution under engine relevant conditions. In this study this technique is optimized for simultaneous vapor concentration and temperature measurements in DISI sprays. The fluorescence tracer 3-pentanone is added to a non-fluorescent surrogate fuel such as iso-octane. A multi-hole injector spray is excited quasi-simultaneously by two excimer lasers (two-line LIF) with the wavelengths of 248 nm and 308 nm. The signals are recorded with one double-shutter ICCD-camera. The temperature field can be calculated from the ratio of these signals, and afterwards the vapor concentration can be determined from a single signal with the known temperature. As the signal intensity of the tracer depends among other factors strongly on temperature, pressure and concentration, it has to be well calibrated. The calibration of the 3-pentanone is carried out in a high temperature / high pressure flow cell .

The spray measurements are conducted in a high temperature / high pressure injection chamber, where engine relevant conditions up to 1000 K and 10 MPa can be adjusted. A 6-hole solenoid-injector is used for the measurements, iso-octane with 20% (by volume) 3-pentanone as tracer is investigated. The figure exemplary shows the averaged temperature over 32 single measurements for 2.5 ms after visual start of injection (vSOI) and the measured and simulated profile at 40 mm distance from the nozzle for 2.5 ms and 3 ms after vSOI. The injection duration is 1 ms, the injection presure is 10 MPa and the chamber temperature and pressure are kept at 673 K and 0.8 MPa, respectively. At this point, a maximum cooling of about 132 K in the center of the spray is measured. To validate the measurement the temperature and concentration are compared to the results of a Computational Fluid Dynamics (CFD) spray model. The model is set up with OpenFoam 1.5 and uses a transient Reynolds-Averaged Navier-Stokes (RANS) approach with all relevant submodels for a realistic spray representation. The model is calibrated regarding spray shape and propagation as well as droplet size distribution for iso-octane. Additionally, an error analysis is carried out. For the averaged measurements a temperature error of 3.8% and mass fraction error of 5.5% are estimated.

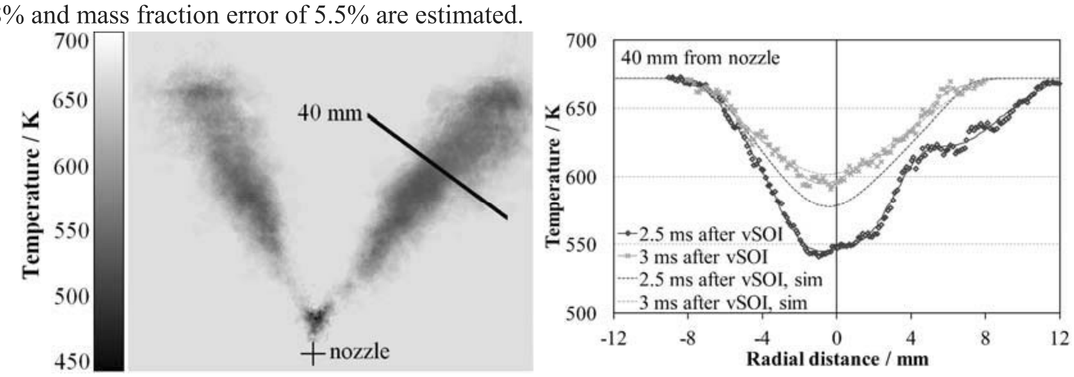


Figure: Temperature of iso-octane spray 2 ms after vSOI (left) and profile at 40 mm distance (right)

* Corresponding author: lars.zigan@cbi.uni-erlangen.de

Schlieren visualization of transient vapor penetration and spreading angle of a prototype diesel direct-acting piezoelectric injector

R. Payri[1*], J. Gimeno[1], J.P. Viera[1], A.H. Plazas[2]
[1] CMT Motores Termicos. Universitat Politecnica de Valencia, Spain
rpayri@mot.upv.es, jaigigar@mot.upv.es, juavieso@upv.es

[2] GM R&D, Warren, MI, USA
alejandroherna.plazastorres@GM.COM

Abstract

In this research work, a prototype diesel common rail direct-acting piezoelectric injector has been employed to study the influence of the fuel injection rate shaping on spray behavior (vapor phase penetration) under evaporative non-reacting conditions. This state of the art injector allows a fully flexible control of the nozzle needle lift through a parameter referred to as piezo stack charge level, enabling various fuel injection rates typologies under a wide range of test conditions. The tests have been performed employing a novel continuous flow test chamber that allows an accurate control of temperature and pressure up to 1000 K and 15 MPa respectively. The transient evolution of the spray has been studied recording movies of the injection event with a fast camera in a controlled 2-pass Schlieren visualization setup.

The effects of ambient temperature, injection pressure and piezo stack charge level have been studied and results are presented. Data obtained is correlated to previous liquid length and injection rate measurements of the same injector. Results show, as expected for all cases, that instant vapor penetration rate is closely related to the instant injection rate. Also for all cases, results show that the piezo stack charge and injection pressure affect the vapor penetration and spreading angle in a similar way. Ambient temperature alone seems not to have an important effect on vapor penetration and spreading angle, mainly because the two temperatures evaluated are close enough not to cause an important variation in ambient density. From the results, this needle control feature has proven to be a very versatile tool to control the injection process.

* Corresponding author: rpayri@mot.upv.es

Large Eddy Simulation of a polydisperse, evaporating spray jet with a presumed function method of moments

P. Dems*[1], J.N.E. Carneiro[2], W. Polifke[1]
[1]Lehrstuhl für Thermodynamik, TU München, Germany
[2]Instituto SINTEF do Brasil, Rio de Janeiro, Brazil
dems@td.mw.tum.de, joao.carneiro@sintefbrasil.org.br and polifke@td.mw.tum.de

Abstract

A polydisperse Eulerian-Eulerian two-phase model has been used to simulate evaporation of acetone droplets dispersed in an air jet. The presumed function method of moments model considers the polydispersity of the spray in terms of droplet diameter by transporting low order moments of the particle size distribution function. This approach takes into account the dependency on the droplet diameter of particle motion, evaporation rates and phase interaction forces, e.g. drag. Transport equations for the moments are obtained by integration of the Eulerian Multi-Fluid equations over the diameter spectrum. The phase interaction forces, acting as source terms in the gas phase equations and the moment transport equations, can be formulated in terms of the moments. Unknown moments, e.g. higher order moments which are not transported but needed for closure, are calculated assuming a functional form of the number density function. Herein a Gamma distribution is used whose reconstruction is possible by three consecutive moments. In order to capture the development and change of the particle size distribution in time and space by convection, i.e. the effect of different inertia of each droplet size class on the particle motion, the moments are transported with their respective moment transport velocities. However, only one equation for the third moment transport velocity is solved for, which is the particle phase momentum equation. Other moment transport velocities are obtained using an interpolation between the third moment transport velocity and the gas phase velocity. The interpolation is based on the particle relaxation times. The development of the model including validity of moment sets and formulations for Reynolds-Averaged Navier-Stokes and Large Eddy Simulations as well as the application to several configurations of cold particle-laden and bubbly flows have been shown in previous publications.

The objective of the present study is the formulation of model equations for non-isothermal flows to be used with Large Eddy Simulation and to validate the model against detailed experimental data. Derivation of the evaporation source terms and the coupling of the compressible gas phase with the particle phase are shown. Since the particle loading is very small in the experimental configuration, it has been neglected in the formulation of the compressible gas phase equations. The evaporation rate is determined using a modified film model due to the Stefan flow and the correction of the Sherwood and Nusselt numbers by the Frossling correlation. To obtain source terms for the moment transport equations, the formulation is integrated over the diameter spectrum. In the filtered LES equations these source terms were determined by using the averaged flow properties. Explicit sub-grid scale effects of drag force and evaporation rates have not been considered. The gas phase sub-grid stresses are closed using a compressible single phase Smagorinsky model. The two-phase moments model is then applied to simulate the evaporation of acetone droplets which are ejected by an air jet into a slow co-flow of air with room temperature. This configuration, investigated experimentally, e.g., by [Stårner et al.(2005) 'Effects of turbulence and carrier fluid on simple, turbulent spray jet flames', *Combustion and Flame* 143:420-432], was designed for well-defined boundary conditions and detailed measurement data for evaporating and combusting sprays are available. Therefore it is very applicable to validate the numerical results. Gas phase and droplet velocities as well as the moments of the particle size distribution function and mean diameters obtained from the numerical simulation are compared to the experimental findings.

---

*Corresponding author: dems@td.mw.tum.de

Analysis of Dispersion Process of Evaporating Spray Droplets Using Novel Scale Adaptive Simulation Approach coupled to a Langevin Dispersion Model

W.Ahmadi1*, A.Mehdizadeh, A.Sadiki
TUD, Dept of Mechanical Engineering, Institute of Energy and Power Plant Technology, Darmstadt, Germany
ahmadi@ekt.tu-darmstadt.de and mehdizadeh@ekt.tu-darmstadt.de

Abstract

In this paper, an advanced scale adaptive simulation approach (SAS) is developed and used to achieve time dependent and three dimensional space resolved simulation of large-scale structures which describe the turbulence dynamics. This features a zonal hybrid URANS-LES strategy that allows to capture needed unsteady flow structures using economical computational costs. In order to quantify the small-scale instantaneous velocity seen by the particle as it appears in the droplet motion equation and its effect on the droplet distribution, three different dispersion models are used. The evaporating droplets are captured using the Lagrangian procedure in which all numerical droplets are tracked by solving their equations of motion that include only the drag and gravitation force. The spray is diluted as the droplet volume fraction has a maximum value of $2.6.10^{-4}$micrometer allowing a full two way coupling. With regard to evaporation, the Uniform Temperature Model is applied. Note that all the assumptions required for the use of the model set are valid in the investigated configuration.

---

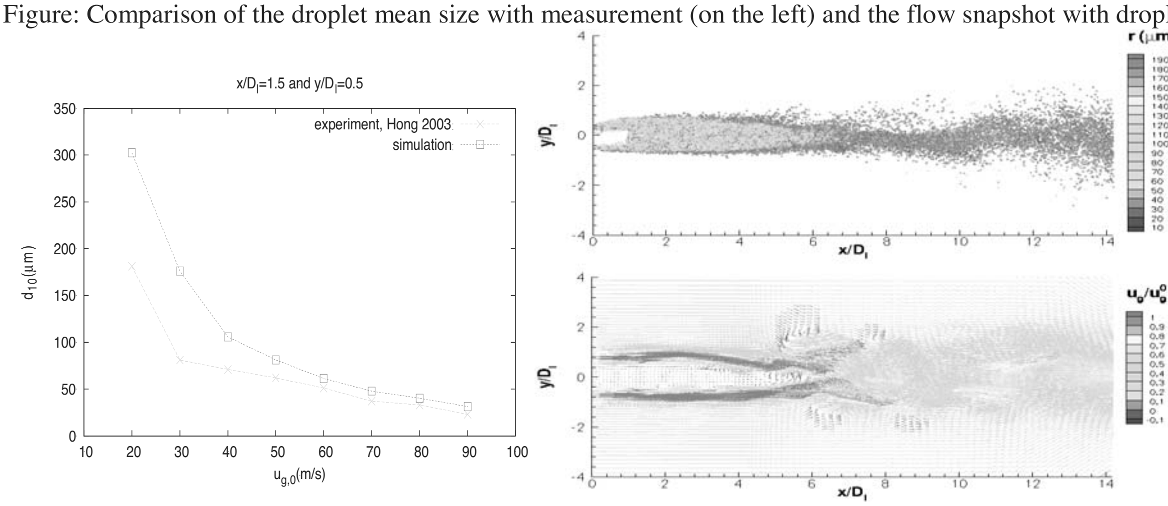
*Corresponding author: ahmadi@ekt.tu-darmstadt.de

Simulation of flow with spray closely to the air-blast injector: stochastic immersed body approach combined with LES

M. Gorokhovski*, T. Deng and C. Le Ribault
LMFA, Ecole Centrale de Lyon, Ecully, France

Abstract

In this paper, the new extension to the stochastic simulation of primary air-blast atomization is introduced, and is assessed by comparison with measurements. The idea of this extension is as follows. In LES of the gas flow, the primary atomization zone (liquid core, network of filaments and detached primary blobs) is viewed as immersed porous solid body with the stochastic structure. Namely, such a solid body is flowing with the inlet parameters for the liquid jet, and it is changing randomly its configuration. The statistics of configuration of this immersed body are used as boundary conditions in LES of the gas flow, thereby it is assumed that the jet fragmentation process is faster than typical time change scale in the gas flow. The statistical structure of the immersed body is defined by specifically introduced stochastic particles, moving in the space, and identifying the random position and curvature of the interface between the liquid and the gas. As it was previously in this approach, the simulation of position of such a particle is based on statistical universalities of fragmentation under scaling symmetry. To this end, the Langevin-type equation is derived and is simulated. Additionally, to each moving stochastic particle, we attribute now the random outwards normal to the interface. Thereby not only the random position of interface is simulated, but also its curvature. The statistics of curvature give statistics of the opening spray angle, and distribution of stripped blobs. For stochastic simulation of the outwards normal, the relaxation towards isotropy is suggested, along with propagation of spray in the down-stream direction. The inter-blob collisions in the primary atomisation zone are also introduced by analogy with standard kinetic approach for the ideal gas. Different closures are proposed for "statistical temperature" of blobs. The sensibility of this approach to the choice of mesh was analyzed. This numerical approach is assessed by comparison with experimental study of air-blast atomization, which was performed in LEGI (Grenoble, France). First, the computation showed a quite good prediction of the liquid core length; the predicted droplet's velocity-and-size statistics (the mean and variance) at different distances from the center plane, and at different distances from the nozzle orifice, are predicted also being relatively close to measurements. The influence of different inlet conditions (different gas velocity at the constant gas-to-liquid momentum ratio, as well as the different gas-to-liquid momentum ratio) was also assessed by comparison with experiment. Second, two qualitative effects were observed. The first one is the presence of the recirculation zone in the front of the liquid core, emphasized previously in the experimental study. The flapping of the liquid core and bursting in the droplets production were observed correlated with the periodic existence of this recirculation zone. The second effect is decreasing of transversal gradient of the velocity with increasing of the gas-to-liquid momentum ratio. Increasing of the gas-to-liquid momentum ratio leads to more intensive atomization process, thereby smearing the velocity field. It was emphasized that such effect may influence the scalar mixing (concentration/temperature) in the close to injector region.
Figure: Comparison of the droplet mean size with measurement (on the left) and the flow snapshot with droplets



* Corresponding author: mikhael.gorokhovski@ec-lyon.fr

Linear oscillations of viscoelastic drops used for measuring the polymer retardation time

G. Brenn*, S. Teichtmeister
Institute of Fluid Mechanics and Heat Transfer, Graz University of Technology, Austria
brenn@fluidmech.tu-graz.ac.at and s.teichtmeister@tugraz.at

Abstract

Linear oscillations of liquid drops have been of interest since the time of Lord Rayleigh [1], who derived the equation for the oscillation frequency of an inviscid drop in a vacuum. Theoretical analyses following this work accounted for the drop viscosity and the density of the ambient gaseous medium [2]. A further generalisation was achieved by account for the viscosity of the ambient medium [3]. Nonlinear drop oscillations were investigated, among others, by Tsamopoulos and Brown [4], and by Becker et al. [5]. Despite the wide literature about drop oscillations available nowadays, literature on oscillations of viscoelastic drops is still sparse [6]. In our contribution to the conference we first treat the linear oscillations of a viscoelastic drop in a vacuum theoretically. In our linearised treatment, the Oldroyd 8-constant model, which we use as the rheological constitute equation, reduces to the Jeffreys model. The characteristic equation derived is analogous to the equation obtained by Chandrasekhar for a globe of a Newtonian liquid [7]. The material law in our analysis is characterized by two time scales - the stress relaxation time and the deformation retardation time of the polymer molecules in the solution. The material law represents the viscoelastic liquid behaviour by a frequency-dependent dynamic viscosity, so that a quasi-Newtonian behaviour is obtained. While the relaxation time (of spinnable liquids) can be conveniently measured using a filament stretching rheometer, the retardation time is not readily obtained experimentally. In simulations of viscoelastic flows it is state-of-the-art to set the retardation time to 1/10 or 1/8 of the relaxation time. The present contribution to the conference proposes a method for determining the polymer retardation time from damped linear oscillations of drops of the polymeric liquid. For this purpose, the deformations of an individual drop in damped oscillation are measured by high-speed imaging. Two parameters are obtained by image processing: the frequency and the damping rate of the oscillations. Making use of these two values, two parameters involved in the characteristic equation of the drop may be derived: the dynamic viscosity in the Jeffreys model, and the polymer deformation retardation time. First tests of this new method with aqueous solutions of two different polyacrylamides at different concentrations revealed reasonable data for the two quantities. They show that, for the polymer solutions investigated here, the ratio of the relaxation time, measured with a filament stretching rheometer, to the deformation retardation time is not a constant and lies outside the range between 1/10 and 1/8. This finding casts doubt on the state-of-the-art method used in simulations. The paper will present the derivation of the characteristic equation of the oscillating viscoelastic drop, the differences of its behaviour from the Newtonian case, and the deformation retardation times derived from measurements on damped oscillations.

References

[1] Lord Rayleigh, J.W.S., *Proceedings of the Royal Society London A* 29: 71-97 (1879)
[2] Lamb, H., Hydrodynamics, 6th edn., Cambridge Univ. Press 475, 1997
[3] Miller, C. A., Scriven, L. E., *Journal of Fluid Mechanics* 32: 417-435 (1968)
[4] Tsamopoulos, J. A., Brown, R. A., *Journal of Fluid Mechanics* 127: 519-537 (1983)
[5] Becker, E., Hiller, W. J., Kowalewski, T. A., *Journal of Fluid Mechanics* 231: 189-210 (1991)
[6] Khismatullin, D. B., Nadim, A., *Physical Review E* 63: 061508 (2001)
[7] Chandrasekhar, S., *Proceedings of the London Mathematical Society* 9: 141-149 (1959)

*Corresponding author: brenn@fluidmech.tu-graz.ac.at

Mechanism of Atomization of Non-Newtonian Suspensions using Hydraulic Spray Nozzles

Addo-Yobo, F*, Kutsanedzie, F., Amevi Acakpovi, Anthony Woode, Edwin Mends Brew,
The Research & Innovation Centre, Accra Polytechnic, Barnes Road, P.O. Box GP 561, Accra, Ghana.
waddoyob@hotmail.com, kingknt11@yahoo.co.uk.

Abstract

A high-speed photographic technique has been used to record the break-up of sheets of suspensions, of Glass particles and Attapulgite particles to form a network of ligaments and nodes and the subsequent disintegration of the network to form droplets. The work is important for applications of atomization in which films are formed. These include combustion of rocket fuel and other single fluid low pressure atomization. The suspension of Glass used in this work, has no strong interactions with the solvent or other particles and is Newtonian. On the contrary, Attapulgite suspension do form structures when hydrated and behave in a non-Newtonian manner when sheared. The secondary disintegration processes have been characterized by studying the connectivity of the nodes to the network. The network of suspension of Glass is simpler and can be accounted for by nodes that are connected to the network by three or four ligaments. Nodes in the network of Attapulgite suspensions may be connected to the network by three, four or as many as five ligaments. The number of holes which surround a node in both suspensions is very close to the number of ligaments by which it is attached. This data ties in with the supposition that ligaments are formed from material between holes which contract. The nodes are the meeting points of three or more ligaments. The observation that the Ligaments and rims of the Attapulgite suspensions persists longer, compared to those of Glass suspensions accounts for the observation of higher number of ligaments per nodes (and therefore perforations per nodes). Thirteen types of the break-up of the ligaments are postulated although from the analysis of the photographs, five are observed for Glass suspension and nine for the Attapulgite suspensions. The droplets formed from the above can be grouped into three: nodal droplet, mid section droplet, satellite droplets. The perforations which lead to the formation of the above drops have different ages giving rise to the possibility of the formation of multimodal distribution of drop sizes. The ligaments formed from the suspensions of Attapulgite break-up give a higher proportion of satellite drops compared to those made up of Glass suspensions. The analysis of the two dimensional images of the drops formed indicate that multi-modal distributions of drops could result from this type of secondary disintegration. Further studies are required to make it possible to use the existing knowledge of the atomization of Newtonian suspensions to account for the atomization of non-Newtonian suspensions. More experimental as well as theoretical research on perforation statistics and wave growth under forced convection and single fluid conditions, are required. More theoretical work is required in order to be able to predict important measures of size such as the Sauter Mean Diameter of drops, which are produced from sprays of non-Newtonian suspensions.

* Corresponding Author: waddoyob@hotmail.com

A Study of Gelled Propellant Simulants Using Impinging Jet Injectors

J. A. Mallory*, P. E. Sojka
Department of Mechanical Engineering, Purdue University, USA
jamallor@purdue.edu and sojka@purdue.edu

Abstract
The present study experimentally and analytically investigates the effect of liquid rheology on the flowfield resulting from non-Newtonian impinging jets. Sheet instability wavelengths, sheet breakup lengths, ligament diameters, and drop diameters were measured from high-speed video images and compared to predictions from a linear stability theory, which accounted for the Bird-Carreau pseudoplastic rheology, and to semi-empirical theories of sheet breakup length taken from the literature. Analytical results showed an improvement over previous linear stability theories in that they no longer consistently over-predict measurements.

Introduction

The primary objective of this study is to determine how gelled propellant rheology and atomizer geometric parameters control sheet and drop formation. Key issues addressed include: the impingement zone sheet formation and impingement wave evolution, the shedding of ligaments and the subsequent breakup of ligaments into drops, and the effect of rheological behavior on all of these.

Materials

The shear-thinning, inelastic, solid-like gel propellant simulants used in this study were 1 wt.-% Kappa carrageenan and 1 wt.-% Agar. Due to the solid-like nature of the gel propellant simulants, accurate surface tension measurements are difficult to obtain. Therefore, several shear-thinning, inelastic, liquid-like carboxymethylcellulose (CMC) solutions were used to validate the analytical model: 0.5 wt.-% CMC-7HF, 1.4 wt.-% CMC-7MF, 0.8 wt.-% CMC-7MF, and 0.06 wt.-% CMC-7MF 75 wt.-% glycerin.

Bulk rheological properties were determined through the use of rotational and capillary rheometers. Two approaches were used to experimentally measure the surface tension of the solid-like gel propellant simulants.

Results and Discussion

The first linear stability analysis using a nonlinear rheological model for non-Newtonian liquids was applied to predicting the maximum instability wavelength, sheet breakup length, and drop diameter from 0.5 wt.-% CMC-7HF, 1.4 wt.-% CMC-7MF, 0.8 wt.-% CMC-7MF, 0.06 wt.-% CMC-7MF 75 wt.-% glycerin, 1 wt.-% Kappa carrageenan, and 1 wt.-% Agar impinging jet sprays. The developed theory either accurately predicted or bracketed experimental results, with the exception of 1 wt.-% Agar. As a result, it is more precise than previous theories [1], which consistently over-predicted experiment data.

Comparison studies between Kappa carrageenan and Agar gelling agents, which are similar in rheology, lead to the conclusion that molecular structure may have a greater affect on atomization behavior than investigated spray parameters and needs to be considered in predictive models for improved accuracy.

A preliminary demonstration of this concept is the drop diameter equation of Ryan *et al*. [5], which incorporates both sheet and fluid property characteristics. Comparing the three investigated drop diameter expressions, the one of Ryan *et al.* consistently accurately predicted drop diameter more than the others. This shows the importance of incorporating both fluid and sheet characteristics into a predictive analytical model.

An investigation was also conducted comparing measured ligament and drop diameter ratios. It was found non-Newtonian liquids have a ligament to drop diameter ratio of roughly one. As viscosity is increased, this ratio becomes larger. However, the ligament to drop diameter ratio is consistently lower than the Newtonian value of 1.89 regardless of what non-Newtonian liquid was tested.

References

[1] Chojnacki, K., 1997. *Atomization and Mixing of Impinging Non-Newtonian Jets*, Ph.D. Dissertation, The University of Alabama, Huntsville.
[2] Ryan, H., Anderson, W., Pal, S., and Santoro, R. (1995). Atomization Characteristics of Impinging Liquid Jets. *Journal of Propulsion and Power, 11*(1), 135-145.

* Corresponding author: jamallor@purdue.edu

Influence of Ambient Pressure on Twin Fluid Atomization
R&D work for high pressure entrained flow gasification

T. Jakobs[1], N. Djordjevic[1], S. Fleck[1], N. Zarzalis[2], T. Kolb[1,2]

Karlsruhe Institute of Technology (KIT), Germany
[1] Institute for Technical Chemistry, ITC vgt
[2]Engler-Bunte-Institute, EBI ceb

Tobias.Jakobs@kit.edu and Neda.Djordjevic@kit.edu

Abstract

High pressure entrained flow gasification is a very promising technology for the conversion of low-grade fuels to a high quality syngas to be used for production of chemicals and chemical energy carriers, e.g. liquid fuels or SNG. High gasification efficiencies using liquid fuels or slurry can only be achieved through generation of a fine homogenous spray at high ambient pressure and low gas to liquid ratio (GLR). Influence of spray quality on gasification performance is demonstrated by experiments in an atmospheric Research Entrained flow GAsifier (REGA).The present work is therefore focused on the atomization of liquids at increased ambient pressure using an external mixing twin fluid nozzle, typical for entrained flow gasification. For the experimental work reported, ambient pressure is varied in the range of 2 – 16 bar using water as model fuel. Spray quality, i.e. drop size distribution and axial velocity distribution is detected by Shadow sizer and PDA-System, respectively. A high speed camera is used for qualitative investigation of primary atomization. The experiments show the influence of reactor pressure on spray quality in terms of SMD, for operational conditions at We = const. and alternatively at u_{rel} = const. Basic finding is that keeping the We-number constant is not enough to guarantee for constant atomization quality at varying reactor pressure, but the relative velocity has a dominant effect. The experimental results are discussed on the background of a short literature review.

The Influence of Atomizer Geometry on Effervescent Atomization

D. D. Konstantinov*, P. J. Bowen, R. Marsh, P.J. Kay, A.P. Crayford, M. Derksen
Cardiff School of Engineering, Cardiff University, UK
sce4ddk@cardiff.ac.uk

Abstract

After an extensive literature review of the current state of this technology, an adjustable geometry effervescent atomizer was designed, built and studied at the Cardiff School of Engineering. Water and air were used as the operating fluids. The sprays produced by the atomizer were characterized using a Phase Doppler Anemometry (PDA) system which allowed for simultaneous real-time droplet size and velocity data to be obtained. High quality data was achieved, with data rates over 10 kHz and validation rates over 90 % in 2-D PDA mode in the high density sprays. A PDA probe designed for dense spray applications was utilized. A number of important operating parameters identified during the literature review phase could be altered on the atomizer, and their effects on fuel spray quality investigated. The operating parameters investigated in this manner included a number of operating parameters, geometric parameters as well as fluid viscosity. This paper discusses and analyzes the influence of geometric parameters on the quality of atomization. Geometric parameters investigated include exit orifice diameter, nozzle length-to-diameter ratio, mixing chamber diameter, mixing length and air injection geometry. Comparisons are made with previous studies performed using earlier versions of the hardware or alternative sampling techniques. Ongoing work will assess and optimize the performance of the atomizer using simulated biofuels mixtures – these will be presented in future publications.

---

* Corresponding author: sce4ddk@cardiff.ac.uk

Energy Conversion in Effervescent Atomization

J. Jedelsky* and M. Jicha
Faculty of Mechanical Engineering, Brno University of Technology, Czech Republic
jedelsky@fme.vutbr.cz and jicha@fme.vutbr.cz

Abstract

Atomization of liquids is, from the energy point of view, a process of conversion of an input fluid energy into surface energy of produced droplets accompanied with transformation amongst several energy types. In the paper, the spray generation in twin-fluid internally mixed atomizers is qualitatively described, starting with the gas injection into the liquid and ending with a far-nozzle droplet motion. Mechanisms involved in the two-phase mixture formation during internal flow, discharge of two-phase mixture and the spray formation are characterised based on our experimental results and available literature data with particular focus on the effervescent atomization. Near nozzle spray visualization with classical and thermovision camera elucidates the liquid breakup at different operation modes.

General energy equation for a steady one-dimensional homogeneous equilibrium flow without mass and energy conservation is used to explain the energy forms involved in the atomization process. Our estimation of the energy balance in effervescent atomization shows that the gas/liquid surface formation process during the internal mixing and during the discharge consumes a minor part of the input energy. Most of the input energy is spent for expansion work of the discharged gas, air entrainment process and losses related to the two-phase flow and discharge. Effectiveness of transfer of the input fluid energy (both gas and liquid) into the increased surface energy of the atomized liquid is quantified by the atomization efficiency η, which in the case of twin-fluid atomizers is given by a ratio of the surface energy of droplets in the spray, E_a, to the total energy E_i required to produce the spray. E_i consists of energy introduced by the pressurized gas, E_g, and energy of the supplied liquid E_l. Thus:

$$\eta = \frac{E_a}{E_i} = \frac{3 \cdot \sigma / ID_{20}}{p_l + GLR \cdot \frac{\rho_l}{\rho_g} \cdot \left(p_g + p_b\right) \cdot \ln\left(\frac{p_g + p_b}{p_b}\right)}$$

where p_g and p_l are gas and liquid gauge pressures (relative to ambient atmospheric pressure, p_b) upstream the exit orifice, ρ_g and ρ_l are their densities, ID_{20} is overall surface diameter of the droplets measured using PDA and σ is liquid/gas surface tension.

Atomization efficiency of effervescent atomizers is found to be in fragments of % for common operation pressures and gas-to-liquid-ratios (GLRs) (Fig. 1 top) and it is inferior by about one order to the efficiency of simple pressure and pressure-swirl atomizers for comparable conditions. Atomization efficiency depends on operation conditions of the atomizer and declines with an increase in GLR and with increase in input pressure with approximately logarithmic tendency (Fig. 1 bottom). An increase in the pressure and GLR promotes atomization reducing Sauter mean diameter, D_{32}, as well as D_{20}. However atomization efficiency drops down so the input fluid energy is utilized less effectively similarly as in the case of airblast atomizers, where it is explained by an increased gas/liquid separation that reduces the fraction of gas participating on the breakup process.

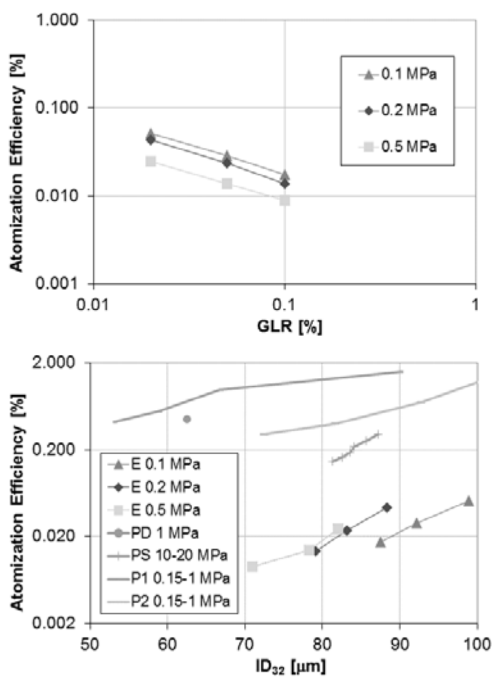


Figure 1 (top) Atomization efficiency of effervescent atomizer. (bottom) Atomization efficiency of different atomizers with regard to produced ID_{32}: e - effervescent atomizer with orifice diameter D_o=2.5 mm, operated with LHO/air; PS – pressure-swirl atomizer with D_o=2.8 mm, LHO; PD – simplex atomizer, D_o=0.4 mm, LHO; P1 – new pressure-swirl atomizer, D_o=0.36 mm, Kerosene; P2 – old pressure-swirl atomizer, D_o=0.36 mm, Kerosene.

Authors acknowledge financial support from project No. 101/11/1264 funded by the Czech Science Foundation and from „NETME centre - New Technologies for Mechanical Engineering" Reg. No. CZ.1.05/2.1.00/01.0002.

* Corresponding author: jedelsky@fme.vutbr.cz

Experimental study of the droplets evolution upon impulse spray formation

A.N. Ishmatov*, B.I. Vorozhtsov
Laboratory of Physics of High-Energy Materials Energy Conversion,
Institute for Problems of Chemical and Energetic Technologies of the Siberian Branch of the Russian Academy of Sciences (IPCET SB RAS), Russia
ishmatoff@rambler.ru

Abstract

This research is aimed at developing the experimental base and investigating the impulse formation of sprays, considering the evolving droplets in the dispersed flow. The work is urgent in view of the necessity to investigate the spray generation processes for prompt neutralization and deactivation of noxious aerosol and gas emissions as well as for suppressing fires and blast waves in coal mines. The spray formation process was simulated in laboratory conditions by applying a modified hydrodynamic tube-type device using a gas-generating pyrotechnic charge to intensify atomization. Considering the specific character of the impulse liquid atomization, we developed an experimental research complex comprising both the known and dedicated methods and procedures.

The high-speed video visualization of the impulse atomization showed that the spray has a conical symmetric shape, the liquid ejection from the atomizer is completed in 3 ms, the mean flow rate corresponds to ~200 m/s, the cloud is formed in 8 ms.

The laser measurement setup based on the low-angle laser light scattering method using a line of sight forward diffraction technique was employed to study the droplets dispersiveness parameters and spray evolution. The study revealed a limitation on measurements, which is caused by the multiple light scattering in the dispersed flow at the initial stage of its formation. To minimize the effect of the multiple scattering, it was suggested to employ a device in the form of a protective tube twice reducing the optical path length. The protective tube is placed in such a way as to isolate the probe laser beam on the half of the dispersed flow. The experimental investigation showed that under the used conditions the measurements using the protective tube can be run starting from 8 ms after the onset of the liquid atomization, whereas the protective tube is not employed, the measurements can be performed starting from 50 ms. The liquid spray produced by the impulse atomization was found to be fairly stable and stationary at the spray evolution stage.

Also to perform the work, an experimental method has been suggested that permits using the whole ensemble of droplets resulting from the atomization. The method consists in atomizing NaCl solutions, evaporating the solution droplets, and subsequently determining the disperse composition of droplets in the spray from the study results for the dry salt residue. The method is informative so far as the dry residue particle size is directly associated with the content of a nonevaporable impurity in the droplet, and the disperse composition of the initial droplets. As a result of the electron microscopy study of the droplets formed upon the impulse atomization of a 20% NaCl solution, it was found that their morphology may be different, that is, solid polycrystalline/monocrystalline structures and hollow spheroids which are also distinct in structure and sizes of crystals constituting their surface. The evaluation of the particles morphology showed that the droplets at the first several milliseconds after the formation underwent various interactions with the environment and their evaporation rates were significant. In other words, the salt residue particles are a sort of ‘artifacts’ reflecting the ‘marks’ of processes that occurred during atomization. In addition, when designing and testing impulse-type atomizing devices, the existence of hollow spheroids can serve as an ‘indicator’ of a maximally effective interaction between the dispersed flow and the environment at the initial stage of the flow formation and, hence, of a more effective liquid atomization.

As a consequence of the study, a general picture has been described for the droplet evolution upon the impulse atomization.

High flow-rate ultrasonic seeder for continuous operation

A. Lozano*, A. Rodríguez, J.A. García, E. Calvo, J. Barroso,
F. Barreras, A. González-Espinosa
LITEC, CSIC – Universidad de Zaragoza
María de Luna, 10, 50018 Zaragoza, Spain
alozano@litec.csic.es, juanto@unizar.es, calvober@unizar.es, jbarroso@unizar.es, felix@litec.csic.es, agonzalez@litec.csic.es

Abstract

Application of optical techniques to gaseous flows usually requires the previous seeding of the fluid with adequate tracers. To obtain satisfactory results, the tracers have to follow the flow as accurately as possible. Rapid response to fluctuations can be characterized by the Stokes number, St, and is strongly dependent on the marker diameter. Hence, small tracers are desirable, especially for highly varying flows. As compared to solid particles, small droplets are sometimes preferred because they can be generated *in-situ,* avoiding storage and agglomeration problems.

Popular devices for seeding purposes are fog generators, in which small droplets are produced by evaporation and re-condensation of specific fluids, usually glycerol, silicone oil or other mineral oils diluted in water. Liquid absorbed from a deposit is evaporated in a heat exchanger and expands in a nozzle, usually impelled by the pressure built up during vaporization. Without the aid of an auxiliary propellant gas, this operation principle prevents the device from operating in a continuous manner, because it requires a certain time to evaporate the liquid and allow its pressurization.

Here, an alternative is proposed in which droplets are generated by ultrasonic atomization. Ultrasonic atomization is common in household humidifiers, but the low flow rates of most commercial devices do not satisfy the needs imposed by many gaseous flows. This work describes the design and manufacture of a seeder based on ultrasonic atomization capable of operating in a continuous way with high atomization rates. It includes a dozen commercial piezoceramic disks that oscillate at 1.65 MHz, generating droplets with SMD in the range of 4-5 µm with rates over 0.6 g/s when operating with water. The twelve resonators are excited by individual oscillators connected to a common power supply that can give a maximum of 80 V, below the safety limit that could damage either the disks or the power transistor in each electronic oscillator circuit. The system has a float switch that disconnects the power if the fluid level decreases below a determined minimum so that the disks never operate without being immersed in liquid. This is very important, because without this cooling effect, they disks would get damaged instantly. Finally, the system has been designed to maintain a fixed liquid level over the disk surfaces, in a similar way as a bird drinking fountain.

To test its performance, the device has been used to seed a simple free air jet issuing from a 3.5 cm diameter nozzle with exit velocities of 2.9 m/s, 5.8 m/s and 8.7 m/s, corresponding to Re numbers of 6322, 12622 and 18944 respectively. Initially the seeding density is very satisfactory, but if only water is nebulized the droplets evaporate in a short time and the concentration becomes too low when moving downstream. The situation can be greatly improved if a small percentage of glycerol is mixed with the water (here 5% vol. has been tested), although the atomization rate strongly decreases when increasing the viscosity, as demonstrated in previous experiments using a single atomizer disk.

* Corresponding author: alozano@litec.csic.es

Designing Thread forming Rotary Atomizers by Similarity Trials

A. Mescher*, P. Walzel
Laboratory of Mechanical Process Engineering, Technical University of Dortmund, Germany
axel.mescher@bci.tu-dortmund.de and peter.walzel@bci.tu-dortmund.de

Abstract

Rotary atomizers operated in the regime of laminar thread disintegration can be used for the production of sprays with narrow distributed droplet sizes. The liquid threads formed are attenuated due to centrifugal acceleration and small drop sizes can be achieved, compared to the thread detachment diameter.

LAMROT can hold the desired break-up mode for an extensive range of liquid feed rates and revolution rates. This kind of rotary atomizer has a low plugging tendency and can be designed according to the principles described in [1]. For process design the mean drop size $d_{50,3}$ of the spray and the droplet *span*, representing the width of the droplet size distribution (DSD), have to be known.

The present work demonstrates how the operating parameters in spiraling thread atomization, i.e. volumetric flowrate and atomizer revolution rate, influence the drop size and size distribution. It is also demonstrated that similarity trials can be used for elaborating proper operating parameters.

In [2] we discussed the similarity of spiraling thread disintegration to the break-up of threads stretched in the field of gravity. As spiraling threads are subject to a gas-relative-velocity to the environmental gas, also in similarity trails a gas-crossflow to the threads must also be maintained. In [3] it was shown that thread break-up in the field of gravity is influenced by the non-dimensional volumetric flowrate $\dot{V}^*$, the non-dimensional viscosity μ^* and the gas-Weber-number We_g. $\dot{V}^*$ describes the influence of the flow momentum due to discharge and acceleration by gravity related to the capillary effect. The viscosity parameter μ^* represents the influence of viscous shear within the two-phase system. The gas-Weber-number We_g is defined as ratio of the dynamic gas pressure, caused by the gas-relative-velocity of the threads to the capillary pressure. For definition of the non-dimensional numbers, see Figure 1. The characteristic length of the system is given by the capillary length $L_c = (\sigma/\rho g)^{-0.5}$, scaling the equilibrium between hydrostatic pressure and capillary pressure. In [3] non-dimensional numbers were introduced and empirical correlations for non-dimensional thread break-up length, non-dimensional mean drop size d^* and droplet *span* were formulated.

In figure 1, experimental results from rotary atomization ($a = R\omega^2$) are shown and compared to the correlations formulated from the similarity experiments in the field of gravity ($a = g$). We_g is defined with the atomizer circumferential velocity $u = R\omega$ in order to describe the gas-relative-velocity of the spiraling threads. As figure 1 demonstrates a suitable coincidence of the data, the elaborated correlations can now be applied to the design of spray processes including spiraling thread disintegration. As an example the design of a spray drying process including a LAMROT atomizer for a given practical task will be discussed.

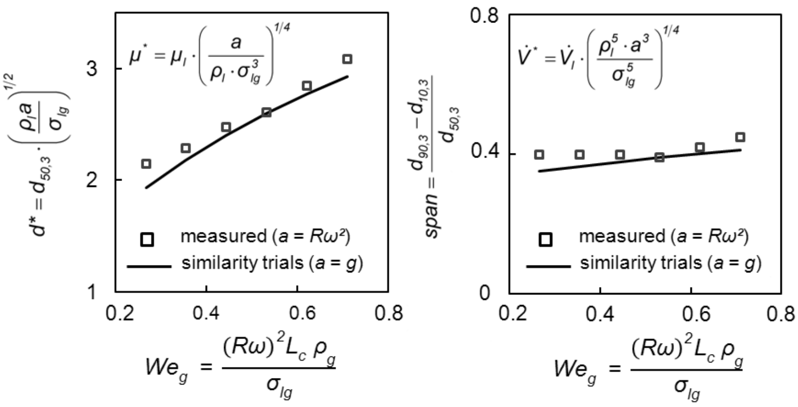


Figure 1: non-dimensional drop size d^* and droplet *span* for different gas-Weber-numbers We_g.
Comparison of rotary atomization experiments to calculated values. Experimental condition:
Glycerol/water-mixture, viscosity $\mu = 24$ mPas, atomizer diameter $D = 66{,}6$ mm, 40 threads per atomizer, liquid flowrate: 14,2 l/h, atomizer speed: $3000 < n < 8000$ rpm, Non-dimensional operating conditions: $0{,}24 < \mu^* < 0{,}4$ and $9{,}4 < \dot{V}^* < 42$.

[1] T. Schröder, P. Walzel, *Chem. Eng. Technol. 21-4*, 349-354, (1998).
[2] A. Mescher, P. Walzel, *ILASS - Europe 2010, 23rd Annual Conference on Liquid Atomization and Spray Systems*, Brno, Czech Republic, 5 - 8 September 2010.
[3] A. Mescher, A. Möller, P. Walzel, *ILASS - Europe 2011, 24rd Annual Conference on Liquid Atomization and Spray Systems*, Estoril (P), 05. – 07. September 2011.

* Corresponding author: axel.mescher@bci.tu-dortmund.de

Dropsizing of Near-Nozzle Diesel and RME Sprays by Microscopic Imaging

C. Crua*, G. de Sercey, M. R. Heikal
Centre for Automotive Engineering, University of Brighton, UK
C.Crua@brighton.ac.uk, G.DeSercey@brighton.ac.uk and M.R.Heikal@brighton.ac.uk

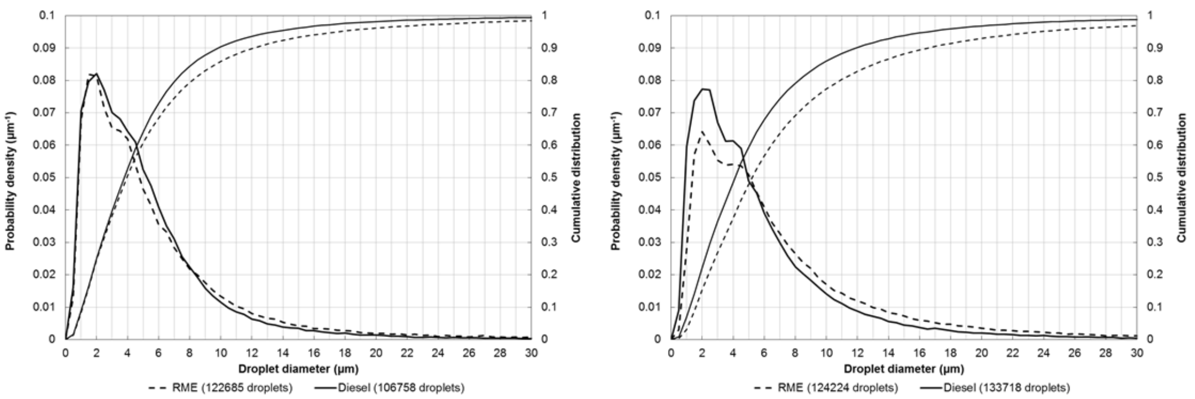
M. Gold
BP Global Fuels Technology, Pangbourne, UK
Martin.Gold@uk.bp.com

Abstract

The morphological composition of a typical modern Diesel spray is known to include complex structures such as ligaments and amorphous droplets, but most laser dropsizing techniques cannot diagnose drops that deviate from the spherical shape. Whilst direct imaging has potential for resolving arbitrary shapes, challenges remain to measure microscopic droplets in dense sprays. To this end, progress made with ongoing experimental investigations of the atomisation of diesel and biodiesel fuels is reported for the near-nozzle region using a long working distance microscope. A number of image processing techniques were explored and described to identify both small and large liquid structures. A discrete wavelet transform was found to improve the detection of droplets smaller than 10 µm, and contrast-limited adaptive histogram equalisation provided the best detection of medium droplets and large ligaments. The measured diameters were compensated based on an analysis of the droplets' local contrast and size. Droplet size distributions were measured for a non-additised diesel fuel and rape-methyl ester. The image processing algorithm was found to successfully discriminate between the fuels.

Dual frame at nozzle exit (dt = 400 ns) — 10 mm (0.3 mm off-axis) — 15 mm (0.3 mm off-axis)

Figure 1 Examples of shadowgraphic micrographs recorded for RME at 3 locations (100 MPa pressure).



(a) 10 mm from nozzle exit (b) 15 mm from nozzle exit

Figure 2 Droplet size distributions recorded for RME and diesel sprays injected at 100 MPa.

* Corresponding author

Near-Field Dynamics of Diesel and Biodiesel Sprays at 200 MPa Injection Pressure

S. MOON
National Institute of Advanced Industrial Science and Technology, Tsukuba, JAPAN
ss.moon@aist.go.jp

X. ZHANG, Y. GAO, J. WANG*
Argonne National Laboratory, Argonne, IL, US
xzhang@aps.anl.gov, ygao@aps.anl.gov, wangj@aps.anl.gov

T. TSUJIMURA
National Institute of Advanced Industrial Science and Technology, Tsukuba, JAPAN
tsujimura-taku@aist.go.jp

N. KURIMOTO, Y. NISHIJIMA
DENSO CORPORATION, Kariya, JAPAN
NAOKI_KURIMOTO@denso.co.jp, YOSHIAKI_NISHIJIMA@denso.co.jp

Abstract

High-pressure, high-speed diesel fuel sprays are complex multiphase flow phenomena. Great efforts have been devoted to understand their dynamics that is essential to the breakup, especially, in this near-nozzle region. However, conventional optical techniques are not effective to probe the dynamics in the first several millimeters of the optically dense region, where liquid is fast and in a complex morphology. By taking advantage of high-intensity and high-brilliance x-ray beams available at the Advanced Photon Source (APS), the morphology of the sprays can be imaged with ultrafast x-ray micro-imaging techniques and with sub-ns temporal resolution.

Furthermore, two short x-ray pulses (sub-nanosecond to a few nanoseconds) with a variety of intervals can be used to visualize the high-speed sprays. By tracking the movement of features in the double-exposure images without the need of seed particles, it becomes well possible to derive velocity fields of the sprays in the near-nozzle region. To understand near-field flow dynamics of diesel and biodiesel sprays injected at 200 MPa and travelling at a velocity exceeding 600 m/s, double-exposed images were taken using x-ray pulses with time interval of 68 ns. By using auto-correlation analysis, the near-field spray velocity can be obtained quantitatively.

We recorded the double-exposed x-ray images and derived auto-correlation functions of diesel and biodiesel sprays under various injection pressures up to 200 MPa. We note that 200 MPa injection pressure is not the maximum limit for the x-ray method rather it was limited by the specification of the fuel-injection system. The theoretical velocity was calculated using Bernoulli equation, which was compared against measurement. The results showed that the axial velocity increased with increase in injection pressure and reached over 600 m/s at 200 MPa injection pressure. We also made comparison between diesel and biodiesel sprays under various injection pressures. To probe the interaction between the sprays and surrounding gas, the local flow velocities were measured at different axial and radial locations. The results will prove valuable in providing validation of internal-flow and spray modeling.

* Corresponding author: wangj@aps.anl.gov

Time-Resolved X-Ray Radiography of Diesel Injectors from the Engine Combustion Network

A. L. Kastengren, F. Z. Tilocco, D. Duke, C. F. Powell, S. Moon, X. Zhang
Argonne National Laboratory, USA

Abstract

A significant hurdle to the understanding of sprays is the link between nozzle geometry and the fluid distribution in the spray. X-ray radiography can help to clarify this link by providing quantitative measurements of the spray density in the near-nozzle region, including at the exit plane. The current work describes x-ray radiography measurements performed at Argonne National Laboratory under the “Spray A” conditions of the Engine Combustion Network. Four injector samples have been studied, and model-dependent reconstructions have been used to generate 3-D maps of the average fuel density as a function of time. These measurements reveal differences between the sprays from nominally identical injectors which can be interpreted in terms of geometric differences in the injector nozzles that have been measured previously.

Wednesday

September 5, 2012

Contents

Diagnostics: Imaging Techniques II

Drop Ligament Breakup II

Interface Modeling II

Spray Impact II

Modeling of Primary Breakup II

Automotive Gasoline II

Medical Sprays

Diagnostics: Imaging Techniques V

Physics behind Diesel Sprays

Masataka Arai*
Department of mechanical system engineering, Gunma University, Japan
arai@gunma-u.ac.jp

Abstract

Diesel spray formation that is a dominant phenomenological event for performance of diesel engine and its combustion emission, has been received much attention not only from engineering but also from scientific field. Cavitation in an injector, breakup process of a high-speed fuel jet, fuel spray development in a combustion chamber, and combustion process following these atomization processes contribute to the scientific and engineering progresses in hydrodynamics. Diesel spray cavitation suggests the physical importance of turbulence caused by cavitation bubble disruption. Liquid surface stability problem in a breakup process leads us to many liquid disintegration models and their numerical simulation methods. Measurement of a diesel spray promotes new scientific achievement of laser diagnostics. The engineering achievements of these items coupled usually with physical considerations of diesel spray. Then finding of unknown physics behind diesel spray is the essential of next engineering approach of diesel spray. This paper highlighted physical approaches concerning spray behavior such as liquid breakup, spray penetration, spray volume, velocity distribution, and air entrainment. Main contents are as follows.

1. Diesel Spray
   - Macroscopic spray parameters: Spray angle, Breakup length, Core of spray, Spray penetration
   - Microscopic spray parameters: Size distribution, Mean diameter, Spatial distribution, Turbulence
2. Demand of diesel spray and Task of Diesel Spray Injection
   - Diesel spray combustion model
   - Task of diesel spray: Fuel transportation to a desired space and at desired timing
3. Nozzle Flow and Cavitation
   - High speed jet and nozzle cavitation, Cavitation number, Cavitation and Atomization
4. Breakup Behavior of Liquid Jet and Its Modeling
   - Breakup model of liquid jet, Wall impingement model, Spray model
5. Diesel Spray Development
   - Spray tip penetration: Catch-up motion of diesel spray and spray tail behavior
   - Combustion phenomena of diesel spray: Set-off (no-flame or lift-off) length of diesel spray
   - Interaction with wall and mutual interaction of sprays
   - Ultra-high pressure injection
6. Air Entrainment and Spray Angle
   - Definition of spray boundary and air entrainment
   - Spray angle: Effect of ambient pressure and ultra-high pressure injection
7. Diesel Spray in High Density Surroundings
   - Ultra-high boost engine, Homogeneous mixture, Spray behavior
8. Velocity Distribution inside a Diesel Spray
   - PIV application for velocity measurement and Gaussian distribution of velocity
   - Model of intense mixing zone

There are still too many unclear phenomena to describe breakup and spray development of liquid jet injected from diesel injector. Surface deformation related with cavitation and internal turbulence in a jet is a physically unclear problem concerning to the liquid jet disintegration. Liquid jet core near the exit of injector is hardly observed because of high dense droplet clouds surrounding it. As for the internal structure of diesel spray, only a little information is available for understanding the mixture formation and combustion of impingement spray. Further, there is almost no information for future diesel spray with ultra-high pressure injection system for ultra-high boost diesel engine. Then, it needs more fundamental measurement and deep physical understanding of diesel spray for future advancement of diesel engine and atomization technology

*Corresponding author: arai@gunma-u.ac.jp

To the Theory of Drop Breakup at a Relatively Small Weber Numbers

A.G. Girin
Odessa National Maritime University, Odessa, Ukraine
club21@ukr.net

Abstract

The theoretical explanation of "bag" and "bag-with-a-stamen" generation is suggested. Hydrodynamic instability of drop surface due to inertia forces of drop acceleration is enlisted as a hypothetical mechanism of breakup at low Weber numbers, $We<100$, when the drop deformation exceeds great values. The range $20<We<300$ of Weber number for drop is important, in particular, in processes of homogeneous fuel mixture preparation for combustion in rockets, turbines and other combustion engines, liquid metallurgy, fire suppression, etc.
A simplified model of deformed drop, regarded as a thin liquid accelerating layer, is elaborated for instability treatment. Two opposite kinds of boundary conditions on leeward side of drop surface were tested and it is shown, that at $We<70$, the conditions influence essentially on the dominant disturbance development. By the small perturbation techniques the characteristic equation of boundary-value problem for disturbances was derived, and unstable root, the range of unstable wavelengths and equation for wavenumber of dominant unstable disturbance were obtained. The dependencies of wavelength and characteristic time of dominant disturbance on Weber number for various values of deformed drop thickness are calculated and analyzed. The analysis of the numerical results allowed to determine minimum degree of deformation and wavelength needed for "bag" and "claviform" modes of breakup.
The linear instability analysis showed, that in the considered non-viscid hydrodynamic system with Weber numbers of the range $5 \lesssim We \lesssim 100$ a development of aperiodic unstable disturbances can be real mechanism of formation of "bag" and "stamen", which is the preliminary of disintegration itself. Results of present investigation based on mathematical ground allowed to give simple theoretical explanation of such a complicated modes of shattering as "bag" and "claviform". This simplicity as well as uniformity of explanation of other modes at greater values of Weber number [1] get evidence to the favor of the original hypothesis. Some reasons for causing "chaotic" mode of shattering are also suggested.
Obtained here from the linear instability analysis dependences help us to find the critical values of Weber criterion, as a conditions for one- and three half-wavelengths aperiodic disturbances to work under thin liquid layer. This values agree well with known experimental boundaries of "bag" and "claviform" modes of breakup for inviscid drops, despite of some roughness of adopted assumptions about streamlining and weak influence of conditions at the edge of deformed drop. Suggested in present communication elementary theory allowed to give simple visual explanation of preliminary and most complicated stage of these modes of breakup.
Following rupture of liquid film of "bag" is of indubitable interest as final stage of breakup, which forms aerosol cloud of fine daughter droplets. It was the aim of another investigation [2], because mechanism of rupture of liquid film of "bag", which is exposed to free stream, possibly has instability nature too. Connecting results of present paper with published earlier for greater Weber numbers, we can get general qualitative conclusion, that hydrodynamic instability of drop surface is indeed a universal mechanism, which proceeds at various modes of drop breakup.

[1] Girin A. G., *Journal of Engineering Physics and Thermophysics* **48**: 560-564 (1985)
[2] Girin A. G., Ivanchenko Ye. A., *12th ICLASS 2012*, Heidelberg, Germany, Sept. 2-6, 2012, # 1226

Breakup of Liquid Droplets

P. Khare*, D. Ma, X. Chen and V. Yang
Georgia Institute of Technology, Atlanta GA, 30332, USA
Prashant.Khare@gatech.edu and vigor.yang@aerospace.gatech.edu

Abstract

In this paper, deformation and breakup of liquid droplets has been studied. Detailed physics pertaining to four different breakup regimes, oscillatory, bag, multimode and shear breakup, has been investigated using an incompressible interface tracking methodology. Critical Weber numbers for the three regimes have also been indentified for a wide pressure range. A generalized regime diagram valid for $Oh < 0.1$ was developed to predict the breakup mechanism, taking into account the pressure effect on the critical Weber number, using data from previous experimental investigations and simulations conducted during the current study. Child droplet diameters were also characterized during the present research effort and it was concluded that for $We > 300$, the droplet size distribution follows a universal log-normal distribution. A theoretical correlation for sauter mean diameter (SMD), d_{32}, was also developed and it showed decent agreement with the simulation and experimental outcomes.

* Corresponding author: Prashant.Khare@gatech.edu

To the Theory of Drop Shattering in a Speedy Gas Flows

A. G. Girin
Odessa National Maritime University, Odessa, Ukraine
club21@ukr.net

Abstract

The general differential equations of shattering kinetics for mass efflux from fragmented in speedy gas flow drop and for quantity of stripped daughter droplets are derived on a base of mechanism of gradient instability in conjugated boundary layers on drop surface. At some assumptions the system is integrated and laws of parent drop mass diminishing, shattering drop motion, as well as distribution function for stripped droplets by sizes, are obtained theoretically. Intermediate and final distributions of stripped droplets by sizes are calculated and discussed for various values of definitive parameters of the problem. Comparison of approximated results with those obtained by more precise numerical scheme get evidence of good enough agreement. Some general peculiarities of dispersion kinetics are described.

Distribution function of quantity of torn-off daughter droplets by sizes is obtained at arbitrary ratio $h = A/3H$ of mass efflux rate $A = 0.46(1+(\alpha\mu)^{1/3})^{-1}(\mu^2/\alpha)^{1/6}$ to rate H of relaxational decreasing of relative velocity of parent shattering drop and gas flow. The result is based on earlier investigation of local surface instability with due regard to changing of velocity profile across the boundary layers, and along drop surface. For weak-viscosity liquids it revealed a new type of hydrodynamic instability – “gradient instability”. Mechanism of this type differs from that of Kelvine – Helmholtz type and it is caused by large enough velocity gradient inside liquid boundary layer. The theory explains the “stripping” mode of breakup as quasi-continuous high-freequency dispersing from unstable part $\varphi_{cr} < \varphi < \pi/2$ of drop surface. In speedy flows, when $\mathrm{GI} \gg \mathrm{GI}_{cr} \approx 0.3$, values of polar angle of critical point are small: $\varphi_{cr} \ll \pi$, so, most part of drop surface generates a mist of droplets (here $\alpha = \rho_g/\rho_l$ and $\mu = \mu_g/\mu_l$ are the media density and viscosity ratios, $\mathrm{GI} \equiv \mathrm{We_d}\,\mathrm{Re_d}^{-0.5}$ is criterion of gradient instability).

The equations of drop mass efflux (ablation) and of torn-off droplets quantity demand simultaneous solution of equation of drop motion in order to determine relative velocity, and equation for critical conditions of instability – to determine $\varphi_{cr}(\tau)$. For speedy flows and nearly spherical shape of drop we have obtained: $M(\tau) = \left(1 - A\left(\tau - \alpha^{0.5}X_d(\tau)\right)/3\right)^3$, that indicates direct influence of drop motion law $X_d(\tau)$ on its ablation law. Then, at approximation of experimental data for drop velocity versus time in the form $W = 1-\exp(-H\tau)$, $H = 2\sqrt{\alpha}$, or $\sqrt{\alpha}X_d(\tau) = \tau - (1-\exp(-H\tau))/H$, we have integrated the equation of drop mass efflux and have obtained drop mass history: $M = \left(1 - h(1-\exp(-H\tau))\right)^3$.

Distribution function was then obtained by integrating the equation of torn-off droplets quantity and the distributions $\Delta n(\tilde{r})$ for various $h, \mathrm{Re_d}$ were calculated. Analysis showed, that values of h slightly higher than $h=1$ are inherent to flows behind shock and detonation waves, the values $h>4$ correspond to ablation of liquid meteoroids and the case $h<1$ – to incomplete shattering of viscous drops. When $h>1$, the whole drop is dispersed to the moment $\tau_b = H^{-1}\ln\left(h/(h-1)\right)$. When $h<1$ dispersion terminates before drop is completely shattered, because of quick reducing of main reason of dispersing – relative velocity.

In view of lack of empirical data about laws of drop motion for various gas-droplets systems the main relations of shattering drop kinetics were obtained on reliable ground of theoretical laws. They were found analytically as solutions of system of non-linear differential equations of drop motion, drop mass efflux and quantity of torn-off droplets in a speedy uniform gas stream at neglecting by drop deformation. The two approaches of determination of drop motion law, which are based on empirical and theoretical methods, have led eventually to similar distributions, but theoretical approach has advantage being independent from lack of empirical data, so obtained formula is applicable to any gas – droplets system.

Direct numerical simulation of compressible multiphase flow using a discontinuous Galerkin based multiscale approach.

F. Jaegle*, S. Fechter, M. Boger, C. D. Munz
Institute of Aerodynamics and Gas Dynamics, University of Stuttgart, Germany
{jaegle,fechter,boger,munz}@iag.uni-stuttgart.de

Abstract

Many scientific or industrial applications for the direct simulation of multiphase flow warrant the use of a compressible formulation, as high temperatures and pressures or even shock waves may be encountered. Examples are rocket engines or automotive injection systems. However, the simulation of compressible multiphase flow is in certain respects more complex than the incompressible treatment that is often found in numerical solvers today.

Typically, two-phase flow solvers rely on two crucial elements: the first is a method that allows to define the geometry and the temporal evolution of the interface between the two phases. In the present study, a level set formulation is used. The second is a numerical strategy to treat the discontinuous nature of the interface as well as the related physics such as impinging waves, phase change or surface tension.

Mainly for incompressible flows, diffuse interface methods that allow numerical smearing of the interface are common. For compressible flow, this approach can be problematic as simple averaging of the equations of state (EOS) may not be consistent with the intermediate states. The alternative that we adopt in the present study is a sharp interface approach, which keeps the different fluids separate from a numerical perspective. Our multiscale method solves the Navier Stokes equations for the macroscopic scales of the flow. At the interface, jump conditions are provided by a separate solver for the microscale. Many microscale solvers are conceivable, in the present study, we rely on Riemann-type solvers. The approach is suitable for general EOS and has been tested for different formulations. In the present study, results for the combination of an ideal gas and the Tait EOS are presented.

The numerical method consists of the following components: for the macroscale solver, a discontinuous Galerkin (DG) spectral element method is used. This scheme does not enforce continuity between the elements, therefore, it relies on numerical fluxes similar to a finite volume scheme. In the bulk phases, numerical fluxes are obtained from a standard Riemann solver. The level set equation is solved using the same DG scheme. Near the phase interface, the jumps in certain flow variables are assumed to coincide with the nearest element boundary, where the Riemann solver is replaced by the microscale solver. Instead of evaluating a single numerical flux, the microscale solver yields discontinuous fluxes, which are applied on the liquid and gaseous side respectively. This procedure allows to conserve a sharp discontinuity at the interface. It is in a certain sense comparable to the well-known ghost fluid method. The model for surface tension is included in the microscale solver. It therefore takes its effect on the macroscales via the numerical fluxes, which eliminates the need for volume source terms to apply the surface force, as for example in the continuum surface force approach.

The DG scheme allows high orders of accuracy with subcell resolution, which is particularly valuable for the calculation of interface curvature that can be obtained from the derivation of the ansatz polynomials for the level set variable. However, high polynomial degrees are usually combined with large grid cells - the error introduced through the assumption of the interface coinciding with element faces therefore becomes large in configurations, where the DG scheme is most efficient. To avoid this contradiction, an adaptive grid refinement strategy is used in the vicinity of the interface. This framework allows to replace the high-order spectral elements by zones with very fine grids and first order accuracy. This is limited to the flow equations near the interface, where it increases the accuracy of the interface treatment and in addition guarantees oscillation-free results in the presence of shocks. At the same time, the level set remains unrefined, retaining the ease of curvature calculation from polynomial data.

The method has been validated for one-dimensional tests such as multiphase shock-tube problems in the past. We present several three-dimensional tests. For a quantitative validation, the interaction between a droplet or a bubble and a concentric shock wave are simulated in 3D and compared to well-resolved reference solutions obtained with a 1D solver assuming spherical symmetry. In addition, we present a 3D interaction between a planar shock and a spherical droplet, which shows the typical shock reflections and deformations in- and outside the droplet.

*Corresponding author: felix.jaegle@iag.uni-stuttgart.de

A pressure-based numerical method for the simulation of compressible two-phase flow.

M. Boger*, F. Jaegle, S. Fechter, C.-D. Munz
Institute of Aerodynamics and Gas Dynamics, University of Stuttgart, Germany
{boger,jaegle,fechter,munz}@iag.uni-stuttgart.de

Abstract

Our motivation is the extension of an originally incompressible multi-phase flow solver to the compressible regime. We aim at the simulation of evaporating droplets under extreme ambient conditions of high pressure and temperature where the compressibility effects inside the liquid phase can no longer be neglected. As a consequence, the flow equations require additional thermodynamic conditions given by an equation of state (EOS). This leads to stronger coupling effects at the material interface, which can cause spurious oscillations in the vicinity of the interface that have to be dealt with numerically by a special interface treatment. In order to stay as close as possible to the original incompressible numerical method, a pressure-based algorithm for the compressible multiphase flow simulation is chosen. Therefore, as it is the case for incompressible methods, pressure is used as primary variable. In a first approach, viscosity is neglected and we restrict ourselves to the compressible Euler equations. To circumvent their singular incompressible limit, the multiple pressure variables (MPV) scheme for compressible and incompressible flows is used in a conservative formulation. It is based on an asymptotic expansion of the pressure in terms of a global flow Mach number parameter M

$$p(x,t) = p^{(0)}(t) + M^2 p^{(2)}(x,t). \tag{1}$$

The pressure is split into a background pressure $p^{(0)}$ that formally satisfies the EOS in the incompressible limit case $M = 0$ and a hydrodynamic pressure $p^{(2)}$.

The above pressure decomposition is inserted into the Euler equations and the energy equation is reformulated in terms of pressure and kinetic energy using an appropriate EOS. The flow equations are discretized in a semi-implicit manner, similar to incompressible schemes. The spatial discretization is carried out on a Cartesian staggered grid.

With respect to the extension of the scheme to multi-phase flows, the material interface has to be tracked. This is done with a level set approach. Furthermore, the ideal gas EOS and the Tait EOS for liquids are used to describe the thermodynamic behavior of the different materials. It is well-known from the literature that density-based numerical schemes can suffer from unphysical spurious pressure and velocity oscillations at the interface location. Often complex interface treatment techniques are required to control this spurious behavior. It can be shown that our pressure-based scheme prevents this kind of oscillations due to the fact that pressure is used as primary variable in combination with an adequate spatial discretization. The material interface can therefore be treated as a contact discontinuity, which avoids more complex interface treatments like the ghost fluid technique.

The numerical scheme proves to give excellent results for one-dimensional multi-material shock tube test cases. The corresponding results agree very well with the exact, analytical solution of the considered Riemann problems. The method has also been assessed for three-dimensional flows such as the simulation of a planar shock wave moving through air and finally impinging on a water droplet. The results show typical wave structures inside and outside the droplet. The wave propagation inside the droplet is clearly visible including wave reflections at the gas-water interface.

The scope of the ongoing work is directed to further enhance the tracking and the resolution of the interface. At present, the discretization of the level set transport equation with a high-order discontinuous Galerkin method is investigated. A more sophisticated and accurate interface tracking scheme is a prerequisite for the inclusion of additional physical phenomena like surface tension to our simulation framework. Moreover, we plan to consider the effects of viscosity, extending the approach to the Navier-Stokes equations.

---

*Corresponding author: boger@iag.uni-stuttgart.de

A Mesoscale Study of Pinch-off under High Strain

M. Arienti[1,*], X. Li[2]
[1]Sandia National Laboratories, Livermore, 94550 CA, USA
[2]United Technologies Research Center, East Hartford, 06108 CT, USA
marient@sandia.gov and lixy2@utrc.utc.com

Abstract

We study the dynamic behavior of a pinching liquid thread as a function of the time to pinch-off. The novelty of this work resides in the use of a particle method (the Many-body Dissipative Particle Dynamics method, or MDPD), and in the inclusion of the interaction with a surrounding gas. As an MDPD calculation can be carried out at a scale below the continuity limit as a coarse-grained molecular simulation, this work represent a mesoscale approach to the study of spray formation.

Two issues are discussed in this paper. First, adding the effect of a second MDPD fluid requires the characterization of the friction interaction between particles of two immiscible fluids: unlike interfacial tension or solubility, this parameter does not have a directly related physical property. Second, in order to subject the liquid thread to a straining field, a two-phase, Non-Periodic Boundary Condition (NPBC) needs to be implemented. In the proposed NPBC method, two layers of particles are built into the domain on each side of the computational box. The outermost layer is modified at every iteration by placing particles of the prescribed type: this buffer works as a barrier whose composition depends on the instantaneous location of the boundary. The innermost layer contains thermalized particles that are otherwise free to move according to the distribution of the surrounding particles.

By enabling the simulation of pinch-off under extensional flow, an arbitrary strain rate can be imposed via the gas phase. The capillary number Ca therefore appears as an additional parameter controlling pinch-off, and the simulations illustrate the role of stochastic effects for a range of Ca values.

* Corresponding author: marient@sandia.gov

Progress with the analysis of dimethyl ether sprays with a moments spray model

N. G. Emekwuru
Midlands Simulation Group, School of Technology, University of Wolverhampton, United Kingdom.
n.emekwuru@wlv.ac.uk

Abstract

The representation of spray characteristics in internal combustion engines using spray models has taken on greater significance with the increased need to reduce engine emission levels. Apart from work on improving existing diesel and gasoline engines, research on evaluating other fuel types, including dimethyl ether, for internal combustion engines has intensified. Thus, existing spray models, developed for diesel engine spray assessments for example, have to be applicable to the new fuel types being investigated. A diesel spray model has been developed that represents diesel sprays by means of three moments of the droplet-size distribution function calculated from transport equations and one moment obtained from a Gamma size distribution. Typically, diesel spray models use the so-called Discrete Droplet Model (DDM) method in which the diesel sprays are represented by tracking droplet size classes. To attain stochastically significant solutions with the DDM method can require tracking a large number of groups of droplets and this can be computationally expensive. As droplet size classes are not tracked in the moments spray model, the computational intensity is reduced. The applicability of the moments spray model is analysed for dimethyl ether sprays. The results are characterized by dimethyl ether spray penetration at different injection nozzle sizes and spray pressure values. These are compared with experimental data. The results indicate that the moments spray model can be applied as a predictive tool for dimethyl ether spray penetration. This is similar to observations from other scholars using the DDM method. However, there are discrepancies in the observed and predicted penetration values at early and late injection times. Thus, it might still be useful to develop fuel initial injection correlations specific to dimethyl ether spray simulations.

Numerical Study of Dense Turbulent Sprays using a Coupling of the Direct Quadrature Method of Moments with an Eulerian Multi-Size Moment Model

W. Gumprich*, A. Sadiki
Institute of Energy and Powerplant Technology,
Graduate School of Computational Engineering,
Technische Universität Darmstadt, Germany
gumprich@ekt.tu-darmstadt.de and sadiki@ekt.tu-darmstadt.de

Abstract

The polydisperse spray character of the injected liquid fuel predominantly influences the mixing of oxidizer and fuel vapour and, consequently, the flame structure in combustion processes, considerably affecting combustion efficiency and emissions in IC-engines and gas turbines. Especially due to increasing fuel prices and stronger emissions regulations, there is a need for a reliable numerical tool to accurately describe and optimize the dense spray behavior in modern engines.

This contribution aims at including droplet-droplet interactions into an Eulerian framework using the Direct Quadrature based Sectional Method of Moments (DQbSMOM), a novel hybrid approach which combines the Direct Quadrature Method of Moments (DQMOM), proposed by Fox et al. [1], with a Sectional Method (SM). This is made possible by approximating the number density function (NDF) through the Maximal Entropy formalism in order to calculate the droplet coalescence term, and by adopting the Eulerian Multi-Size Moment model (EMSM), proposed by Massot et al. [2], to describe evaporating droplet polydispersity. The EMSM model provides not only an accurate prediction of the evaporative flux, but also the possibility to couple DQMOM with a SM, once the moment flux between two sections can be calculated. The major advantage of this hybrid approach is a higher accuracy related to convective transport and drag. Among the advantages of the Eulerian approach are a lower computational cost through optimal parallel computing and a straightforward liquid-gas phase coupling.

To assess the designed tool, numerical results are compared to Phase Doppler Anemometry (PDA) measurements of a hollow-cone water spray as experimentally investigated by Rüger et al. [3]. The experiment provides comprehensive validation data that include gas velocities, droplet size distribution and droplet velocities. Turbulence of the disperse phase is captured by two different k-ϵ based models. Transient simulations using the standard DQMOM and the DQbSMOM with different number of sections were performed to assess the predictability of the suggested numerical model.

Satisfactory agreement is observed with increasing number of sections, allowing a more accurate prediction of droplet dispersion. Figure 1 illustrates that, by considering just 3 sections, the results are considerably more accurate than the standard DQMOM, with one section. The importance of droplet coalescence can be seen in figure 2.

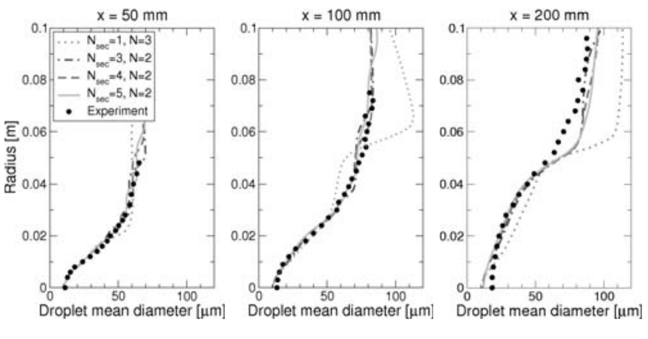


Figure 1. Droplet mean diameter over different measurement profiles in radial direction.

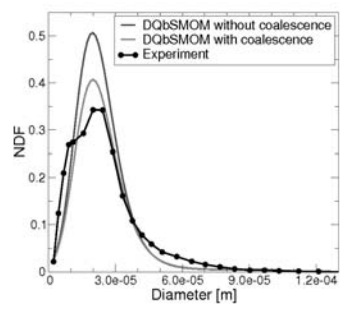


Figure 2. Local droplet diameter distribution at (x,y) = (50mm, 10mm).

References

[1] Fox, R. O., Laurent, F., Massot, M., *Journal of Computational Physics* 227:3058-3088 (2008).
[2] Massot, M., Laurent, F., Kah, D., de Chaisemartin, S., *SIAM J. on Applied Mathematics* 70:3203-3234 (2010).
[3] Rüger, M., Hohmann, S., Sommerfeld, M., Kohnen, G., *Atomization and Sprays* 10:47-81 (2000).

*Corresponding author: gumprich@ekt.tu-darmstadt.de

Development of Low-Order Regression Models for Selected Flat Spray Characteristics

K. M. Bade*, K. Cronce, R. J. Schick
Spray Analysis and Research Services, Spraying Systems Company, Wheaton IL 60187, USA
Kyle.Bade@Spray.com, Keith.Cronce@Spray.com, Rudi.Schick@Spray.com

Abstract

Dimensional as well as non-dimensional independent variables are used for predicting spray droplet size statistics and spray pattern distributions. The independent variables are: pressure, flow rate, spray angle, spray distance, viscosity, and surface tension. In the first approach, the dimensional values are used to develop a regression model for the general spray characteristics of interest; namely, Sauter mean diameter, spray width, and spray distribution profile shape. Secondly, a similitude approach is employed to generate dimensionless quantities from the independent variables, which are then used in formulating regression models. The similitude method allows for an assessment of the underlying balance of forces, which ultimately serve to form the spray characteristics. The final model development uses linear, or nonlinear where advantageous, regression models to fit the operational and rheological inputs to the spray characteristic outputs. The predictor models using dimensionless quantities showed a improved accuracy over the dimensional models. Additionally, the included number of dimensionless quantities is systematically reduced, revealing the most influential independent variables for each output. It is found that the Reynolds, Weber, and Froude Numbers are most influential. All models are developed for a particular nozzle, which limits the resulting models to this particular nozzle; however, the model *development process* has further reaching utility.

---

* Corresponding author: Kyle.Bade@spray.com

Experimental and theoretical investigations of Twin-Jets

F. Durst[1], Y.Han, A.Handtmann, M.Zeilmann
FMP TECHNOLOGY GMBH, Albert Handtmann Holding GmbH & Co. KG
info@fmp-technology.com, y.han@fmp-technology.com, info@handtmann.de,
m.zeilmann@fmp-technology.com

Abstract

Twin-Jet sprays have been extensively studied and numerous publications exist that document their properties. This yields to the finding that the theoretical treatment, forwarded by Dombrowski & Johns [1], is often used to provide an insight into the physics of Twin-Jet sprays. The theory in [1] deals, however, with an aerodynamically stretched and excited plane liquid lamella, a spray production process that does not apply to the Twin-Jet sprays. The authors forward the theoretical treatment of Twin-Jet sprays. Based on experimental observations, a theory applicable to high pressure Twin-Jet sprays is developed that is also extended to low pressure Twin-Jets by experimentally based correlations using dimensional analysis, to yield the Sauter-mean-diameter and the liquid jet lengths, which agree well with the experiments. The results are generalized by providing them as Oh-Re-correlations. The latter can be used to lay out Twin-Jet sprays for different applications, i.e. for designing Twin-Jet injectors for Otto- and Diesel-engines.

The liquid sheet formed by two identical impinging jets breaks down due to two principal mechanisms. Firstly, the interaction between liquid and ambient air produces super-imposition of aerodynamic waves. When the wave length is greater than the jet radius, the sheet will break up into liquid ligaments and further into fine droplets. Secondly, the impact of two jets results in hydrodynamic waves starting from the impinging point. These waves cause the sheet to disintegrate into bands of drops, particularly at high jet velocities. The detailed mechanism for hydrodynamic breakup has not been clarified until now. However, it can be determined that both mechanisms of disintegration of a sheet are dependent upon the jet parameters like impinging angle, liquid density, viscosity, surface tension, jet velocity and jet diameter. Therefore, the mechanism of disintegration can be summarized in a function of Reynolds number, Ohnesorge number and impinging angle.

Starting with the assumption that the region of drop formation by two inclined, interacting jets decreases to the periphery of a small ellipse, the mass flow of the two impinging jets are set to be equal to the mass flow out of the periphery. This yields to an equation for the thickness d_L of the lamella by assuming that the velocity of the fluid in the lamella is equal to the initial jet velocity

$$d_L \approx \frac{D}{3/2(1+1/\sin\theta) - \sqrt{1/\sin\theta}}$$

A ligament is only formed out of the liquid lamella if the fluid momentum exceeds the surface tension force. By taking also the viscosity into account one can derive

$$f_3\left(\frac{d_{32}}{D}\right) \geq f_1(\theta)\, f_2(Oh, Re)$$

Finally, by applying a dimensional analysis, the Sauter-mean-diameter of the resultant spray droplets can be obtained as

$$\frac{d_{32}}{D} = C \cdot f_1(\theta) \cdot f_2(Oh, Re)$$

Reference

[1] Dombrowski N. and W. R. Johns, The aerodynamic instability and disintegration of viscous liquid sheets, Imperial College, London, S.W.7.

[1] Corresponding author: info@fmp-technology.com

Improvement of Atomization of High Viscosity Liquid through Injection Rate Modulation

A. Azetsu*, G. Kobayashi
Dept. of Mechanical Engineering, Tokai University, Kanagawa, Japan
azetsu@keyaki.cc.u-tokai.ac.jp

Abstract

The improvement of atomization of high viscosity fuels such as heavy fuel oil is one of the most important issues in the field of marine diesel engines, since the quality of heavy fuel oil used in this field is becoming worse, whereas the requirements of cleaner exhaust emission is becoming more stringent. In the field of diesel engines for automobiles, fuel injection pressures have been increased up to 200 MPa to improve the atomization characteristics, which have successfully improved the combustion and emission performances of recent engines. However, it is difficult to use such high injection pressures in marine diesel engines as they will reduce the reliability of the injection pump and nozzles. Consequently, an alternative technique for improving the atomization other than the high pressure injection is required. Against this background, the authors are trying to improve the atomization characteristics of high viscosity liquids by employing high-frequency modulation of the fuel injection rate, i.e., periodical fluctuation of the injection rate. In previous studies, the authors examined the effect of injection rate modulation on the spatial dispersion of fuel droplets and the inner structure of the fuel spray under atmospheric condition, and found that the spray becomes wider with the increase of modulation amplitude and with the increase of modulation frequency in the case of spray with diesel oil. Following on from these studies, the present study investigates the effect of injection rate modulation on the spatial dispersion of droplets of high viscosity silicon oil.

The fuel injection system used in the present study is an electronically controlled accumulator type fuel injection system. In this system, the pressure pin of a conventional diesel nozzle was extended to attach a piezoelectric actuator by which to control the movement of pressure pin directly, enabling the fuel injection rate to be set arbitrarily. The nozzle used is a single hole type one with the hole diameter of 0.24 mm. The main experimental parameters were the modulation frequency, the modulation amplitude, the viscosity of the liquid, and the ambient pressure. The liquids used in this study were silicon oils with kinematic viscosities in the range of 10 to 30 cSt to simulate the viscosity of heavy fuel oil. The fuel injection system was installed on top of a constant volume high pressure vessel with two large observation windows. The fuel was injected downward into the vessel. The pressure inside the vessel was varied from atmospheric pressure to 1.5 MPa. The spray was illuminated by light from a metal halide lamp and photographed by an ICCD camera. To obtain an image of the averaged spray shape, the exposure time of ICCD camera was set to 2.5ms. To visualize the instantaneous images of spray, the ICCD camera was used in high-speed photography mode with an exposure time of 10 μs. From the systematic experiments it is explored that applying injection rate modulation improved the atomization characteristics of a viscous liquid. The spray shape becomes wider and the spray angle becomes larger with increasing amplitude and frequency of the injection rate modulation. This tendency was observed at different ambient pressures, although the atomization was improved as the increase of the ambient pressure. The mechanism of this phenomenon is discussed based on the temporal movement of the droplets cloud visualized by the high speed camera. It is found that interactions between injected fuel droplets close to the nozzle, such as catching up and overtaking motions, produces the expanding motion of fuel droplets perpendicular to the spray movement, resulting in a wider spray.

---

* Akihiko Azetsu: azetsu@keyaki.cc.u-tokai.ac.jp

Pneumatic Nozzle with Modified Internal Mixing Geometry

B. Schlinge*, A. Mescher, P. Walzel
Laboratory of Mechanical Process Engineering, Technical University of Dortmund, Germany
britta.schlinge@bci.tu-dortmund.de, axel.mescher@bci.tu-dortmund.de and
peter.walzel@bci.tu-dortmund.de

Abstract

Twin-fluid atomizers with internal mixing operation require less energy and provide small drop sizes compared to orifice flow cross-sections. For small drops a good mixing performance of the two phases in front of the orifice is necessary. In commercial nozzles a gas distributor usually disperses the gas into the mutual continuous liquid phase. The newly designed nozzle** is based on the distribution of the liquid into the gas flow by capillaries. The liquid forms jets at the end of the capillaries and lead to their extension by the coaxial gas stream. A continuous liquid flow at the nozzle diameter is always ensured leading to a continuous spray without pulsation within a wide operational range. Drop size distribution and pulsation frequencies of this nozzle design in comparison to conventional nozzle are presented.

A large number of pneumatic nozzles mainly effervescent nozzles with different design are known. In this new nozzle type the disperse air phase is introduced into a mutual continuous liquid phase through holes inside the mixing chamber. These nozzles show good spray performance for low ALR ($ALR = \dot{m}_{gas}/\dot{m}_{liquid}$) [1, 2] compared to external mixing nozzles. For high and very low ALR however, the spray tends to become unstable or pulsating [3, 4] and the assumption of a continuous liquid phase may be questioned. The flow regime inside the nozzle obviously becomes unstable, i.e. the void fraction of liquid and gas within the orifice fluctuates.

Liquid is sprayed by compressed air with a new twin-fluid atomizer as shown in Figure 1. The gas enters through the two side ports into the mixing chamber. The liquid is introduced through seven capillaries with diameters of 0.5 mm in contrary to [5] with only one capillary. At the end of the capillaries jets are formed, which are elongated by the pressure field up to the orifices with diameters in a range of 1 to 2 mm. Due to the flow pattern, the mean liquid flow rate is widely constant over the cross-section. An unstable or pulsating spray is avoided reducing the span of the sprays. The spray data are compared to those obtained with a conventional nozzle with equal orifice diameters and comparable flow rates. The measurement program includes the drop size distribution and the pulsation frequencies observed at unstable process conditions. For the liquids a viscosity range of 1 to 300 mPas is considered. Water and mixtures of water and glycerol as well as aqueous solutions of polyvinylpyrrolidone (PVP) are sprayed. The observed operating parameters cover pressures up to 0.5 MPa and ALR in the range of 0.1 to 10.

In the first experiments with low viscous liquids like water the drop size distribution shows quite similar results as the conventional nozzle, shown in Figure 2. Benefits of the spray characteristics are visible at high ARL as the pulsation of the spray is significantly less for the capillary nozzle in comparison to the conventional nozzle. Further advantages of the capillary nozzle are evident for higher liquid viscosities.

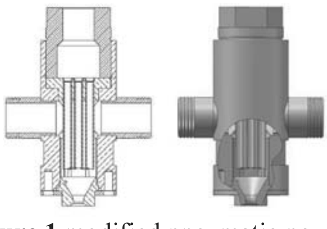


Figure 1 modified pneumatic nozzle

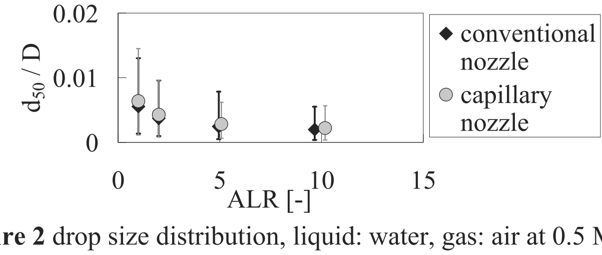


Figure 2 drop size distribution, liquid: water, gas: air at 0.5 MPa

[1] Sutherland, J. J., Sojka, P. E., Plesniak, M. W., *International journal of multiphase flow*, 23 (5): 865–884, (1997)
[2] Lörcher, M., Schmidt, F., Mewes, D., *Atomization and Sprays*, 15:145-168, (2005).
[3] Luong, J. T. K., Sojka, P. E., *Atomization and Sprays*, 9 (1): 87-109, (1999).
[4] J. Schröder, M. Schlender, P. E. Sojka, V. Gaukel, H. P. Schuchmann, *ILASS-Europe*, Brno, CZ, 2010.
[5] S. Groom, PhD Thesis, TU Dortmund, 2006.

* Corresponding author: britta.schlinge@bci.tu-dortmund.de
** German patent application DE 102010 012 555.5-51

Diesel spray velocity and break-up characterization with dense spray imaging

D. Sedarsky*, S. Idlahcen, K. Lounnaci, J.B. Blaisot, C. Rozé
CORIA, University of Rouen, France
david.sedarsky@coria.fr

Abstract

High pressure diesel sprays often exhibit large variations in structure, breakup length, cone angle, droplet and structure size, and overall morphology which cannot be directly attributed to or characterized by the main injection parameters (pressure, orifice geometry, fluid properties, etc). In addition, the large optical depth of the near-field and the sensitivity of the spray formation to disturbances in the flow field effectively prevent the application of conventional imaging or velocity diagnostics in many regions which are important for understanding spray formation. However, a number of emerging optical techniques have demonstrated that specialized optical measurements can be applied in the near-field, despite the severe attenuation and scattering noise endemic to dense spray regions.

Ballistic imaging (BI) is a technique which selectively attenuates light transmitted through the spray to form a 2-D spatial intensity dominated by light which has been minimally distorted by scattering interactions. The primary effect of the filtering is to increase the image contrast, while decreasing overall signal intensity.

Ultrafast shadow imaging (USI) is a modified shadowgraphy arrangement, which takes advantage of intense ultrashort laser illumination and the efficiency of a set of collection optics to create a spatially resolved 'shadow image'. This arrangement trades the sensitivity of a shadowgram for a high-resolution, spatially resolved view of very strong refractive index gradients in the object plane of the collection optics.

This work presents quantitative methods to categorize breakup morphology in a diesel spray produced by a single-hole, plain orifice diesel injector issuing into ambient atmospheric conditions. Velocity data and high-resolution images of the diesel spray were obtained using both time-gated BI and high-resolution USI measurements. The USI results provide high-resolution visualization of the spray edges and resolved droplets within the depth-of-field of the collection optics, while the BI results provide a view of the spray at a modified dynamic range which is unbiased by internally refracted light (caustics) and multiple-scattering noise, revealing additional spatial information.

Time-correlated image-pairs obtained by both techniques were filtered and cross-correlated on a variety of scales to produce velocity profile data and identify structures and features which differentiate the breakup modes observed in the diesel spray over a variety of injection pressures.

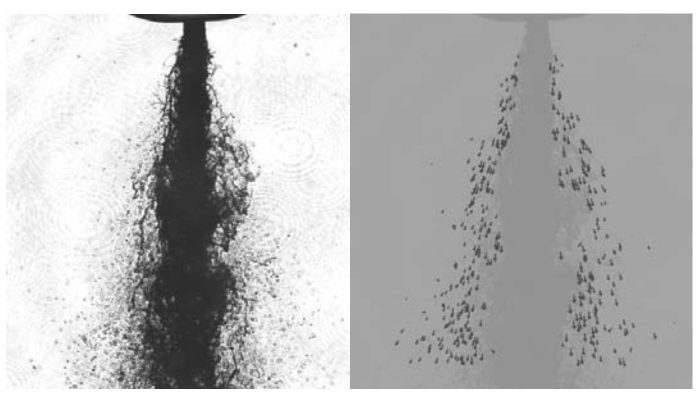


Fig 1. Diesel spray (400 bar injection pressure, 100µm plain orifice nozzle) shadow image and velocity vectors calculated from normalized cross-correlation of time-resolved image-pairs.

* Corresponding author: David.Sedarsky@coria.fr

Optical measurements of bubbles and spray in wind/water facilities at high wind speeds

W. Mischler*, B. Jähne
HCI & IUP, University of Heidelberg, Germany
wolfgang.mischler@iwr.uni-heidelberg.de and bernd.jaehne@iwr.uni-heidelberg.de

Abstract

An optical imaging technique for the measurement of the size and velocity distribution of bubbles and droplets is presented. This method was used in a wind-wave tank at high wind-speeds to investigate bubble populations and spray generated by breaking water-waves. These measurements help to model bubble and spray induced gas exchange between atmosphere and ocean.

The setup consists of a 2040 × 1088 CMOS-camera (Basler acA2000-340km, 5.5 µm pixel size), which is used in a telecentric bright-field setup. Thus bubbles or droplets between the light source and the camera become visible in the images as dark disks because of light scattering. For illumination, a single high-power LED (Cree XP-E) in a telecentric illumination system is used. This LED is operated in a pulsed mode with pulse lengths smaller than 2 µs. This is the effective exposure time. In order to acquire two images with a small temporal distance, the camera is set to maximal exposure time. The first image is recorded by flashing the LED at the end of the first exposure and the second image by flashing at the beginning of the second exposure. The effective time difference of consecutive exposures was between 1 µs and twice the exposure time of the camera. The telecentricity (only rays parallel to the optical axis contribute to the images) of the optics reduces the error of the size estimation of the bubbles/droplets. In addition the position along the optical axis can be estimated by a depth from defocus method. With this information it is also possible to determine the measuring volume to estimate the bubble/spray-number density. The size range of this technique is determined by the pixel and sensor size and by the resolution of the lens. Here, a lens with a working distance of 135 mm and a magnification of 0.37 is used, so that one pixel corresponds to 15 µm in object space. With this setup a radius range of 40 µm - 5000 µm can be imaged. The main source for the error of the radius is the blurring of the edges due to defocus. Therefore the accuracy depends on the quality of telecentricity and complexity of image processing. For bubbles with a radius larger than 10 px the error is below 1 % when using simple image processing. This error can be reduced significantly by using elaborate image processing, which is also able to detect bubbles as small as 3 px in size. Different image processing algorithms for spherical objects are presented and investigated regarding accuracy. Bubbles or droplets which show deviation from sphere shape introduce problems for the image processing. For bubbles in still water, such deviations are typically found for radii of greater than 500 µm.

Capabilities and limits of the technique are addressed on the example of two experiments. Test measurements were conducted at the high wind speed wind-wave tank in Kyoto, Japan at a wind speed of 42 m/s. This experiment showed that distributions of bubbles can be measured at high wind speeds with no problems in regions, which are not near to the water surface (> 2 cm). For regions close to the interface the image processing algorithms need to be adjusted, since portions of the illumination can be occluded by waves. It is shown with an experiment in the small linear wind/wave facility in Heidelberg, that this technique also works for spray measurements. Systematic measurements of spray could not be conducted in Kyoto, since the walls of the tank were covered by a thin film of water caused by the spray, which heavily disturbed the optical path.

---

*Corresponding author: wolfgang.mischler@iwr.uni-heidelberg.de

Simultaneous Measurement of Evaporating Droplet Diameter Using Phase Doppler Anemometry and High-speed Camera

N. Kawahara[1*], E. Tomita[2], H. Okano
Okayama University, JAPAN
[1] kawahara@mech.okayama-u.ac.jp, [2] tomita@mech.okayama-u.ac.jp

In order to understand the spray characteristics formed by PFI injector, many investigations of gasoline injection sprays using several measurement techniques like laser sheet method with high-speed camera, laser-induced (exciplex) fluorescence (LIF), laser and phase Doppler anemometor (LDV/PDA), particle image velocimetry (PIV) have been carried out for better control of spray and combustion characteristics. However, one of the key processes affecting spray behavior is the primary spray break-up, because it defines the starting conditions for the spray distribution, the evaporation process, and mixture formation. Liquid fuel forms the liquid column very close to the nozzle exit, and then makes liquid ligament and droplets due to the break-up of liquid ligament. Droplets, which were formed by the break-up of liquid ligament, were non-spherical shape. Therefore detection of non-spherical droplets was needed to understand break-up of liquid ligament. Some of the authors discussed primary fuel break-up very close to nozzle exit using high-speed video camera with long-distance microscope. Jet breakup and droplets breakup have been visualized using photographic method. Droplet stroboscope techniques, which are good for freezing the movement, have been usually used for droplet deformation process and droplet breakup. On the other hand, Dual-PDA system can measure the non-spherical droplets. Dual PDA system combines two-detector standard PDA and a planer PDA. In the standard PDA, the detectors are arranged at an off-axis angle perpendicular to the plane of the transmitting beams, on the other hand, in the planar PDA, the detectors are put in the same plane of transmitting beams. This combination allows us to measure the sphericity of droplets.

The purpose of this study is to investigate measurement accuracy of evaporating droplet (ethanol) diameter and sphericity measured by Dual PDA with comparing to visualization result using CCD camera with long-distance microscope. Measurement accuracy of PDA is not unclear in primary atomization region of PFI injector and non-spherical droplet. Dual PDA can measure droplet's vertical and horizontal sphericity. Simultaneous measurement of Dual PDA and visualization using CCD camera with long-distance microscope is necessary for understanding of measurement accuracy of Dual PDA. Single ethanol droplet is formed by ultrasonic levitator (vibrating frequency: 60 kHz). Levitated droplet is evaporated in time. Measurement accuracy of droplet diameter and sphericity of evaporating ethanol droplet using Dual PDA is discussed. Three main conclusions can be drawn in this research. Dual PDA can measure the droplet diameter of evaporating droplet avoiding 2π ambiguity. Measurement error due to 2π ambiguity in Dual PDA can be understood for larger droplet over maximum measurable droplet diameter. It is possible to measure droplet sphericity using Dual PDA. However, further consideration for more oblate droplet should be needed to discuss the measurement accuracy of droplet sphericity using Dual PDA precisely.

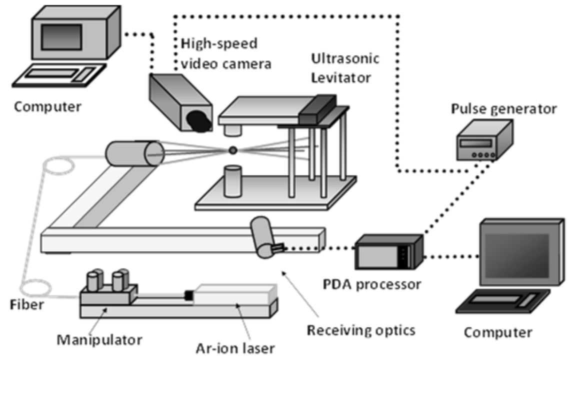


Fig.4 Experimental setup

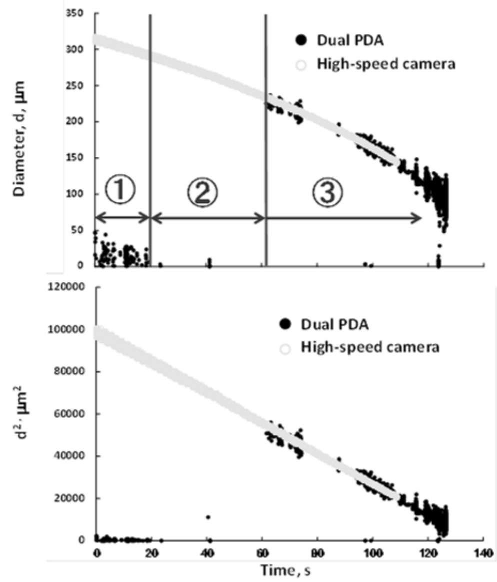


Fig.9 Time variation of droplet diameter under the condition #2

* Corresponding author: kawahara@mech.okayama-u.ac.jp

Droplet Dynamics under Extreme Boundary Conditions: The Collaborative Research Center SFB-TRR 75

B. Weigand *, C. Tropea**
* Institut für Thermodynamik der Luft- und Raumfahrt, University of Stuttgart, Germany, bernhard.weigand@itlr.uni-stuttgart.de
**Fachgebiet Strömungslehre und Aerodynamik, Center of Smart Interfaces, Technische Universität Darmstadt, Germany, ctropea@sla.tu-darmstadt.de

Abstract

Processes involving droplets play a central role in both nature and in engineering applications. While some of these examples and applications can be extremely complex, they can often be well understood in terms of very basic drop dynamic processes, which is also the first step to improvements and/or optimization. The Collaborative Research Center (CRC) SFB-TRR 75 was established in January 2010 to focus on research about such basic drop processes, but in particular on those processes involving extreme boundary conditions, for example, near thermodynamic critical conditions, very low temperatures, under strong electric fields or in situations involving extremely large gradients of boundary conditions.

Researchers from the University of Stuttgart, the Technische Universität Darmstadt and the DLR at Lampoldshausen participate in this CRC, coming from various departments, including Mathematics, Chemistry, Electrical Engineering, Mechanical Engineering and Computer Sciences. The goal is to gain a better physical understanding of the essential processes as a basis for new analytical and numerical descriptions, thereby leading to an improved prediction of large systems in nature or in technical applications. This contribution gives an overview of the projects being pursued at the SFB-TRR 75 and highlights scientific results from the first two years of operation. The main purpose of the paper is to familiarize colleagues with this extensive and dedicated research effort in the area of drop dynamics and to motivate and initiate future collaboration with others in the field.

Effects of Liquid Physical Properties for Liquid Bubble Breakup Due to Airstreams

Teruhiko Yoshida*1
*1: Department of Mechanical and Precision System Engineering, Teikyo University, Utsunomiya, Japan
yoshida@uccl.teikyo-u.ac.jp

Abstract

To reduce the greenhouse gases, bio fuels are developing as an alternative fuel instead of fossil fuels. As carbon dioxide and so on are the causes of global warming, low carbon combustion is one of the crucial issues to conserve the global environment. Liquid atomization technology should contribute to the field of fuel combustion and others, such as the cooling of exhausted warm water at geothermal power plants.. A liquid bubble contains gas in a liquid drop. So, it has a larger surface area than a liquid drop of the same mass. It could have the advantage for fuel combustion and cooling of exhausted warm water. In order to apply it in these areas, effects of liquid physical properties for liquid bubble breakup due to airstreams were investigated. Experiments were conducted using a horizontal air-suction-type wind tunnel. Uniformly sized liquid bubbles were produced. The breakup processes were precisely observed using a digital high-speed video camera.

Introduction

The liquid bubble has many important scientific applications such as the cooling of warm exhaust water, fuel combustion and flow visualization. If liquid bubbles are used to combust bio-ethanol fuel, biodiesel fuel (BDF), and glycerin which is a by-product of BDF, more effective combustion could be obtained than in the case of liquid droplets.

Effects of liquid physical properties for liquid bubbles breakup due to airstreams are investigated. The breakup pattern is impacted due to surface tension, viscosity. Several characteristic deformation and breakup patterns are observed. Some of deformations and breakups are never observed for liquid drops. Breakup ratio in the transition breakup and the breakup time were measured by analyzing the breakup processes recorded by the digital high-speed camera. As the airstream velocity increases, the number of broken-up liquid bubbles increase. And finally at a certain airstream velocity, all the liquid bubbles breakup. The breakup pattern that some liquid bubbles break but others do not is defined as a transition breakup. The relationship between airstream velocities and the breakup ratio is investigated for some liquid bubble sizes.

Results and Discussion

In Fig.1, one of the characteristic figures for liquid bubble deformation and breakup is shown. A small droplet is separated from a main liquid bubble, but a main body continues to keep a liquid bubble. The liquid of the liquid bubble is ethanol in water solution. This phenomenon, the separation of a small droplet from around a stagnation point of a main body, cannot be observed for a water liquid bubble. In the author's experiment on the droplet breakup, the phenomenon of a small droplet separation from a main liquid drop has not been observed. Other characteristic deformation and breakup types are observed. It is revealed that the deformation and breakup pattern of liquid bubbles differs greatly on the physical properties of liquid.

Figure 2 shows the relationship between time and normal diameter of liquid bubble. In one case, a liquid bubble repeats its expansion and depression with time like a sinusoidal wave but eventually it tends to return to its original shape. In another case, a liquid bubble continues to expand its size and finally breaks up.

Breakup conditions with airstream velocity and bubble diameter are investigated. Breakup conditions with Weber number and equivalent liquid bubble diameter are investigated as well. As the equivalent liquid bubble diameter increase, critical Weber number decreases.

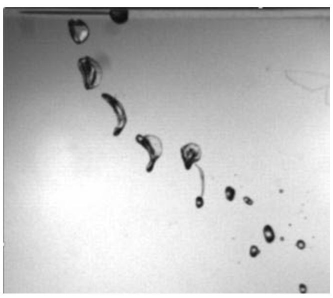


Fig.1 Separation of a small droplet from around a stagnation point of a main body

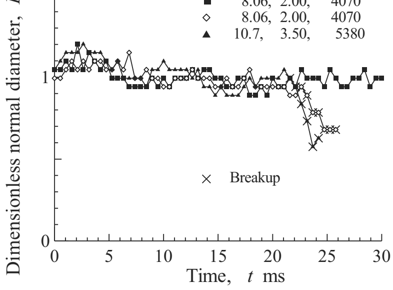


Fig.2 Relationship between time and normal diameter of liquid bubble

Aerodynamic Fragmentation of Drops: Dynamics of the Liquid Bag

L. Opfer*[1], I.V. Roisman[1,2], C. Tropea[1,2]
[1]Institute of Fluid Mechanics and Aerodynamics, Technische Universität Darmstadt
[2]Center of Smart Interfaces, Technische Universität Darmstadt, Germany
lopfer@sla.tu-darmstadt.de, roisman@sla.tu-darmstadt.de and ctropea@sla.tu-darmstadt.de

Abstract

Aerodynamic breakup of drops and ligaments is one of the most important phenomena responsible for the atomization of bulk liquid in single fragments. It occurs when the relative velocity between drops or ligaments and the dispersed phase exceeds a certain limit. The aerodynamic breakup is responsible for the final spray characteristics in a variety of technical applications and everyday phenomena, such as: fuel injection in jet engines and internal combustion engines, coatings and rain. Due to its importance the aerodynamic fragmentation has been interest of research for some decades. Recent comprehensive reviews about this topic can be found in Guildenbecher et al. [2009] and Theofanous [2011].

The development and outcome of the aerodynamic breakup mainly depends on the gas Weber number. It can be subdivided in particular modes, which are widely known as bag, multi-mode, sheet-thinning and catastrophic breakup. Among these, the bag breakup is the dominant fragmentation mode for most applications, since it occurs at relatively low Weber numbers.

While the qualitative development and morphology of single bag breakup events is well understood, the exact physics that lead to deformation and breakup of the drops are still unclear. Especially the mechanism that leads to bursting of the bag is not understood up to now. This study is devoted to the further experimental and theoretical investigation of bag breakup phenomena.

A compact open-circuit wind tunnel has been designed in order to conduct the experiments. The test section is made of acrylic glass which provides access for optical measurement techniques. A high-speed video system equipped with microscopic lens is used for shadowgraph visualizations with high temporal and spatial resolution. A typical example for the evolution of a bag breakup is depicted in figure 1. The experiments are carried out for various Weber and Ohnesorge numbers.

(a) spherical primary droplet | (b) drop deformation | (c) formation of the liquid bag | (d) bag growth | (e) bursting of the liquid film

Figure 1. Typical evolution of a bag breakup

A qualitative analysis of the breakup evolution is performed by means of digital image processing. Algorithms are developed in order to track the leading and trailing side of the drop and growth of the liquid bag as well as its aspect ratio.

A new theoretical model is developed which describes the dynamics of the liquid bag. The governing equations of the bag motion are solved taking into account for the aerodynamic pressure, inertia and surface tension in the liquid bag. It is shown that the model predictions agree well with the experimental results.

References

Guildenbecher, D.R., López-Rivera, C. and Sojka, P.E., 2009. Secondary atomization. Experiments in Fluids, 46(3), p.371-402.

Theofanous, T.G., 2011. Aerobreakup of Newtonian and Viscoelastic Liquids. Annual Review of Fluid Mechanics, 43(1), p.661-690.

*Corresponding author: lopfer@sla.tu-darmstadt.de

LES of Single Droplet and Liquid Jet Primary Break-up Using a Combined Level Set/Volume of Fluid Method.

F. Xiao*, M. Dianat, and J. J. McGuirk
Dept. of Aero. & Auto. Eng., Loughborough University, UK.
F.Xiao2@lboro.ac.uk M.Dianat@lboro.ac.uk J.J.McGuirk@lboro.ac.uk

Abstract

An accurate method for prediction of liquid jet atomisation is of utmost significance in many industrial applications. The engineering driver of prime interest in the present work is the aeroengine gas turbine fuel injector. In this area, substantial experimental and computational research has been carried out to understand the first part of the atomisation process - primary break-up. However, numerical modeling of primary break-up of a liquid jet is very challenging, especially for high liquid/gas density ratio. Simulations can become unstable due to errors in dealing with discontinuous conditions across the interface. As a consequence, many published numerical simulations are limited to a liquid/gas density ratio less than O(100). Since the majority of liquid jet atomisation experiments are carried out at atmospheric pressure with high density liquids, quantitative comparison between numerical modeling and experiment is quite rare. The simulation reported by Li et al [1] applied the algorithm proposed by Sussman et al [2] to a liquid (Jet-A) jet in an air cross-flow and hence had a high (650) density ratio. Adaptive Mesh Refinement and the removal of under-resolved small liquid structures were necessary as the only experimental data available was far downstream; in spite of the advanced modelling, agreement with measurements was relatively poor. The current research is focussed specifically on the early stage of primary break-up and its objectives are: (i) to develop a Large Eddy Simulation (LES) methodology for liquid jet atomisation at high density ratio, and (ii) to validate the method against experimental data for: single droplet break-up, a liquid jet in a coaxial air flow, and a liquid jet in an air cross-flow.

A coupled Level Set and Volume of Fluid (CLSVOF) technique is adopted as the interface-tracking method. Benchmark test cases are used to demonstrate that the CLSVOF method conserves liquid mass and provides a superior interface representation, combining the advantages of LS and VOF methods. The pressure jump across the interface is treated using the ghost fluid approach. Since the density is discontinuous across the interface, momentum is also discontinuous. Thus, an appropriate numerical scheme is required to treat such discontinuities when discretising the momentum equations. A single set of momentum equations are solved to provide the local liquid/gas velocity depending on the local value of the Level Set. In the region of the interface, however, an extrapolated liquid velocity field is calculated, so that momentum errors are minimised; appropriate boundary conditions at the interface are imposed leading to stable solutions even for liquid/gas density ratio O(1000).

The current paper provides details on the developed method and uses three test cases for validation, all with water as liquid and air as gas. The break-up of a single liquid droplet in a uniform air flow has been studied extensively and experimental findings are well documented. Such a flow represents an appropriate first test case for validating the proposed two-phase modeling methodology. It is demonstrated that the numerical procedure leads to predictions that are consistent with experimental observations across the entire spectrum of Weber number. It is shown that the simulated droplet correctly undergoes oscillatory deformation, bag break-up, or sheet-thinning break-up depending on the Weber No. Characteristics such as break-up time, maximum cross-stream diameter and drag coefficient agree quantitatively with experimental data. The proposed method is then applied to simulate the break-up of a liquid jet in a coaxial air flow. Predicted break-up lengths at different air and liquid velocities agree closely with measured data [3], although for low Weber No. the turbulence conditions at injector nozzle exit require detailed modelling. Finally, the primary break-up of a liquid jet in a cross-flow has been investigated. The outer boundary of the liquid spray formed by the cross-flow is shown to agree closely with experimental results [4]. A detailed analysis of the mechanism involved in the primary break-up process is provided.

[1] Li X., Arienti M., Soteriou M., C., Sussman, M. M., AIAA-2010-0210, 48th AIAA Aerospace Sciences Meeting, Orlando, USA, 2010.
[2] Sussman M., Smith K. M., Hussaini M. Y., Ohta M., and Zhi-Wei R., Journal of Computational Physics 221-2: 469-505, 2007.
[3] Charalampous G., Hardalupas Y., Taylor A., Int. Jnl. of Spray & Combustion Dynamics 1-4: 389-415, 2009.
[4] El Shamy, O. M., Experimental Investigations of Steady and Dynamic Behaviour of Transverse Liquid Jets, PhD Thesis, Univ. of Cincinnati, 2007.

* Corresponding author: F.Xiao2@lboro.ac.uk

Development and analysis of a Lagrange-Remap sharp interface solver for stable and accurate atomization computations

V. H. M. Le Chenadec*
Department of Mechanical Engineering, Stanford University
Stanford, California, 94305, USA
vlechena@stanford.edu

H. Pitsch
Institute for Combustion Technology, RWTH Aachen
Templergraben 64, 52056 Aachen, Germany
h.pitsch@itv.rwth-aachen.de

Abstract

There exists a variety of computational models for treating multiphase flows, with increasing levels of complexity. In the direct numerical simulation (DNS) approach that is considered in the present work, the physics of the flow are all solved for directly. In the presence of complex topologies such as those encountered in atomization, the underlying equations are quite stiff, and therefore the development and improvement of numerical methods that treat these flows has been an area of active research. Among algorithms that have emerged, sharp interface methods including Volume-of-Fluid (VoF), Level Set (LS) and their combination (Coupled LS/VoF) have been shown to perform well on a variety of test cases. The application of these methods to realistic configurations such as primary atomization of a turbulent round jet is not common, and the rare convergence studies available show strong grid dependence. This suggests the need for a careful analysis of the numerical schemes predicting such flows.

The present work focuses on a Lagrange-Remap approach developped for free surface flows and involving two recent improvements which are presented in details: first, a second order unsplit Coupled LS/VoF interface capturing method designed for arbitrary grids [1], and second, a robust and conservative flow solver for incompressible Navier-Stokes equations in the presence of an liquid/gas interface [2]. The resulting approach is shown to be stable, and the interface capturing to be second order accurate. In addition, the scheme is shown to have excellent mass and momentum conservation properties.

As mentioned, these improvements rely on an unsplit geometric Lagrange-Remap approach. Firstly, the interface is advanced using an unsplit Coupled LS/VoF interface capturing method, therefore benefitting from the conservation properties of the VoF method and the geometrical accuracy of the LS method. In addition, the unsplit nature of the volume fraction advection offers meshing flexibility and reduces numerical errors known to be detrimental to split VoF methods. Secondly, the predictor and corrector steps used in the incompressible flow solver are designed so that the momentum advection and Ghost Fluid projection are carried consistently with the volume fraction transport. This improves momentum conservation and stability, even at high density ratio. Furthermore, the analysis of the polyhedral operations used during the transport of volume fraction and momentum will be presented along with a proof of the stability of the scheme.

Finally, the proposed method is applied to a series of round jet computations at realistic density ratio and moderate Weber/Reynolds numbers. Mean flow statistics, including liquid penetration length and drop size distributions, are used to quantify the impact of grid resolution in an attempt to characterize the grid dependence of the atomization process.

[1] V.H.M. Le Chenadec and H. Pitsch, A Hybrid Lagrangian-Eulerian Method for Multiphase Flow Computation, Journal of Computational Physics, *submitted*.

[2] V.H.M. Le Chenadec and H. Pitsch, A Robust and Accurate Lagrange-Remap Sharp Interface Solver for Multiphase Flow Computation, Journal of Computational Physics, *in preparation*.

*Corresponding author: vlechena@stanford.edu

Modified Level Set Equation for Gas-Liquid Interface and Its Numerical Solution

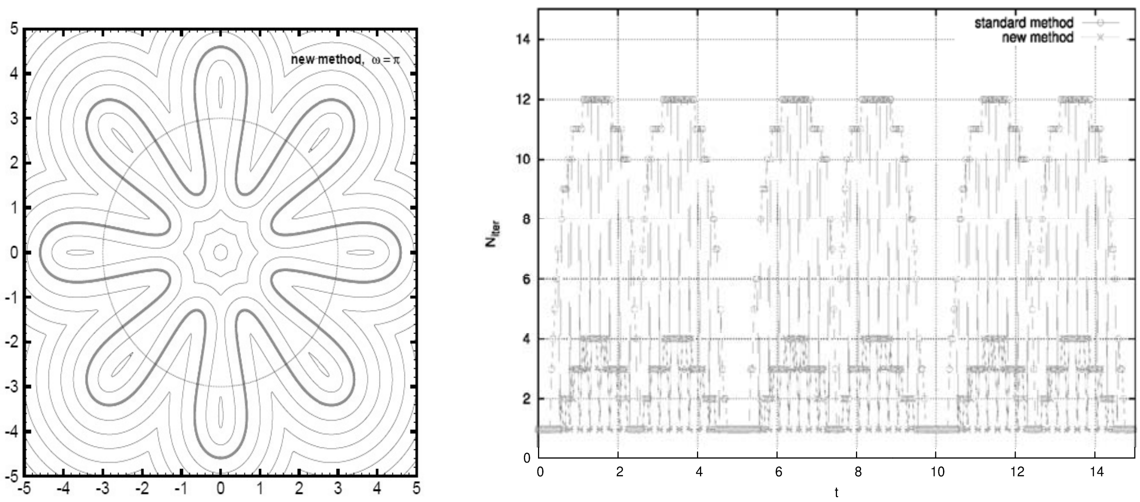
V. Sabel'nikov†, A. Yu. Ovsyannikov‡ and M. Gorokhovski*‡

† ONERA, DEFA/EFCA, France
‡LMFA, Ecole Centrale de Lyon, France

vladimir.sabelnikov@onera.fr
andrey.ovsyannikov@ec-lyon.fr, mikhael.gorokhovski@ec-lyon.fr

Abstract.

This paper is devoted to further modification of the Level Set approach, which is well-known for simulation of gas-liquid flows with the interface. In our development, we addressed to the case of a strong velocity gradient at the free interface. This is a typical situation, for example, when this interface interacts with the turbulent flow. In this case, the gradients of the level set scalar, in the vicinity of the interface, increase with time very rapidly. In order to maintain the accuracy of the numerical solution, the Level Set methods are combined usually with the Eikonal equation for a signed distance function from the zero level set. In the standard procedure (Sussman et al., in J. Comput. Phys. 114, 1994), in order to be consistent with evolutional type of the Level Set equation, the non-evolutional Eikonal equation is replaced by quasi-evolutional one, with the artificial time, providing iterations at each time step. Our idea is to modify the Level Set equation, in such a way that the Eikonal equation is satisfied directly by the form of the modified equation. This was done in the proposed paper. The efficiency of the proposed method is demonstrated by using various tests problems with interface.



Isolines of the level set function and the number of iterations for the standard method with reinitialization procedure and new methods proposed in this paper

* Corresponding author: mikhael.gorokhovski@ec-lyon.fr

Computational study of stratified charge compression ignition engines with late injection under low-load conditions

H. Zhang [a,*], E. R. Hawkes [a,b], S. Kook [b], W. Hwang [c]
[a] School of Photovoltaic and Renewable Energy Engineering,
The University of New South Wales, Australia
[b] School of Mechanical and Manufacturing Engineering,
The University of New South Wales, Australia
[c] GE Global Research, U.S.A
haoyang.zhang@unsw.edu.au, evatt.hawkes@unsw.edu.au,
s.kook@unsw.edu.au and hwang@ge.com

Abstract

The fuel stratification introduced by direct injection (DI) of iso-octane in an optically accessible homogeneous charge compression ignition (HCCI) engine is numerically investigated by using a multi-dimensional model. The primary purpose of the study is to provide an understanding of the effects of late DI on the in-cylinder fuel-air mixture distribution under low-load conditions. HCCI engines have the potential to provide high diesel-like efficiency with rather low harmful emissions of nitrogen oxides (NO_x) and soot. One of key hurdles, however, is the precise control of ignition timing and the rate of heat release or pressure-rise under a wide range of operating loads. At low loads, combustion efficiency is quite low and the associated carbon monoxide (CO) and unburned hydrocarbons (uHC) emissions are quite high. DI has the potential to improve the combustion efficiency and reduce emissions by introducing charge stratification to increase local combustion temperatures. To understand and optimise performance, detailed transient information about in-cylinder flow motions, fuel-air mixing, and mixture distributions is required.

In this study, the model is firstly validated by comparing the fuel distributions between calculations and experimental measurements made using planar laser induced fluorescence for three different injection timings (Hwang *et al.*, SAE2007-01-4130). The predicted fuel distributions at different injection timings show a good agreement with the measurements. It is found that with more retarded injection timing the fuel distribution is increasingly concentrated in the central region and less fuel in the crevice volume, corresponding to the potential improvement of the combustion efficiency and emissions of CO and uHC and a potential increase of NO_x emissions. The results emphasise the importance of the combustion efficiency versus NO_x trade-off and suggest that it is likely to be highly dependent on the details of the geometry and flow conditions. Generally, it is expected that the combustion efficiency/NO_x trade-off improves when fuel can be injected as late as possible with acceptable levels of NO_x emissions (Hwang *et al.*, SAE2007-01-4130). Therefore, techniques that can provide faster mixing have the potential for further improvement.

Then, the fuel distributions under different flow and injection parameters, such as swirl ratio, injection pressure and included angle of the spray, are studied. It is observed in the modelling that at a high injection pressure the trade-off between combustion efficiency and NO_x emission could be potentially improved. However, a high swirl ratio and a low injection pressure show adverse effects due to poor global mixing. The modelling results correctly match the phenomena observed in the experiments. Next, the influences of included angle of the spray are qualitatively investigated. It is shown in the modelling that with a high included angle, the spray shows prolonged vapour penetration and fuel mixture entering in the crevice volume even at late injection, leading to considerable reduction of combustion efficiency. For a low included angle of the spray, it is found that the global mixing was significantly slowed due to intensified wall impingement, resulting in a potential increase of NO_x emissions.

---

* Corresponding author: haoyang.zhang@unsw.edu.au; jerry.haoyang.zhang@gmail.com

Numerical study on the influence of simplified spray boundary conditions for the characterisation of large industrial safety spray systems used in nuclear reactors

J. Malet[1], A. Foissac[1,2], C. Vayaboury[1], T. Gélain[1], S. Mimouni[2]
[1] IRSN, PSN-RES/SCA Saclay, France
[2] EDF, MF2E, Chatou, France
jeanne.malet@irsn.fr; celine.vayaboury@irsn.fr, thomas.gelain@irsn.fr;
stephane.mimouni@edf.fr; arnaud.foissac@irsn.fr

Abstract

During the course of a severe accident in a Pressurized Water Reactor (PWR), hydrogen can be produced due to reactor core oxidation, leading to potential combustion and deflagration, as observed in Three Mile Island and Fukushima accidents. In some reactors, spray systems (over 500 nozzles) are placed at the top of the containment to prevent overpressure. Spray modelling is thus part of thermal-hydraulic containment codes. The two major phenomena involved in spray behaviour under such accidental conditions are the thermodynamical effect of a spray (steam condensation on droplets, leading to a local increase of hydrogen concentration) and the dynamical effect (mixing of gases, leading to a decrease of hydrogen concentration). The competition of these two coupled phenomena is an important issue for nuclear safety and can be assessed using CFD codes.

For nuclear reactor (containment vessel of around 60 000 m^3), simplifications have to be done to simulate a nuclear accident in the containment where gas mixture (steam, hydrogen and air) is mixed by the spray systems. Up to now, no CFD calculations are available in the open literature on spray systems in a real-scale nuclear containment, using detailed spray initial conditions, accurate droplet modelling and droplet-gas momentum interaction. Many choices can be performed to reduce the computational time of such sprays in a very large containement: atomization zone is neglected, some calculations consider only one droplet size and velocity at one single injection point, other calculations consider a dynamical equilibrium between gas and droplet, etc. The objective of this paper is to evaluate the influence of several choices, performed on spray boundary conditions, on some selected ‘output’ parameters that can influence the overall gas mixing in nuclear reactors.

This evaluation is performed on a real-scale PWR spray nozzle (hollow cone) having an outlet diameter of 1 cm and a maximal diameter of its envelope of about 2 m. CFD calculations are performed using the ANSYS/CFX-FLUENT code (lagrangian approach) and the EDF NEPTUNE_CFD code (eulerian approach).

Experimental results are obtained on the CALIST (Characterisation and Application of Large Industrial Spray Transfer, 160 m^3) facility at the IRSN. The droplet measurements are performed using Phase-Doppler Interferometry (PDI, ARTIUM) and fog sprays are used to have an estimation of the characteristics of the gas entrainment. Measurements performed at the very first zone where droplets are formed and spherical, i.e. 20 cm from the nozzle outlet, are used as ‘boundary conditions’ in the CFD calculations. Other measurements are recorded down to around 1 m from the nozzle outlet.

Sensitivity studies are performed using these ‘boundary conditions’ post-processed in different ways: for example, a radial profile of a variable (droplet size or velocity) can be reduced into a single mean value of this variable. Furthermore, ‘boundary conditions’ on droplet characteristics are not sufficient, since the gas is already entrained at this location. Sensitivity calculations to the choice of the gas ‘boundary conditions’ are also performed, as well as sensitivity studies to several parameters like mesh density, number of particles in the lagrangian approach, turbulence model, etc. Example of code-experiment comparisons is presented in Figure 1. Influence of different injection conditions on the entrained gas velocities is presented in Figure 2.

The final paper will present the experiments, the CFD codes, the code-experiment comparison and the sensitivity studies used to evaluate the importance of different parameters regarding to nuclear safety analysis.

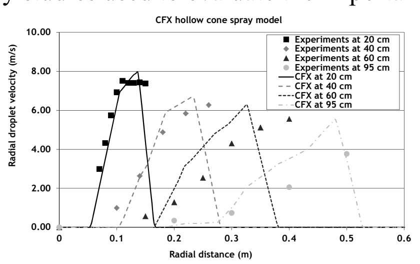


Figure 1: Code-experiment comparison of radial velocity profiles

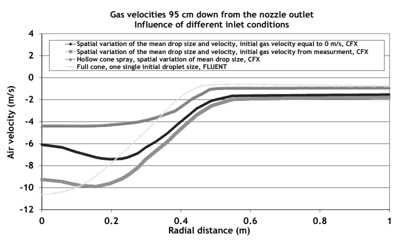


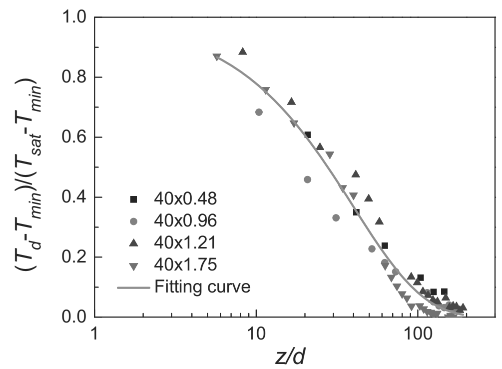
Figure 2: Influence of various initial conditions of the entrained gas velocities

Influence of Nozzle Diameter on Spray Characteristics and Surface Heat Transfer Dynamics in Cryogen Spray Cooling for Dermatologic Laser Surgery

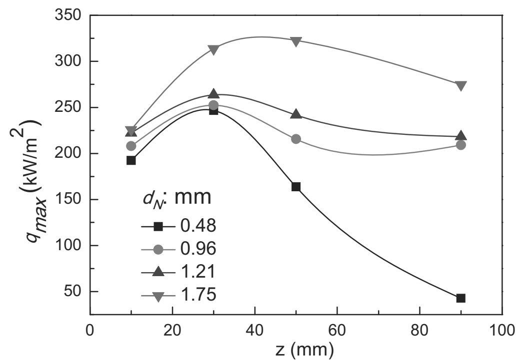
Zhifu Zhou[1], Rui Wang[1], Guoxiang Wang[1, 2*], Bin Chen[1*]
[1] State Key Laboratory of Multiphase Flow, Xi'an Jiaotong University, xi'an, China
[2] Department of Engineering, University of Akron, Akron, USA
chenbin@mail.xjtu.edu.cn and gwang@uakron.edu

Abstract

Pulsed Dye Laser (PDL) at wave length of 595nm or 585nm has been the common choice for the treatment of vascular skin lesions, such as port wine stain (PWS), based on the principle of selective photothermolysis. The objective of laser treatment for PWS is to cause selective thermal damage to subsurface targets (chromophores) without causing damage to the overlying normal epidermis. However, melanin existing in epidermis will greatly absorb laser energy, which not only negatively influences the therapeutic outcome but also causes irreversible thermal damage to the normal epidermis. Cryogen spray cooling (CSC) with cryogen R-134a (-26.1 ^{o}C boiling point at 1 atm) may selectively precool the superficial layers of skin to minimize or eliminate laser-induced irreversible injury to the epidermis. In order to optimize the nozzle design and enhance their cooling efficiency, an experimental investigation was carried out on the effect of nozzle size on the atomization characteristics and heat transfer dynamics during pulsed cryogen spray cooling using eight straight-tube nozzles with different length and diameter. A phase Doppler particle analyzer (PDPA) is used to measure the droplets diameter and velocity, while a micro-flowmeter is used to monitor the volume flow rate of the pulsed spray. It is found that smaller diameter nozzle presents a better atomization capacity than that with larger diameter. The droplets temperature as a function of spray distance is measured by an inserted micro-thermocouple with a bead diameter of 100 µm. It's found that the temperature of droplets produced by small diameter nozzle deccreases much quickly than that by large diameter nozzle. A thin film thermocouple (TFTC, 2µm) is deposited directly onto the epoxy resin substrate or the so-called skin phantom to measure the surface temperature variations induced by the CSC. An analytical expression based on Fourier's law and Duhamel's theorem is used to calculate surface heat flux from the temperature measurements. It's found that the droplets with high velocity and large diameter produce higher heat flux at the cooling surface. Based on the measurements and calculations, the effect of the eight straight-tube nozzles on the heat transfer dynamics of the cooling surface are comparatively studied and their atomization characteristics are compared. Additionally, the criterion to evaluate the cooling efficiency of different nozzles is proposed, and the variation of heat extraction from the cooling surface with different spray distance by different nozzles is given. The results can be used to guide the selection of nozzles during cryogen spray cooling.



Variation of average non-dimensional temperature of droplet with the non-dimensional axial distance along the centerline



Variation of the maximum surface heat flux with the spray distance for different diameter nozzles.

Experimental spray characterisation of air-assisted impinging jets

Madan Mohan Avulapati[*1], Anand T N C[2], Ravikrishna R V[1]
[1]Department of Mechanical Engineering, Indian Institute of Science, India
[2]Department of Mechanical Engineering, Indian Institute of Technology Madras, India
madan.avulapati@gmail.com, anand@iitm.ac.in and ravikris@mecheng.iisc.ernet.in

Abstract

Air-assisted atomization, in which kinetic energy of air is used to aid the liquid atomization, has been used in the past to atomize various liquids. In most studies, either an annular/central liquid sheet is blasted with air jets, or the gas is allowed to mix with the liquid inside the atomizer to form the spray. On the other hand, impinging jet atomizers are used in applications such as bipropellant rocket engines and chemical processes which require rapid mixing between two fluids. In the present study, an arrangement consisting of impinging jets along with air-assist has been studied to take advantage of both methods.

The atomizer employed in the current study consists of two liquid nozzles with 0.76 mm diameter orifices, and a gas nozzle with a 1.1 mm diameter orifice. The liquid jets impinge and form a liquid sheet in the plane perpendicular to that containing the axes of the injectors, and the gas supplied through the gas orifice is used to assist the break-up of this sheet. The gas injector is placed above the impinging point, making equal angles with the two liquid jets. In the present study, spray structure measurements are taken at three different liquid jet impinging angles viz. 60°, 90° and 120°, and at different gas flow rates at each angle. The liquid jet velocity has been varied between 1.8 m/s and 7.3 m/s at each impingement angle and gas flow rate. Spray images are taken using backlit imaging technique. A pulsed Nd:YAG laser along with a diffuser is used to back-illuminate the spray and an Imager Pro X 4M CCD camera with 2048 X 2048 pixel CCD resolution was used to capture images of the spray. The images show that introducing a small amount of gas assists in breaking up the sheet formed by the liquid jets. Increasing the gas flow rate resulted in complete breakup of the sheet as shown in Fig. 1a. The spread of the spray is similar in both the plane of sheet and plane of jets at low liquid jet velocity. However, at higher liquid jet velocity, the spray spreads more in the plane of the sheet compared to the plane of the jets. The breakup length of the sheet is computed from the images as the distance from the impinging point to the point where sheet detaches completely. The results indicate a reduction in breakup length with increase in gas flow rate. Particle/droplet imaging analysis (PDIA) technique is used to measure the spray Sauter mean diameter (SMD). Droplet size measurements were performed in an area 5 mm X 5 mm, at an axial distance 75 mm downstream of the impingement point. These measurements were performed at four different radial locations, with impinging angle of 90° and gas flow rate of 10 LPM. Experimental observations showed that introducing gas flow results in a significant reduction in droplet diameter, as expected. The effect of gas is predominant at low liquid jet velocities and is reduced for increased liquid jet velocities. Variation in the SMD is observed with radial location as shown in Fig. 1b, and this variation in SMD with radial distance is observed to be more at higher liquid jet velocities. Air-assisted impinging jets resulted in a spray with larger droplets at the center and smaller droplets away from the axis of the spray.

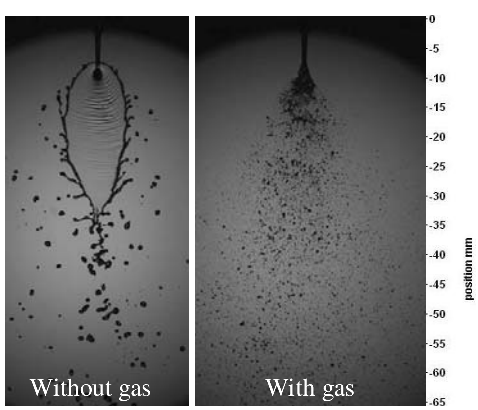


Figure 1 a) Effect of gas on spray

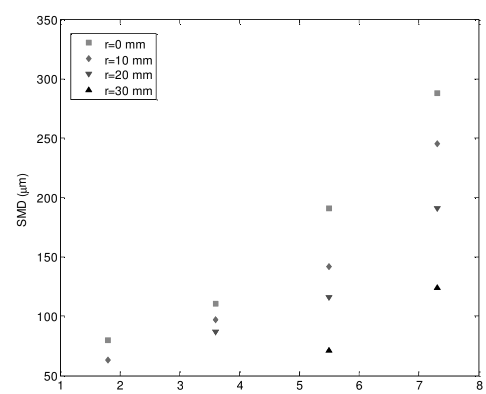


Figure 1 b) Variation of SMD with radial location

* Corresponding author: madan.avulapati@gmail.com

The Observation of the Atomization and Mixing of Doublet-Jets Impinging Sprays at Elevated Ambient Pressures

Tony Yuan*, Cetera Chen, Yu-Da Chen, and Berlin Huang
Department of Aeronautics and Astronautics Engineering, National Cheng Kung University, Tainan, Taiwan, R.O.C.
yahn@mail.ncku.edu.tw

Abstract

The technique of liquid-liquid impingement is commonly adopted in the injector design of the liquid rocket propulsion system, because of its simultaneously atomizing and mixing characteristics. The structure of liquid-liquid impinging spray is intricately affected by jet's velocity, momentum flux, physical properties of the liquids, the size of the orifices, and the impinging angle. It has been shown that the momentum flux as well as the velocity of the liquid jets and the size of the orifices affected the droplet size and mass distributions of impinging sprays, and ambient pressure affects the aerodynamic instability of the jets so as to the primary and secondary break up of the liquid. Moreover, the kinetic energy density ($\rho v^2 d$) of the jets influent the mutual penetration of the droplets from different jets is the key to the mixing behavior of impinging sprays.

This study researched the atomization and mixing behaviors in sprays of like-doublet water impinging jets at different ambient pressures (1.0 bar, 6.8 bar and 10.0 bar) and momentum fluxes of the jets. The orifice (0.4 mm) was arranged to have jet's impinging angle at +30°/-30°. The orifice size and the momentum flux ratio of the two jets were kept the same in each experiment. PLIF (Planar Laser Induced Fluorescence) technique was adopted to observe the 2-dimensional mass distribution of liquid spray from either or both jets at 10mm downstream from the impinging point. And Malvern Spraytec particle size analyzer were used to measure the SMD of the water sprays. The parameters of Patternation Index ($P.I.$), penetration percentage ($P.P.$), and mixing efficiency (E_m) were used to analyze the degrees of atomization and mixing. All the observed impinging sprays were at fully-developed conditions. For 0.4 mm impinging jets, the sprays reach their fully-developed condition near the momentum flux of 0.76×10^5 kg/m×s^2. At higher momentum fluxes, the impinging sprays have more effective break up and more uniform distribution. From the analysis, $P.P.$ variations with the momentum flux can be distinguished by three stages. $P.P.$ increases with increasing momentum flux right after the fully-developed condition, $P.P.$ rapidly decreases after reaching a maximum value, and $P.P.$ slowly increases after reaching a local minimum value. When $P.P.$ decreases to the local minimum (close to 50%) at a characteristic momentum flux (5.7×10^5 kg/m×s^2), the impinging spray have the best mixing efficiency.'

In the higher pressure environment, the impinging jets are less stable aerodynamically, and tend to rippled before impinging and break up more uniformly after impingement. The $P.P.$ variations with the momentum flux at various pressures are qualitatively similar as described above. However, at higher ambient pressures, the maximum $P.P.$ and the best mixing efficiency occur at relatively lower momentum fluxes. And for the higher aerodynamic instability, the decrease of the mean droplet size of the sprays near the impinging point was also observed at the higher ambient pressure.

In this study, the atomization and mixing of the impinging jets with 0.4 mm orifice diameter were also compared with that of the impinging jets with 0.3 mm orifice. At the same momentum flux, a smaller jet has a lower Reynolds number thus to a lower hydrodynamic instability. That is, smaller jets were comparatively more difficult to be uniformly atomized than that of the larger jets. The variation of $P.P.$ with the momentum flux of the 0.3 mm jet is also qualitatively similar to that of the 0.4 mm jet, however, the momentum fluxes to have fully developed, the maximum $.P\ P$, and the best mixing efficiency occur are at relatively higher values.

This study showed that the ambient pressure and orifice size directly influence the disintegration of the impinging jets and the droplet size distributions of the sprays. And the mixing behavior of the impinging jets is closely related to the instability of the liquid jet before impingement.

Shape Deformation and Atomization of Functional Droplets in Contact with a Vibrating Surface

P. Deepu[1], Abhishek Saha[2], Saptarshi Basu[1*], Ranganathan Kumar[2]
[1] Indian Institute of Science, India
[2] University of Central Florida, USA
pdeepu@mecheng.iisc.ernet.in, abhishek.saha@ucf.edu, sbasu@mecheng.iisc.ernet.in and ranganathan.kumar@ucf.edu

Abstract

In this work we experimentally study the shape deformation, bulk oscillation, spread rate and dynamic contact angle variation of functional droplets with varying surface tension, viscosity and density. Seven major types of droplets were studied, namely, (a) water; (b) colloidal suspension of SiO2 nanoparticle in water at three different volume fractions, namely, 0.27, 0.1 and 0.05; (c) glycerol and (d) aqueous solution of glycerol at volume concentrations of 0.2 and 0.9.

The experimental setup consists of a piezoelectric transducer vibrating at 100 kHz and a high speed camera. Due to the presence of the reflector opposite this vibrating surface a standing wave was created in the vertical direction. The droplet is first generated on the transducer surface such that it is located asymmetrically in the standing wave. Subsequently, the transducer is made to oscillate at same amplitude for each droplet resulting in a shape oscillation in the droplets. A high speed camera with a resolution of (896 x 392) was used to image the deformation at 20000 fps.

Once the transducer starts oscillating the droplets undergo different stages of deformation. Droplets of low viscosity are found to eventually atomize, whereas high viscous fluids resist atomization thanks to viscous damping. Different regimes of the process are identified as follows: (1) Small-scale Oscillation Regime; (2) Droplet Spreading Regime; (3) Two-lobe Regime and (4) Film-spreading/Atomization Regime. The physics of each of these regimes is explained. The droplet spreading regime is further analyzed in depth. A theoretical model is formalized to simulate the droplet deformation in this regime, elucidating the major parameters controlling the process. The spread-rate of droplets is found to be a characteristic of the fluid viscosity. The non-Newtonian behavior due to presence of nanoparticles in the case of nanofluid can affect the droplet spreading characteristics substantially. Based on the model and experimental data, correlations are proposed to relate the fundamental non-dimensional numbers that are involved.

Detailed Numerical Analysis of X-ray Phase Contrast Imaging in Sprays

Mark A. Linne*
Combustion Engine Research Center, Department of Applied Mechanics
Chalmers University, Gothenburg, Sweden
mark.linne@chalmers.se

Abstract

Recent studies of spray-related flowfields using synchrotron-based x-ray phase contrast imaging (PCI) have produced results that are sometimes straightforward to interpret in terms of the fluid structure, but in other cases the images do not reflect generally accepted physics of fluid motion. It has been unclear why some images have the appearance of a normal fluid stream while others depart significantly from expectation. The detailed numerical modeling presented in this paper is meant to explain the images and resolve common questions about the technique. The simulations show that collimated x-ray beams will always contain signatures from every possible encounter, from the input plane to the exit plane, and these signatures generate overlapping phase contrast patterns that can prove at times impossible to interpret. Clouds of moderate- to large-size drops produce a complex, mottled x-ray phase contrast image that cannot be interpreted. Small drops generate something akin to one gray pixel image each, and their size is close to the resolution limit of the instrument, so the diffraction pattern is broadened by the instrument response into something more like a small diffuse gray blob. Dense clouds of small drops produce a composite image that is a fairly uniform gray mass that cannot be interpreted. Moreover, we show that it is not possible to image intact liquid structures behind clouds of drops. Whenever a meaningful number of drops is present, therefore, x-ray phase contrast images are dominated by unavoidable artifacts of the technique.

* Corresponding author: mark.linne@chalmers.se

Ballistic Imaging of Sprays at Diesel Relevant Conditions

S. P. Duran, J. M. Porter, T.E. Parker*
Colorado School of Mines, Golden, CO USA
sduran@mines.edu, jporter@mines.edu and tparker@mines.edu

Abstract

Diesel engines are an important aspect of our transportation infrastructure, whose performance is greatly affected by the characteristics of the spray from their injectors. Characterization of the injector's spray is therefore tantamount to clean-running efficient diesel motors. In this study, a ~15 ps pulsed Leopard D-10 laser is used along with an optical Kerr Cell to realize ballistic images of high-delivery-pressure dodecane sprays injected into a quiescent air environment at elevated temperature and pressure. Using CS_2 as the Kerr switching media and by optimizing the overlap of the gate and imaging beams, an effective high speed shuttering effect is obtained. This effort resulted in the first usage of a ballistic imaging technique to successfully capture the spray behavior of dodecane injected at pressures of 100 MPa or greater, via a single 160 micron hole injector, into pre-combustion environmental conditions typical of diesel engines. Control of the imaging beam arrival time after the start of injection allowed for the capture of images from separate experiments that indicate the spray's development over time. The resulting images demonstrate significant differences in spray behavior over a range of chamber temperatures and pressures. Specifically, at low pressure and temperature, some smooth wave structure is apparent, but at high chamber pressure, sprays show significant signs of violent mass shedding from the spray periphery.

Representation of Laser Diffraction Diameter Distribution with a 3-Parameter Generalized Gamma Function

Christophe Dumouchel*, Jean-Bernard Blaisot, Viet Dung Ngo
CNRS UMR 6614 – CORIA Université et INSA de Rouen
Avenue de l'Université, BP 12, 76801 Saint Etienne du Rouvray, France

Abstract

The ability of a mathematical function to represent liquid spray drop-size distributions is usually quantified by exploring its capability of representing experimental data. Experimental diagnostics are not sensible to the same spray characteristics. We therefore have reasons to believe that the more appropriate mathematical function to represent experimental data depends on the diagnostic. Thus, the question of the appropriate mathematical distribution to represent measured data should be addressed for a given diagnostic. This work adopts this strategy and concentrates on the relevance of representing Laser Diffraction Technique (LDT) volume-based drop-diameter distributions with 3-parameter Generalized-Gamma functions.

The first part of the paper addresses the question of the number of adjustable parameters. The 3-parameter Generalized-Gamma function is mathematically identical to the Nukiyama-Tanasawa distribution whose ability to reproduce liquid spray drop-size distribution has been reported by many previous investigation and has not to be proved. However this distribution might suffer from parameter instability that manifests when different parameter triplets provide almost identical volume-based drop-diameter distributions. This suggests that two parameters might be enough when fitting the general shape of this distribution. However, it is demonstrated here that these distributions have a very different characteristic feature in the small drop population. Being analogous to a fractal dimension, this characteristic feature is related to the physics of fragmentation in the literature, and is believed to be of paramount importance. Therefore, the three parameters allow having independency between the distribution shape and its fractal characteristic. Furthermore, it is demonstrated that the three parameters are required to reproduce mean-diameter series as those reported by previous LDT measurements.

These results demonstrate that three parameters allow reproducing LDT drop-size distributions and going above the simple exercise of curve fitting. A protocol to determine these parameters is proposed. One of the parameters is determined from the actual spray small-drop population. However, the small-drop population reported by the LDT distribution might be affected by the presence of non-spherical droplets. Thus, the information related to the actual spray small-drop population is obtained with another diagnostic.

The second part is an experimental investigation in which water sprays are analyzed by two different diagnostics, i.e., LDT and Image Analyzing Technique (IAT). These diagnostics perform a different spray sampling. LDT performs a line-of-sight spray sampling in the direction of the incident light propagation whereas the spatial integration of IAT in this direction is limited to the depth of field of the image which is less than the spray thickness and dependent on the size of the drops. To overcome this problem, IAT is improved in order to perform tomographic spray sampling. The information reported by each spray slice is cumulated and allows characterizing the whole spray as does the LDT. This tomographic exploration reports interesting information on the spray structures. Some examples are presented in this paper.

The surface-based diameter distributions reported by the two diagnostics are not identical and in particular, in the small diameter range. This disagreement is believed to be due to the contribution of non-spherical drops. The small drop population reported by IAT is constituted of small and spherical elements. It is seen as the actual spray small-drop population and used to determine one parameter. The two other parameters are determined by fitting the LDT distribution. The results show excellent fits and no instability of the mathematical parameters is reported: they show a clear relationship with the injection pressure that was the experimental parameter of the present work. Furthermore, besides the shape of the distribution, good reproductions of the fractal characteristic and of the mean-diameter series are obtained. Finally, one of the adjustable parameter reported a clear correlation with the average lack of sphericity of the droplets that was measured from the images. This demonstrates that the LDT distributions are well influenced by the droplet shapes and include average information on this very point.

* Corresponding author ; Christophe.Dumouchel@coria.fr, 00 33 2 95 36 23

Preliminary shock-tunnel experiments on liquid fragmentation and atomization in hypersonic flows

G. Schlöffel*, J. Srulijes, H. Albers, M. Bastide, C. Rey
French-German Research Institute of Saint-Louis (ISL), France
gunther.schloeffel@isl.eu and julio.srulijes@isl.eu

Abstract

The present work describes the objectives, the set-up and the preliminary results of an experimental study on the fundamental mechanisms of fragmentation and atomization of a bulk of liquid in hypersonic gas flows. Additionally, we introduce a former series of experiments and calculations also carried out at ISL.

The current experiments are conducted by means of a horizontal shock tube which is operated as hypersonic wind tunnel (shock tunnel, see Fig. 1). The experimental set-up used is capable of reproducing conditions of real atmospheric flights from Mach 3 at ground-level conditions up to Mach 14 at a flight altitude of 70 km.

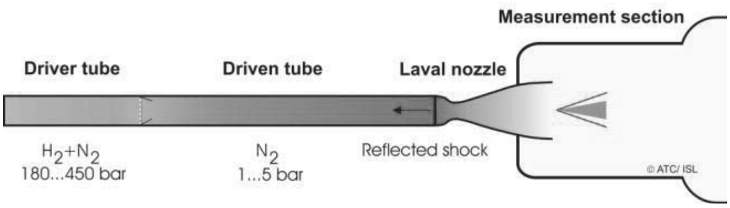


Figure 1. Principle sketch of the shock tunnel with a contoured nozzle

This work focuses on shock tunnel experiments on the fragmentation and atomization of a bulk of liquid suddenly exposed to hypersonic atmospheric flow. The liquid substance to be studied is filled into a latex balloon which is placed in front of the shock tunnel nozzle. A needle driven by a magnetic mechanism pierces the balloon shortly before the nozzle flow sets-in, so that the bulk of liquid is introduced into the flow in an almost non-intrusive way (see Fig. 2).

A first series of experiments at Mach 4.5 has been conducted with the help of a high-speed camera to observe the fragmentation of a bulk of 5 ml of water, ethanol and hexane at flight altitudes of 10, 20, 30 and 40 km (see Fig. 3). The analysis of the atomization and the evolution of the drop sizes require more sophisticated optical measurement techniques. Therefore, in a series of experiments already started a special Particle Image Velocimetry (PIV) technique is used to determine the velocity of the drops. A first result is presented in this paper. Beside this, a LIF-technique is being developed and adapted to observe the evolution of drop size and liquid evaporation.

Figure 2. Piercing mechanism

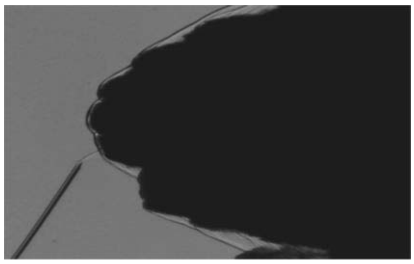


Figure 3. High-speed camera shadowgram

* Corresponding author: gunther.schloeffel@isl.eu

Droplet Breakup Modelling in Spraying

I. Sher[1*], E. Sher[2,3]

[1]School of Engineering, Cranfield University, Cranfield, Bedfordshire, United Kingdom
[2]Department of Mechanical Engineering, Ben-Gurion University, Beer-Sheva, Israel
[3]Visiting Professor, Faculty of Aerospace Engineering, Technion - Israel Institute of Technology, Haifa, Israel

Abstract

A study of the problem of a liquid droplet in high speed trajectory through ambient gas is presented. The behaviour of such a droplet is of importance to the overall spray characteristics, through the secondary breakup mechanism.

The droplet breakup mechanism is classified in the literature into several regimes according to the relative velocity between the droplet and the surrounding. Krzeczkowski [1] classified the droplet deformation into four different types using the Weber and Ohnesorge numbers, while Pilch and Erdman [2] modified the classification into five regimes, and defined a critical Weber number below which droplet breakup does not occur. In the present study, however, we consider the energetic limiting conditions of breakup as related to possible breakup products, thus defining the borders between droplet deformation without breakup and different modes of breakup.

We consider the following fundamental scenario: Under certain circumstances, a single droplet that travels through ambient can undergo a spontaneous breakup process to yield travelling product droplets. Due to the ambient hydrodynamic drag forces, the travelling droplet is deformed, while the deformation extent and the deformation rate are strongly coupled to the magnitude of the drag force. The drag force itself depends mainly on the trajectory velocity and to a lower extent [3] on the droplet shape (deformation extent). Thus, when a droplet is shot into ambient, deformation develops, which may lead to either droplet breakup or alternatively to droplet further deceleration and reformation of its shape. Breakup can occur into different number of products, depending on trajectory parameters, e.g. length and time. The fate of the droplet depends on the droplet velocity, on the droplet thermo-physical properties, and on the ambient properties, through the time scales of the droplet's trajectory deceleration and deformation rate.

In the theoretical analysis of the problem, we examine an overall energy balance that includes surface and dissipation energies. Transformation between a droplet to its breakup products under trajectory conditions that include drag forces, is a spontaneous process that occurs at a characteristic finite time. The transformation involves reversible surface energy transitions, and irreversible viscous dissipation energy. Results propose simple relations for time of breakup, trajectory length, and breakup limits.

Results of breakup limit show good agreement with various available published results. A breakup limit criterion correlates Weber and Ohnesorge numbers.

References

[1] Krzeczkowski, S.A., "Measurement of Liquid Droplet Disintegration Mechanisms", Int. J. Multi-phase Flow, vol. 6, pp. 227–239, 1980.

[2]Pilch, M. and Erdman, C.A., "Use of Breakup Time Data and Velocity History Data to Predict the Maximum Size of Stable Fragments for Acceleration-Induced Breakup of a Liquid Drop", Int. J. Multiphase Flow, vol. 13, no. 6, pp. 741–757, 1987.

[3]Stiesch, G., *Modeling Engine Spray and Combustion Processes (Heat and Mass Transfer)*, Springer Verlag, 2010.

* Corresponding author: i.sher@cranfield.ac.uk

Thermally Induced Breakup of Levitated Droplet

A. Saha[1], S. Basu[2*], R. Kumar[1]
[1]University of Central Florida, Orlando, USA
[2]Indian Institute of Science, Bangalore, USA
abhishek.saha@ucf.edu, sbasu@mecheng.iisc.ernet.in and ranganathan.kumar@ucf.edu

Abstract

In this work we experimentally study the thermal effects that lead to instability and break up in acoustically levitated vaporizing fuel droplets. We report two kinds of instabilities: 1. Short wavelength (Kelvin-Helmholtz) instability for diesel and bio-diesel inducing secondary atomization near the droplet equator and 2.Bulk deformation results in wide fluctuations in droplet aspect ratio analogous to a spring mass system with low damping coefficient that ultimately leads to catastrophic breakup involving bag type and capillary wave induced atomization. Temperature dependent fuel properties and external heating rate are responsible for the inception and relative strength of these instabilities. If breakup occurs, it is always preceded by secondary atomization when droplet Weber number exceeds a critical value. Catastrophic breakup is induced in deformed droplets when the Bernoulli pressure due to the levitator pressure field is significantly high and surface tension and viscous damping is comparatively low so that the highest energy containing frequency of aspect ratio vibration leads to bag type instability.

The experimental setup consists of an ultrasonic levitator (100 kHz, 154dB) and a 30W tunable CO2 laser (wavelength 10.6 µm) to irradiate 500 micron droplets. The heating event was recorded simultaneously by a high speed (Phantom V12) camera whose images were processed to obtain instantaneous diameter, aspect ratio and shape changes, and an infra-red (IR) camera to obtain the surface temperature of the droplet. Both cameras used microscopic lenses to increase the spatial resolution. The resolution was nearly 2 µm in every pixel.

This work reports and explains that secondary atomization in levitated fuel droplets is primarily due to thermal effects. We showed that the properties such as vapor pressure, latent heat and specific heat govern the vaporization rate and temperature history. This in turn affects the surface tension gradient and gas phase density, ultimately dictating the onset of KH instability. We laid down a criterion for the inception of this instability in terms of Weber number. Low vapor pressure and latent heat leading to high surface temperatures achieved in diesel, bio-diesel and kerosene favor small scale atomization through KH instability. The temperatures for fuels having high vapor pressure and high latent heat will remain rather low and they tend to be more stable like ethanol due to lower surface tension gradient.

A second type of instability occurs in kerosene droplet due to a decrease in surface tension and viscosity with temperature. The change in surface tension causes the droplet to flatten due to increased Bernoulli pressure at the poles resulting in an increase in aspect ratio. The imbalance in pressure force and surface tension force near the equator creates shape oscillation. If the viscous damping of this oscillation is not strong enough, the droplet goes through unbounded stretching morphing into a disk-like shape, followed by bag type catastrophic breakup.

Secondary Atomization of Newtonian Liquids in the Bag Breakup Regime: Comparison of Model Predictions to Experimental Data

V. Kulkarni, D.R. Guildenbecher[†], P.E. Sojka[*]

Maurice J. Zucrow Laboratories,
School of Mechanical Engineering,
Purdue University, West Lafayette, IN, U.S.A.

vkulkarn@purdue.edu , drguild@sandia.gov, sojka@ecn.purdue.edu

Abstract

Secondary atomization refers to the fragmentation of liquid drops due to aerodynamic forces exerted by a surrounding gas-phase. It is important in the context of combustion applications as improved understanding of the underlying physical phenomena can lead to better engine efficiencies. In the current work we examine one aspect of the secondary atomization process i.e., the deformation of an inviscid drop when exposed suddenly to a flowing air stream. As outlined by Guildenbecher et al. (2009) depending on the air velocity and the fluid properties we observe a range of different deformations and fragmentation behavior. The bag breakup mode is one such deformation/fragmentation process characterized by flattening of the spherical drop leading to the formation of a bag bounded by a thick rim which eventually collapses to form smaller droplets. The present work seeks to advance the current understanding by comparing experimental evidence with theoretical predictions. The experimental setup comprises of a high-speed digital imaging system along with backlight illumination. This is used to capture details of bag/rim breakup and the ensuing fragmentation. Bag initiation time is computed from the set of videos obtained for various *We* corresponding to the liquid drop. Drop size measurements are obtained from PDA to supplement these results. The theoretical analysis is essentially divided into two parts. The first part deals with computing bag breakup times, and the corresponding topological evolution of the drop is estimated along the lines of scaling arguments available in the literature (Villermaux, 2009). This is in slight contrast to the Rayleigh-Taylor type mechanism which is often posited as the mechanism of bag deformation (Zhao, 2011). Theoretical predictions show no dependence of the maximum extent of the bag on *We* which is consistent with experimental findings. The model correctly predictes We_{cr}, which marks the transition from the vibrational to the bag process. In the second part, a linear stability model (Gast, 1991) is used to compute the drop sizes obtained after capillary breakup of the ring. It must, however, be noted that the model for the rim breakup requires information of the characteristic inner and outer toroidal radii which are obtained from the experimental runs and cannot be predicted accurately from just the initial drop diameter before it enters the air stream. Also, before the rim shatters it is corrugated and not perfectly toroidal (Zhao, 2011) which is an important consideration in the analysis and is taken into account. Typically, the bag fragments comprise of smaller drops, large in number and the rim consists of large drops smaller in number and form after the bag has completed ruptured (Dai, 2001). This expected bi-modal distribution is not observed in the experimental runs because the probability of larger drops was insignificant for these tests. Thus, the preliminary analysis and experiments have helped extend current understanding and confirm existing notions about the bag breakup in secondary atomization.

Keywords: Bag Breakup, Secondary Atomization, Sprays

* Corresponding author: sojka@ecn.purdue.edu
† Presently at Sandia National Laboratories, Albuquerque, NM, U.S.A.

LES and Experimental investigation of Diesel sprays

Chawki Habchi* and Gilles Bruneaux
IFPEN, 1&4, avenue Bois-préau, 92852 Rueil-Malmaison, France
Chawki.Habchi@ifpen.fr and Gilles.Bruneaux@ifpen.fr

Abstract

In this paper, a Diesel spray has been investigated experimentally and numerically in order to improve the understanding of transient processes of short injection typically used in multi-injection strategies of internal combustion engine. The spray was observed experimentally in a high pressure high temperature cell that reproduces the thermodynamic conditions which exist in the combustion chamber of a Diesel engine during injection using a pre-combustion technique. A single-hole injector was mounted within the top face of the cell and the spray was injected at the following operating conditions: P_{inj} = 120 MPa, T_{gas} = 900 K and P_{gas} = 6.7 MPa. Planar Laser-induced Fluorescence (PLIF) was used to obtain two dimensional fields of fuel mass concentration. A normalization method based on the determination of the total injected mass was used to derive quantitative information from the fluorescence imaging. Also, 50 single-shot images of the vapor jet were acquired for each timing in order to allow statistical analysis.

Large Eddy Simulation (LES) numerical approach has been used in order to investigate single-shot transient processes and averaged results. Different LES simulations have been carried out using a Lagrangian approach including recently developed models in the AVBP code, for instance, the DVI model for initial formation of dense sprays and the SAB model for the droplets breakup. The injected fuel was a mixture of 70%vol n-decane and 30% vol 1-methylnaphtalène representative of a standard diesel. Five test cases were defined and are presented in Table 1. Since the injection boundary conditions are not fully known, the calculation methodology consisted in adjusting the numerical jet using the experimental results. Case 1 is the result of this adjustment and will be considered as the reference case, subsequently referred to as T9P12. Particularly, an experimental flow rate and α_l = 0.85 were used. The case 2 uses a modified flat flow rate during the full opening period. Then, the case 3 uses a value of α_l equal to 0.80 to study the effects of cavitation in the orifice on the jet. Finally, the case 4 employs the SGS dynamic Smagorinsky model instead of the standard Smagorinski model used in the first 3 cases. For each case, a series of 15 or 30 LES was calculated by setting different seeds for random sampling of the injected blobs.

The LES computations have been carried out with a well refined mesh in the injection zone. The characteristic size of the tetrahedral cells is of the order of 80 microns over a distance of about 100 diameters from the nozzle exit. This mesh correctly resolves the Gaussian profiles of liquid volume fraction and velocity specified by the DVI model. The full paper will includes the discussion of all the cases presented in Table 1.In particular, it has been shown that more than 30 LES simulations are needed in order to obtain converged numerically results. In this case, a satisfactory quantitative agreement has been obtained between the numerical results and the experiments, in terms of mass density, both in the averaged images and for the axial and radial profiles (see Figure 1 and Figure 2).

Table 1 Definition of the LES parameters for the T9P12 case : Pinj = 120 MPa, Tgas = 900 K and Pgas = 6.7 MPa.

Case	Label	Injection rate	Liquid volume fraction, α_l	SGS models	Total number of LES computations
1(15) 1(30)	A85	Experimental	0.85	Smagorinski model	15 30
2(15)	TXE	Experimental, modified	0.85	Smagorinski model	15
3(15)	TXN	Experimental	0.8	Smagorinski model	15
4(30)	DYN	Experimental	0.85	Dynamic Smagorinski model	30

* Corresponding author: Chawki.Habchi@ifpen.fr

Development of Breakup Model for Large Eddy Simulation of Diesel Spray

Koji Kitaguchi, Soichi Hatori, Tsukasa Hori, Jiro Senda[1*]
Mechanical Engineering Department, Doshisha University, Japan
jsenda@mail.doshisha.ac.jp

Abstract

Diesel engines emit soot and NOx. The formation of these emissions is affected by heterogeneous distribution in diesel spray. This heterogeneous distribution is affected by turbulent vortex structure. Thus, the high accuracy prediction for the amount of these emissions needs to estimate finer turbulent vortex structure. Conventional CFD for the engine combustion is performed using Reynolds Averaged Navier Stokes (RANS) approach as turbulent analysis method. However, RANS approach has no enough capability to describe the finer vortex structure. Recently, Large Eddy Simulation (LES) approach has gotten a lot of attention as the alternative way of RANS approach. LES approach is able to describe the finer structure of the turbulent vortex than RANS approach.

LES has been applied for non-evaporative, evaporative diesel spray in earlier studies. It is found that grid length sensitivity is quite high in LES so that finer grid should be set in order to reproduce finer diesel spray structure. In addition, central difference method and forth order Runge-Kutta method for the convection term of momentum equation are adopted in order to reduce the numerical viscosity. As a result, good agreements of the spray shape and spray tip penetration are obtained in LES analyses. However, spray droplet diameter of LES analyses underestimated in the comparison with the experimental results. This is because Kelvin-Helmholtz and Rayleigh-Taylor (KHRT) model is used for breakup model. It is reported that KHRT model underestimate Sauter mean diameter. KH model and RT model are modeled for high Weber number condition. Even if the fuel is injected with high pressure, the downstream region of spray is corresponding to relatively low Weber number condition due to the momentum exchange with ambient gas. To overcome this problem, we develop KH-MTAB model. This is hybrid breakup model where KH model and Modified Taylor Analogy Breakup (MTAB) model are used for primary and secondary breakup respectively. MTAB model is more suitable for description of secondary breakup, compared to RT model. The LES results with KH-MTAB model show that spray tip penetration, spray volume and sauter mean diameter of LES analyses are good agreement with experimental results at relatively low density condition. However, the tendencies of droplet size distribution are not agreement with experimental results due to the problem with hybrid method of KH model and MTAB model.

This study is focused on the investigation of breakup model which improves coupling of KH model and MTAB model. LES analyses of non-evaporating diesel spray are performed using modified KH -MTAB model at different ambient density and the spray injection pressure. Then, LES analyses were compared with experimental results in terms of spray tip penetration, spray volume, sauter mean diameter, droplet size distribution and spray shape.

*Jiro Senda jsenda@mail.doshisha.ac.jp

Simulating Nozzle Flow and Sprays Using An Eulerian Two-Phase Flow Model with a Realistic Equation of State

Y. Wang, R. D. Reitz*
Engine Research Center, University of Wisconsin-Madison, U.S.A.
yuewang05@gmail.com and reitz@engr.wisc.edu

Abstract

The internal flow in a fuel injector nozzle and the external spray atomization are closely related. As a result, CFD modeling of sprays must properly take into account the influence of the nozzle flow. Before its complete atomization into droplets, the fuel is in the form of continuous liquid column in the near-nozzle region. An Eulerian CFD approach assumes a continuum two-phase fluid both inside and outside of the nozzle, and hence provides a natural coupling between the two flows. Most previous studies using the Eulerian approach assume incompressibility. However, considering the fact that the sound speed is greatly reduced in the two-phase regime, this assumption limits the model's application to high-speed fuel injection cases.

The current study presents an equilibrium Eulerian approach for compressible flows. For single-component flows, the thermodynamic states and phase transition are modeled with the Stiffened Gas Equations [1, 2] implemented in the KIVA-3V code. The model is then extended to two-component flows under the assumption of ideal mixing, with one component being inert air. The methodology of the model and its validation against classical two-phase flow problems are presented.

Comparison of Various Models for Transient Nozzle Flow Simulations Including Time-Resolved Needle Lift

S. Jollet*, T. Willeke, F. Dinkelacker
Institute of Technical Combustion (ITV)
Leibniz University of Hanover
Welfengarten 1a, 30167 Hannover, Germany

Injection systems of modern diesel engines are one of the key components to increase the fuel efficiency and to lower pollutant emissions. Therefore, a detailed understanding of the spray generated by the injector nozzle is crucial to optimize the process of the mixture formation, ignition, combustion, and emission. In diesel injection systems the spray formation is significantly influenced by the internal flow of the injector and often influenced by cavitation [1]. Increasingly important is the transient flow behavior during the needle lift. The aim of this study is the time dependent numerical simulation of the internal flow processes including the multiphase processes with special emphasis to the time dependent variation of the needle. The injector consists of a needle with an adapted needle cone geometry and a nozzle body, which is a 8-hole blind hole nozzle. The holes have a diameter of $d = 247\ \mu m$ with cylindrical geometry and sharp inlet. Experimental test data from a cold injection chamber provide boundary conditions and serve as a plausibility check. The injector needle movement is realized by a dynamic mesh. Several models have been evaluated. These were the combinations of two multiphase models (Eulerian and Mixture model) [2], two cavitation models (Schnerr-und-Sauer, Zwart-Gerber-Belamri) [3] [4]. A suitable combination is found from subsequent studies. Also the influence of vapour pressure was investigated. First results show strongly asymmetric two phase regions inside the nozzle for the transient initial and final part of the injection.

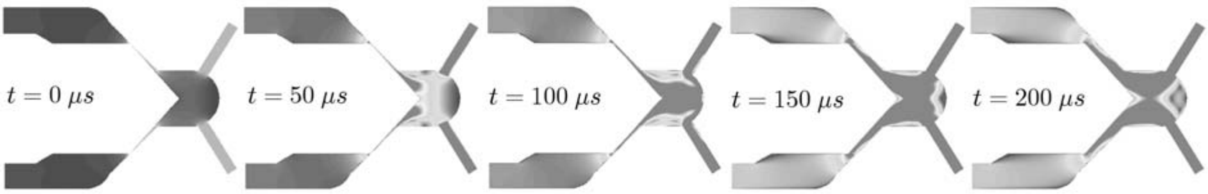


Figure 1. Time-dependent velocity inside the injector during the opening of the needle ($t = 0;\ 50;\ 100;\ 150;\ 200\ \mu s$)

During the opening process (Figure 1) of the needle the cavitation area increases, while in the steady open state it reduces. The influence of details of geometry between needle and needle seat is clearly visible, as here the momentum wave of the fluid is starting. It is found that the multiphase models have a bigger influence on the nozzle flow compared to the cavitation and the vapour phase models. In accordance with literature, a cavitation area was observed for all of the mentioned models at the flow-induced orifice of nozzle hole inlet. In the nozzle appears a cavitation zone at the opposite side of the inflow.

As a conclusion, the multiphase models have a bigger influence on the nozzle flow compared to the cavitation and the vapour phase models. In accordance with literature, a cavitation area was observed for all of the mentioned models at the flow-induced orifice of nozzle hole inlet.

For further work, the comparison of the macroscopic spray simulation with measured data obtained from an injection chamber is planned.

[1] Walther, J. *Quantitative Untersuchung der Innenströmung in kavitierenden Dieseleinspritzdüsen*, Dissertation, TU Darmstadt, Deutschland(2002).
[2] Brennen, C.E. *Fundamentals of Multiphase Flow*, Cambridge University Press, (2009).
[3] Schnerr, G.H., Sauer, J. *Physical and Numerical Modeling of Unsteady Cavitation Dynamics*, In Fourth International Conference on Multiphase Flow, New Orleans, USA, 2001.
[4] Zwart, P.J., Gerber, A.G., Belamri, T. *A Two-Phase Flow Model for Predicting Cavitation Dynamics*, In Fifth International Conference on Multiphase Flow, Yokohama, Japan, 2004.

*Corresponding author: jollet@itv.uni-hannover.de

Spatial Analysis of Fuel Density from Automotive Transient Sprays by Polycapillary X-Ray Imaging

L. Allocca[(1)*], S. Alfuso[(1)], L. Marchitto[(1)], D. Hampai[(2)], and S.B. Dabagov[(2)(3)]
[(1)] CNR Istituto Motori, Napoli, Italy
[(2)] INFN Laboratori Nazionali Frascati, Frascati, Roma, Italy
[(3)] RAS P.N. Lebedev Physical Institute & NRNU MEPhI, Moscow, Russia

l.allocca@im.cnr.it s.alfuso@im.cnr.it l.marchitto@im.cnr.it dariush.hampai@lnf.infn.it sultan.dabagov@lnf.infn.it

Abstract

Attempts to study fuel density distributions inside transient high-density fuel sprays by X-ray based techniques have been carried out in several laboratories around the world. Synchrotron radiation as X-ray sources are mainly adopted due to their high intensity and pulsed nature. On contrary, this sources have the intrinsic limitations of high costs, beamlines with dedicated instrumentation, poor duty cycles.

In this paper laboratory desk-top X-ray techniques based on polycapillary optics has been used for investigating the structure of a gasoline pulsed spray for automotive applications flowing from a GDI injector. Polycapillary optical elements enable shaping divergent X-ray beams as well as getting high contrast image of the samples due to the suppression of radiation multiple scattering. A Cu Kα X-ray source in combination with a polycapillary halflens (semilens) has been used while the extinct radiation by the sample has been collected on a CCD detector. Two injectors, a single-hole hollow-cone and a six-hole, worked at 8.0 MPa injection pressure and sprayed gasoline at atmospheric backpressure in a Plexiglas chamber. This has been vented to prevent fuel fog accumulation and sheltered by Kapton foils 25 μm thick on the optical beam line to avoid droplet/vapor leakage. The injector-pressure pump coupling has been realized through a complex fuel reservoir composed of two parts: fixed and rotating. The last one permitted the rotation of the injector body around its axis by mean of a stepping motor so enabling to irradiate the spray under different angles. Off-line tomography reconstruction has been made with the images with an acquired angular steps of $\Delta\theta = 1°$. Measurements have been carried out close to the nozzle tip and the computed tomography has permitted average reconstruction of the fuel downstream the nozzle tip.

* corresponding author: l.allocca@im.cnr.it

Large Eddy Simulation of GDI Spray Evolution in a Realistic IC-Engine

K. P. Nishad*, A. Sadiki, and J. Janicka
Graduate School of Computational Engineering, Institute of Energy and Power-plant Technology, Technical University Darmstadt, Germany
nishad@ekt.tu-darmstadt.de, sadiki@ekt.tu-darmstadt.de, and janicka@ekt.tu-darmstadt.de

Abstract

In the present study, an effort is made to investigate the real unsteady behavior of evaporating GDI spray in a realistic engine configuration (with moving piston and valves) by applying large eddy simulation (LES) together with a spray module using KIVA-4mpi Code. A comprehensive model set is integrated in a Eulerian-Lagrangian framework allowing to describe the spray evolving from the injector nozzle and propagating within the combustion chamber. It includes sub-models to account for various relevant sub-processes. The atomization is described using combined primary and secondary atomization sub-models. Instead of performing costly level set method or volume of fluid (VOF) technique a linear instability sheet atomization (LISA)-based sub-model is applied for the primary atomization. The secondary atomization is modeled by a Taylor analogy break-Up (TAB) model. A new model is proposed for droplet-droplet interaction that is independent of mesh size and type. It takes into account possible four regimes, such as, bouncing, separation, stretching separation, reflective separation and coalescence of droplets. The droplet evaporation is described by an appropriate evaporation model and the turbulent dispersion by the filtered velocity only. The spray module is coupled to LES of the carrier phase in which a Smagorinsky model is used for the filtered flow field. The sub-grid scale (SGS) scalar flux in scalar transport equations (of mass fraction and temperature) is captured by a simple gradient assumption. The spray wall interaction is taken care by the wall film model that include droplet splash, film spreading due to impingement forces, and motion due to film inertia. An appropriate approach is used to describe the moving boundary conditions for the piston and valves movement.

An optical gasoline engine, designed to support the development and validation activities for the CFD software is used for the simulation of evaporating spray here. The primitive validation is carried out for cylinder pressure curve (see Figure 1) for motored case. Then, a numerical analysis is carried out on interactions of the GDI spray with high speed intake air. The CFD model is able to capture the transient behavior of evolving spray. The result shows influence of the intake charge motion considerably influences spray dynamics and vice versa, thereby air-fuel mixture formation (see Figure 2). The simulation results also show the evidence of the formation of liquid film on the piston wall which is undesirable for the optimum engine performance. The presented result can be used as a basis for further analysis of unsteady effects along with cycle-to-cycle variations in real engine configurations.

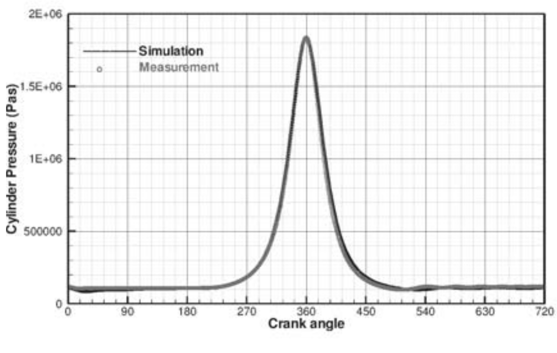
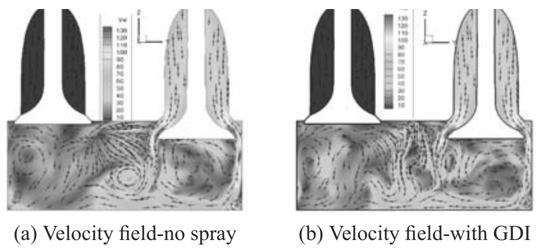
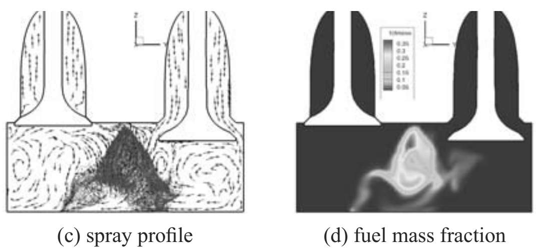


Figure 1. Comparison of pressure curve: computed (black line), and experimental data (gray circle)



(a) Velocity field-no spray (b) Velocity field-with GDI



(c) spray profile (d) fuel mass fraction

Figure 2. Sectional view (x-plane) at 59^o ATDC

*Corresponding author: nishad@ekt.tu-darmstadt.de

Comparing the Cycle-to-Cycle Variations of Pulsing Spray Characteristics by Means of Ensemble Image and Probability Presence Image Analysis Techniques

Jie Zhong[1,2], David L.S. Hung[1,2*], Zhenkan Wang[2], Yuyin Zhang[2], Min Xu[2]

[1]University of Michigan-Shanghai Jiao Tong University Joint Institute,
Shanghai Jiao Tong University, Shanghai, China

[2] National Engineering Laboratory for Automotive Electronic Control Technology,
Institute of Automotive Engineering, Shanghai Jiao Tong University, Shanghai, China

Abstract

This paper presents an investigation to reveal the cycle-to-cycle variations of pulsing spray characteristics of Spark-Ignition Direct-Injection (SIDI) fuel injectors. The objective is to quantify the spray's cycle-to-cycle variation such that further insight of the operating principles in the fuel injection system could be developed to enhance the combustion efficiency and reduce emissions of SIDI engines. The experiments were carried out using a multi-hole SIDI fuel injector under an extended range of test conditions in a spray chamber. Using a strobe light as an illumination source, images of the spray structure were taken by a CCD camera. The analysis approach of the cycle-to-cycle variation was based on constructing two types of images, namely, 1) Ensemble Image, and 2) Probability Presence Image (PPI). The analysis of the ensemble image and PPI reveals that both approaches can be used to extract the variations of the spray structure. While an ensemble image is useful for determining the quantitative variation of the spray characteristics, such as the penetration and spray angle, in terms of average, maximum and minimum limits, a PPI provides a new way to examine the spray variation in terms of a probability defined for the presence of the liquid region. Not only a PPI is able to illustrate the magnitude of cycle-to-cycle variation in penetration and spray angle values quantitatively, it also displays the variations qualitatively in a two-dimensional manner in terms of the liquid presence probability.

* Corresponding author: dhung@sjtu.edu.cn

Numerical Analysis of Multihole Gasoline Direct Injection Sprays

M. M. Khan[1*], J. Helie[*], M. Gorokhovski[†], A. Wood[+], G. Wigley[+,] J. Kashdan[‡], J. P. Dumas[‡], M. Mojtabi[**], P. Guibert[**]
* Continental Automotive SAS, BP 1149 Toulouse Cedex 1, France
† Ecole Centrale de Lyon, Lyon, France, +Loughborough University, Leicestershire, UK
‡IFP Energies Nouvelles, Paris, France, ** Université Pierre et Marie Curie, Paris, France
[1]Muhammad.2.Khan@continental-corporation.com

Abstract

Multi-hole gasoline direct injection (GDI) injector sprays have been studied numerically. This study is actually an extension of the previous work of R. Rotondi [1]. Main part of this work focuses on air entrainment, droplets sizes and velocities in the spray plumes under the hot and cold conditions, which are very important for jet to jet interactions, spray propagation and mixture formation.

A typical advantage of Multi-hole GDI injector is to increase the fuel efficiency by reducing fuel injection timing, penetration and increasing the fuel-air mixture quality for better combustion in the engines. But since there is a presence of multiple spray plumes at the same time therefore it is necessary to avoid jet to jet interactions in order to maintain the intended spray targeting and produce sufficient vapour homogeneity for better evaporation and ignition. Experimentally in multi-hole injectors it is very difficult to look at the air entrainment inside the six holes injectors with narrow cone angle. That is why to go more in detail and understand the physics better, numerical study becomes very important.

The experimental investigations of the PIV air entrainment were performed at IFPEN with operational conditions: injection pressure (P_i) = 200 bar, chamber pressure (P_c) =1.54 bar, fuel Temperature (T_f) = 90°C, chamber temperature (T_c) =33 °C. PDA measurements of droplets sizes had been carried out in the Loughborough University with P_i=100bar, P_c=1 bar and $T_c = T_f$ = 20°. Continental's special XL 3-hole 90° and 6-hole 60° injectors were used for all experiments. The Reynolds Average Navier Stokes (RANS) simulations were performed on OpenFOAM ® version 1.7.1, where the gaseous phase was modeled by the standard K-Epsilon approach and the liquid phase was modeled by the Lagrangian approach. A compressible reacting spray solver equipped with automatic mesh refinement (AMR) was used with a computational domain of size 112 mm x 112 mm x 112mm and minimum cell size of 0.375mm after the AMR. AMR was based on the scalar fields of kinetic energy and vapour mass fraction for non-evaporating and evaporating conditions, respectively. Automatic time step adjustment was also included to keep local courant number to be less than 0.5 with initial time step of 10^{-7} sec.

The droplets' size, liquid penetration and the air entrainment profiles show good agreement with experimental PDA and PIV data respectively. The air entrainment is relatively high in the near nozzle region and at the spray tip leading edge because of the high spray momentum for both the 3 and 6-hole injectors. In 3-hole injector the spray-induced air motion is found to be relatively weak in the zone between the two adjacent plumes and air is entrained in the plumes up to the near nozzle region without any jet to jet interactions. However 6-holes injector shows strong jet to jet interactions with a significant amount of vapour inside the spray cone. The gas inside the upper region of the spray cone is observed to be pushed downwards along the spray propagation which interacts with the oppositely directed air from the bottom half. This interaction produces a radial flow which deflects the spray plumes from their original path and also pushes out the vapor present inside the cone. The deflection angle of the spray cone in the simulations is measured roughly to be between 6° to 9°.

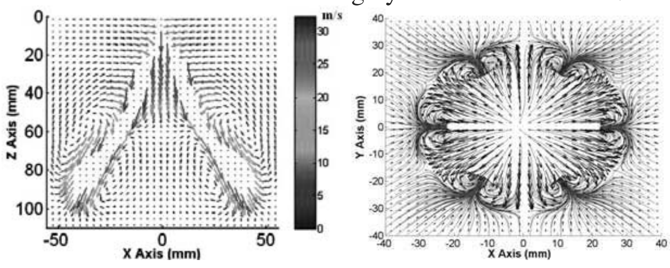


Fig 11: Plots of air & gas entrainment at 3.1ms After the Start of Injection; Axial (left), Radial (right)

Reference

[1] Rotondi R., Hélie J., Leger C., Mojtabi M., Wigley G., *ILASS Europe, Czech Republic, (2010).*

* Corresponding author: Muhammad.2.Khan@continental-corporation.com

Geometric effects in the design of multijet atomizers

M. R. O. Panão*, J. M. D. Delgado, A. L. N. Moreira
Instituto Superior Técnico, Technical University of Lisbon, Portugal
mpanao@dem.ist.utl.pt; joao.delgado@ist.utl.pt and moreira@dem.ist.utl.pt

Abstract

The multijet atomization strategy stands out as competitive in the production of liquid atomizers with relatively simple geometries, producing sprays from the impingement of two or more jets ($N_j \geq 2$), with a relatively low injection pressure and, consequently, low flow rates ($< 1.5l/min$). However, there are two geometric factors affecting the design of these atomizers and, eventually, the atomization process itself, which are scarcely approached in the literature: i) the pre-impingement distance (l_{pi}) of the jets; ii) and the jets' misalignment. This paper analyzes the influence of these parameters on the structure of the liquid sheet formed after the impact of two jets, as well as on drop size. An experimental setup has been built with the possibility of studying multijet sprays up to $N_j = 4$, where each jet has 4 degrees of freedom $(x, y, z, \theta$, the later corresponds to half the impingement angle) and the flow rate is controlled individually. The range of jet Reynolds (Re_j) numbers lie between 1485 and 5729 and the impingement angles from 90° to 120°. The pre-impingement distances vary from 2.5 to 7.5 of the jet diameter (d_j) and the misalignment from 0 to 50% of d_j. The flow is visualized with a high-speed camera and the images are processed in order to quantify the liquid sheet (LS) breakup length and width, as well as drop size. Results evidence that for lower Re_j, when the liquid sheet develops in a closed rim, the pre-impingement distance does not significantly influence the LS structure, or the characteristic size of droplets, which are mainly formed from capillary instabilities detaching from the bounding rim. However, if the Re_j keeps increasing, the LS rim begins opening at the bottom. Li and Ashgriz [1] observed a transition between a closed- and an open-rim LS between Re_j = 2000 and slightly less than 2500, however, we have observed a transition only above 2500. In fact, once the LS regime changes to that of an open rim, a higher l_{pi} leads to a shorter breakup length. This has implications relatively to the application of these sprays when short atomization distances are required. Relatively to the misalignment or skewness, it generates a radial moment in the development of the liquid sheet, leading to its rotation [2]. In terms of the effect of the pre-jet-impingement length and jet misalignment, the analysis of the experimental results can be synthesized in the following:

- the pre-jet-impingement distance l_{pi} produces a significant influence on the liquid sheet breakup length in the open-rim regime, but not in the closed-rim.
- in the open-rim regime, higher impingement distances between the nozzle exit and the impact location (l_{pi}), result in shorter breakup lengths, smaller drop sizes, and more droplets. The interpretation suggested is related with the relaxation of the jet velocity profile which is higher for larger l_{pi};
- the misalignment of the jets induces a rotation of the liquid sheet, enlargement of its breakup length and, consequently, thinning of its thickness. In the case of open-rim regime, this amplifies the instabilities and antecipates breakup, decreasing its length;

Further studies will consider the fundamental hydrodynamics associated with multijet impingement sprays with more than two jets.

[1] Li, R., Ashgriz, N. *Physics of Fluids* 18:087104 (2006)
[2] Gadgil, H. P., Raghunandan, B. N. *Atomization and Sprays* 19:1-18 (2009)

*Corresponding author: mpanao@dem.ist.utl.pt

Modified TAB Model for Viscous Fluids applied to Breakup in Rotary Bell Spray Painting

B. Andersson*[1], V. Golovitchev[2], S. Jakobsson[1], A. Mark[1], F. Edelvik[1], L. Davidson[2], J. S. Carlson[1]
[1]Fraunhofer-Chalmers Centre, Chalmers Science Park, Göteborg, Sweden
[2]Department of Applied Mechanics, Chalmers University of Technology, Göteborg, Sweden
bjorn.andersson@fcc.chalmers.se, valeri@chalmers.se, stefan.jakobsson@fcc.chalmers.se, andreas.mark@fcc.chalmers.se, fredrik.edelvik@fcc.chalmers.se, lada@chalmers.se, johan.carlson@fcc.chalmers.se

Abstract

This paper describes the application of the Taylor Analogy Breakup (TAB) model [1] to simulate droplets size distributions obtained in rotary bell spray painting. This application technique is commonly used in the automotive industry for primer, color layers and clear coat. This paper deals exclusively with the clear coat paint material.

The properties of the clear coat differ from what the TAB model has been used for traditionally. In particular, the viscosity of the paint is approximately 100 times that of water. In addition, it is non-Newtonian but may be treated as Newtonian as long as the large shear rate is large enough. For this reason an empirical correction for large viscosity originally described by Brodkey [2] is applied to the TAB model. This correction introduces a dependency of the Ohnesorge number to the stability criterion that is in effect only for larger Ohnesorge numbers. The Ohnesorge number is expressed as $Oh = \mu/\sqrt{2\rho_l \sigma a}$, where μ is the dynamic viscosity of the liquid, ρ_l its density, σ is the surface tension coefficient, and a is the droplet radius. We see that the number increases with decreasing droplet size and that it therefore is not constant during the course of atomization where droplets break and successively becomes smaller and smaller.

The TAB model has previously primarily been used in combustion engine simulation and the spray paint simulation is a new application to the model. Its parameters therefore need to be tuned such that the simulated droplet size distributions match measured ones. The global optimization algorithm DIRECT [3] was used to this purpose. One of the five parameters of the model was found to not affect the obtained size distributions and the optimization was reduced to four free variables. Optimal parameter values found are presented for both the original and the modified TAB models.

Measurements of the droplet sizes in the spray were performed at the Fraunhofer-Institut für Produktionstechnik und Automatisierung (IPA) in Stuttgart, Germany with a Spraytec RTS 5001 from Malvern Instruments. A histogram of droplet sizes is obtained for each set of process parameters measured. A total of 17 parameter settings were measured where the paint flow rate, the shape air, and the bell rotation speed were varied. The largest effect on the measured size distributions came from the rotation speed, and the focus in the paper is to capture that effect by performing CFD simulations of the near bell region.

Three different rotation speeds are considered: 30, 40, and 50 thousand revolutions per minute (RPM). Parameters in the breakup model are tuned to all three cases simultaneously in order to obtain good agreement between simulated and measured size distributions. The dependency of the size distributions on the rotational speed is well captured and good results are achieved over a wide range of droplet diameters.

[1] O'Rourke P. J., Amsden A. A. *SAE paper 872089* (1987).
[2] Brodkey R. S., Addison-Wesley Series in Chemical Engineering, New York, NY, USA, 1967.
[3] Jones, D. R., Perttunen, C. D., Stuckman, B. E., *J. Optim. Theory Appl.*, 79:157–181 (1993).

*Corresponding author: bjorn.andersson@fcc.chalmers.se

Experimental Measurements of Impinging Jet Atomization at the Vicinity of Liquid Fan

Ryota Sakisaka[1], Jun Hayashi[*1], Yu Daimon[2], Nobuhiro Yamanishi[2],
Fumiteru Akamatsu[1]
Affiliation, Osaka University, Japan[1], Japan Aerospace Exploration Agency, Japan[2]
sakisaka@combu.mech.eng.osaka-u.ac.jp, j.hayashi@mech.eng.osaka-u.ac.jp,
daimon.yu@jaxa.jp, yamanishi.nobuhiro@jaxa.jp, akamatsu@mech.eng.osaka-u.ac.jp

Abstract

Atomization by the impingement of two liquid jets of fuel and oxidizer is utilized as the primary process of combustion in the liquid thrust chamber of liquid propellant thrusters because of the compact size and the reliability of reigniting. Liquid propellant thruster has many merits, for example easily controlled thrust and high specific thrust. Breakup mechanisms of impinging jet atomizers consists of three main steps, i.e. formation of liquid fan, disintegrate to the liquid fragment and breakup into droplets. These processes of atomization are affected by the influence of aerodynamic, viscous, inertial and instability wave appeared on the liquid surface. Atomized droplets are ignited by its hypergolicity in thrust chamber. Computer fluid dynamics (CFD) has been applied to the analysis and development of the combustion chamber to reduce the costs of development. Since it is difficult to deal with the atomization process with combustion process at the same time in CFD, a simple model of atomization is used in the CFD. To make precise prediction of atomization and combustion phenomenon in the liquid propellant thruster, it needs to verify the impinging atomization model to the actual phenomena. In this study, in order to verify the reported numerical model, experimental measurements of impingement atomization process were conducted by using high-speed imaging and phase Doppler anemometry.

To simplify atomization phenomena, like-on-like impinging jet atomizer is used. The water is used as the test liquid, which is fed from N_2 pressurized tank to the test nozzle. Two flow channels are completely separated in order to stabilize flow rate. Droplet size distribution and vertical velocities of droplets are measured by phase Doppler anemometry (PDA). The light source is an Ar^+ laser. The region of liquid fan and ligament appearance is estimated by the results of high speed imaging. The measurements are conducted under two different Weber number with 336 measurement points in the x-y plane. Here, the x-y plane is the same plane of liquid fan. The average characteristic values of spray (e.g. Sauter Mean Diameter (SMD), droplet velocity) are obtained by using 20,000 samples at each point. This is enough number to obtain the average value.

The liquid fan and ligaments region were found from results of the high speed imaging. The region of liquid fan and ligaments were smaller than that of the theoretical analysis. This was because liquid jets and liquid fan had the larger instability in its flow than theoretical analysis. The SMD was small under high Weber number condition at all measurement points. And the SMD increased with increasing azimuthal angle from vertical axis. In addition, the SMD increased with increasing distance from the disintegration point. This is because there are many non-spherical droplets, which is ignored in the PDA measurement, at vicinity to the disintegration point. Non-spherical droplets became spherical with increasing distance from the disintegration point, and became possible to be measured by PDA. And From results of the PDA measurements in horizontal plane, mass flux is weighted in the center axis though SMD doesn't change in a direction perpendicular to the liquid fan. Spatial gradient of the mass flux became smoother with increasing distance from the disintegrated point. Currently, impinging atomization model doesn't consider mass flux distribution in a direction perpendicular to the liquid fan. The results indicated that the mass flux distribution should be taken into account to make a precise prediction of impinging atomization.

Simulation of Electrostatic Rotary Bell Spray Painting in Automotive Paint Shops

A. Mark[*,1], B. Andersson[1], S. Tafuri[1], K. Engström[1], H. Söröd[2], F. Edelvik[1], J. S. Carlson[1]
[1]Fraunhofer-Chalmers Centre, Chalmers Science Park, Gothenburg, Sweden
[2]Swerea IVF, Mölndal, Sweden
andreas.mark@fcc.chalmers.se, bjorn.andersson@fcc.chalmers.se,
sebastian.tafuri@fcc.chalmers.se, klas,engstrom@fcc.chalmers.se, henrik.sorod@swerea.se
fredrik.edelvik@fcc.chalmers.se, johan.carlson@fcc.chalmers.se

Abstract

The main processes in automotive paint shops are electro coating, sealing and cavity wax, spray painting and oven curing. These processes are characterized by multi-phase and free surface flows, multi-physics, multi-scale phenomena, and large moving geometries, that pose great challenges for mathematical modeling and simulation. The current situation in the automotive industry is therefore to rely on individual experience and physical validation for improving the paint and surface treatment processes. Having access to tools that incorporate the flexibility of robotic path planning with fast and efficient simulation of the processes would be advantageous, since such tools can contribute to reduce the time required for introduction of new models, reduce the cycle-time, reduce the environmental impact and increase quality. In a joint ongoing project with Swedish and international automotive industry such tools are being developed.

In this paper the focus is on spray painting with the Electrostatic Rotary Bell Sprayer (ERBS) technique. Paint is injected at the center of a rotating bell; the paint forms a film on the bottom side of the bell and is atomized at the edge. The droplets are charged electrostatically and driven towards the target car body both by shaping air surrounding the rotating bell and by a potential difference in the order of 50-100 kV between paint applicator and target. Some earlier attempts can be found in the literature, but a systematic validation for realistic geometries is missing. Another major drawback is that the simulation times are prohibitively long for the tools to be industrially useful. This is partly due to the fact that the simulation methods do not handle moving geometries in an efficient way. The aim of this paper is to present a new framework for simulation of electrostatic spray painting based on novel algorithms for coupled simulations of air flow, electromagnetic fields and paint droplets. Particularly important for the computational efficiency is the Navier-Stokes solver. The incompressible solver is based on a finite volume discretization on a dynamic Cartesian octree grid and unique immersed boundary methods are used to model the presence of objects in the fluid. This enables modeling of moving objects at virtually no additional computational cost, and greatly simplifies preprocessing by avoiding the cumbersome generation of a body conforming mesh. The virtual paint software is included in an in-house package for automatic path planning, IPS.

The input to the simulations of the paint film build-up on a target is the process conditions (paint flow, shaping air, downdraft speed, and applicator rotation speed), the robot path, a CAD description of the target geometry, and measurements of the droplet size distribution and air velocities close to the bell. In the results section of the full paper the results from an extensive measurement campaign, which was performed to validate the simulation software, is presented. Several test plates and car fenders were painted with different process conditions and robot paths. The same cases were then simulated and overall the agreement between simulations and experiments are remarkably good. The very efficient implementation gives a major improvement of computational speed compared to other approaches and makes it possible to simulate spray painting of a full car in just a few hours on a standard computer.

* Corresponding author: andreas.mark@fcc.chalmers.se

Drop Size and Drop Size Distribution Measurements by Image Analysis

J. B. Blaisot
CORIA UMR 6614, University of Rouen, France
blaisot@coria.fr

Abstract

One important issue in drop sizing by image analysis is the determination of the contour of the drops. As image is a signal particularly well adapted to human perception, it is often considered that raw data from an image can deliver quantitative information on the object imaged without needing any underlying physical model. This is not sufficient for precise determination of drop contours in the image of a spray as the grey level corresponding to the contour strongly depends on the degree of focusing of a drop. It is shown in the present paper how an imaging model can help in determining this contour and thus in measuring the drop size. Recent advances made on an imaging model developed for application to spray sizing and drop size distribution measurements is presented. The imaging system is modeled by its point spread function (PSF) which can be considered as the response of the system to an infinitely small object source. Digital image capturing systems are classified in two categories, depending on the PSF width compared to the pixel width. Optical systems under consideration belong to the optics limited (OL) category for which the PSF is covered by several pixels. This is an important feature the imaging system must fulfill to apply criteria derived from the model. Also, even though objects under consideration are transparent liquid drops, it is shown that refraction can be neglected since those criteria are based on measurement made at the outline of the drop images.

The optical settings of the imaging configuration must be carefully adjusted. Using a backlight configuration and a collimated light source is suggested to obtain high contrast images and a large DOF. An imaging model is used to derive useful relation for drop sizing application. The principle of the model, modeled image profiles and image parameters relevant for drop sizing are introduced. It is shown that image profile is controlled by two parameters that depend on the object properties (size, position, opacity) as well as on the imaging system properties (PSF). Image contrast, image width and grey level gradient at image edge are then used to characterize image profiles properties and to derive relation between object and image properties.

A sizing criterion and a focus selection criterion are derived from these relations. The sizing criterion allows determining the grey level to be used for locating the image contour and measuring the drop size. This is the main advantage of the model. Indeed, no calibration or correction is done on measurements made on the image to obtain the drop size. This criterion is based on the adjustment of the reference grey level, depending on the value of the image contrast. The focus selection criterion is based on the measurement of the PSF half-width and implies that an OL system must be used. It uses a relation between the grey level gradient and the image contrast. The derived criterion does not depend on the size of the drop. It is thus used to select droplets in a spray according to their focus position, independently from their size.

An application to spray measurement is presented. It is shown that the sizing criterion allows measuring drop size over a large DOF. The sizing error remains of the order of the pixel size over the range of measured diameter [10µm – 1500µm] when focus selection criterion is used. The selection criterion, coupled to the calibration of the PSF of the system, determines an effective depth of the measurement volume about 0.6 mm. It is shown that increasing this depth must go with an increase of the minimum diameter of the sizing range. If not, underestimation of the drop size distribution may occur for the smallest droplets. Sizing error due to an underestimation of the reference grey level is also evaluated and found to be less than 7% for acceptable contrast error. Image segmentation and sub-pixel contour extraction techniques are briefly presented at the end of the paper. These techniques are used to enhance the sizing procedure. Indeed, the complex image segmentation technique, using continuous wavelet transform computation, allows detecting of almost all the drops that are visible to the naked eye in an image. Real-coordinate contour obtained from sub-pixel contour extraction is analyzed as a curve in a continuous space described by a limited set of points. Size and shape parameters determined from this contour do not present discrete values, even for the smallest droplets. This leads to an effective enhancement of the sizing and of the shape analysis procedures. Partial overlapped droplet images are detected from the analysis of sub-pixel contour analysis.

Detailed Numerical Analysis of X-ray Radiography in Sprays

Mark A. Linne[1*]
Combustion Engine Research Center, Department of Applied Mechanics
Chalmers University, Gothenburg, Sweden
mark.linne@chalmers.se

Abstract

Recent studies of spray-related flowfields using synchrotron-based x-ray radiography at the Advanced Photon Source (APS), Argonne National Lab, have produced useful results related to fuel mass fraction. For some, however, it is not clear whether or not the technique can observe intact structures (e.g. a liquid core) inside a spray or not. Here we simulate x-ray radiography using a code that solves the full diffraction problem (Maxwell's equations subject to assumptions that apply in this case) to model accurately several common spray architectures found in the literature. One important finding is that radiography detects the total mass along a line of sight, including intact liquid, drops and in some cases vapor and gas. Under appropriate conditions, radiography reveals liquid mass fraction which is related to the spray breakup rate and gas entrainment; both critical for understanding of sprays. The single point system used at the APS provides a spatially and temporally resolved (but averaged over a number of injections) determination of liquid fuel mass fraction. The potential for confusion between local signal reduction by diffraction and local signal reduction by absorption is also discussed. The issue is not unusual in such circumstances. This potential background problem can be avoided by careful arrangement of the experiment and it is clear that it does not affect the results produced by the Advanced Photon Source.

* Corresponding author: mark.linne@chalmers.se

Shadowgraphy investigations of high speed water jet atomization into still air

C. Stevenin[*,1], S. Tomas[1], A.Vallet[2], B. Amielh[3] and F. Anselmet[3]

[1] LERMI, Irstea, 3275 route de Cézanne, 13182 Aix-en-Provence Cedex 5, France
[2] ITAP, Irstea, 361 rue J.F. Breton, 34196 Montpellier Cedex 5, France
[3] IRPHE, 49 rue F. Joliot Curie, B.P. 146, 13384 Marseille Cedex 13, France

Abstract

Irrigation sprinkler system is very popular in Europe. To improve its application efficiency, in particular as regards water distribution uniformity, water losses and environmental impact of spraying, a better knowledge of the size and spatial distribution of the generated droplets is necessary. The present study aims at characterizing the atomization of a large-scale pressure-atomized jet of water. The water jet is injected at high velocity into still air thanks to an industrial nozzle particularly used in irrigation. The nozzle diameter is 4.36 mm and exit velocity is about 25 m/s.

The droplet size and velocity are analysed by a double-pulsed shadowgraph imaging technique. A particular attention is paid to the calibration of the imaging system, both to define the measurement volume associated to each droplet size and to improve the estimation of droplet diameters, especially concerning the apparent size of unfocused droplets. For simplicity glass spheres are used instead of droplets in the calibration procedure. The depth of volume measurement is found to be strongly related to the object size. Furthermore, different image processing methods have been tested to estimate the size of non-spherical liquid fragments.

As the spray width is larger than the depth of volume measurement, a number of droplets are not detected by our imaging system. Moreover the spray is very heterogeneous, with more and larger droplets near the jet axis. In order to take into account the whole population of droplets present in the spray at a given distance from the injector, the spray is supposed to be axisymmetric. The validation of the axisymmetric assumption is in progress by getting volume flux distribution using a mechanical patternation technique.

Eventually, drop size distributions are obtained by image processing as a function of the turbulent scales of the upstream flow, as well as Reynolds and Weber number. Particle Tracking Velocimetry (PTV) is used for the investigation of droplet velocities. Preliminary results show that log-normal distributions give the best fit to the experimentally observed drop size distributions. Moreover the ratio of the mass median diameter (MMD) over the Sauter mean diameter (SMD) is found to be constant and equal to 1.2, which is in good agreement with previous studies, and yields to a relation between the mean and the standard deviation of lognormal distributions.

In the near future, an Eulerian model, developed in a previous work and validated up to one hundred diameters from the nozzle, will be extended to the far field and coupled with a Lagrangian approach. Numerical results will be compared with our experimental data of droplet size and velocity measurements and with Particle Image Velocimetry (PIV) measurements in the liquid core.

* Corresponding author: christophe.stevenin@irstea.fr

Characteristics of 3000 bar Diesel Spray Injection under Non-Vaporizing and Vaporizing Conditions

Jaclyn E. Johnson[1], Seung Hyun Yoon[1], Jeffrey D. Naber[1], Seong-Young Lee[1], Gary Hunter[2], Russell Truemner[2], Tony Harcombe [3]*
[1]Michigan Technological University
Mechanical Engineering – Engineering Mechanics Department
Houghton, MI 49931, USA
[2]AVL Powertrain Engineering, Inc.
Plymouth, MI 48170, USA
[3]Delphi Diesel Systems
London W3 OSE, England

Abstract

Increasing fuel injection pressure has enabled continuous reduction of diesel emissions while sustaining the high thermal efficiency advantage of diesel engines. Current production diesel injectors operate in the range from 300 to 2000 bar. The ongoing trend for fuel injection systems is to higher injection pressures and smaller nozzle hole diameters for further emissions reduction and fuel efficiency improvements. Fundamental understanding of diesel spray characteristics including liquid penetration and cone angle is imperative to improve model development and facilitate the integration of elevated injection pressure systems into future diesel engines.

Studies were conducted in an optically accessible constant volume combustion vessel under non-vaporizing and vaporizing conditions. A 7-hole injector, currently being developed for high injection pressure applications, was studied between 2000 and 3000 bar injection pressures with ultra-low sulfur diesel fuel. The study included two part-load charge density conditions of 7.4 kg/m^3 and 14.7 kg/m^3 along with an elevated density boosted condition of 34.8 kg/m^3. Diagnostics used included Mie back scatter imaging for liquid phase penetration. Experimental results were compared to spray penetration relationships to extrapolate these relationships to the elevated injection pressure conditions. Thus, an improved understanding of the influence of elevated injection pressure on fundamental spray characteristics was gained.

---

* Corresponding Author: jenesbit@mtu.edu

Spatially Resolved Characteristics and Analytical Modeling of Elastic Non-Newtonian Secondary Breakup

Sharon Snyder, Paul E. Sojka*
Department of Mechanical Engineering
Purdue University, West Lafayette, IN USA
snyder22@purdue.edu, sojka@ecn.purdue.edu

Abstract

The secondary breakup of elastic non-Newtonian liquid drops will be investigated experimentally to determine fragment size and velocity distributions. Knowledge of these parameters will help with the design and/or optimization of many technical systems such as gas turbine combustors or film coating processes. To do this, Xanthan gum – water solutions of various concentrations ranging from 0.1% to 1.0% by weight are prepared and injected into a high speed air stream. Both PIV and LDA techniques have been employed to characterize the air stream velocity. The fragment size and velocity distributions of elastic liquid fragments resulting from secondary atomization will be characterized using a dual-PDA system (Dantec Dynamics GmbH, Germany). The fragment size and velocity distributions of the wake region of the fragmenting drop will be plotted against the Weber and Ohnesorge numbers to investigate how the two distributions change with respect to these parameters. The primary focus will be on the bag and bag and stamen (multimode) breakup regimes, with work extending to the sheet-thinning regime if fragment geometries allow. Previous work focusing on the resulting fragments on secondary breakup has mainly studied the fragment sizes, not velocities. Work by Villermaux and Bossa [3] was focused on the distribution of fragment sizes. The end result was a plot of drop size distribution versus initial drop diameter. Since Villermaux and Bossa [3] studied water droplets, the effects of liquid properties (density, surface tension, etc.) were not taken into account. Also, since the drops in this study fell due to gravity, the effects of varying air velocity were not studied. In the work of Gast [2], the focus was only on the breakup of the drop rim, even though the conditions were such that the drops were undergoing bag breakup. Again, water was the only solution used, and the drop diameter was the only variable changed. Gast [2] used time-lapse photography to determine if there was a relationship between initial drop diameter and torus diameter; his focus was mainly on torus geometry, not fragment properties. Most recently, the products of drop fragmentation have been studied by Bartz *et al*. [1]. We expect to produce plots of fragment size versus axial location that are similar to Bartz *et al*.'s [1] findings (as shown in Figure 1):

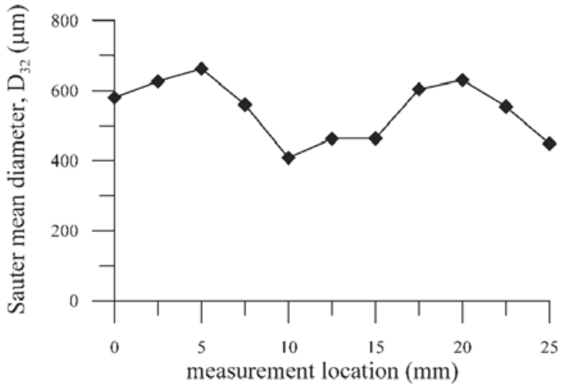


Figure 1: D_{32} versus axial measurement location [1].

This work aims to extend the knowledge of drop fragment size and velocity distribution to the elastic non-Newtonian case. We will attempt to determine how these distributions vary with respect to important non-Newtonian fluid parameters (namely Weber and Ohnesorge number). By studying the bag and bag and stamen breakup regimes, we will cover a wide range of flow and distribution conditions.

Keywords: non-Newtonian, secondary breakup, secondary atomization, liquid drops

[1] Bartz, Frank-Oliver, *et al*. "Comparison of droplet breakup models for single droplet fragmentation under varying accelerations." *ILASS – Europe 2011, 24th European Conference on Liquid Atomization and Spray Systems, Estoril, Portugal*. September, 2011.
[2] Gast, Luis A. "Capillary Instability of a Liquid Ring." (Doctoral Dissertation). Rutgers The State University of New Jersey; 1991.
[3] Villermaux, E., and B. Bossa. "Single-drop fragmentation determines size distribution of raindrops." *Nature Physics*, **5**, p. 697 – 702, 2009.

* Corresponding author: sojka@ecn.purdue.edu

Experimental investigation of liquid ligament fragmentation

C.M. Weickgenannt *[1,2], I.V. Roisman[1,2] and C. Tropea[1,2],
[1]Center of Smart Interfaces and [2]Institute of Fluid Mechanics and Aerodynamics,
Technische Universität Darmstadt, Petersenstr. 32 and 30, 64287 Darmstadt, Germany.
weickgenannt@csi.tu-darmstadt.de, roisman@sla.tu-darmstadt.de,
ctropea@sla.tu-darmstadt.de

Abstract

Fragmentation of liquid ligaments is a phenomenon which is observable in different industrial applications. For example in roll coating or printing, when paint films split between rollers, ligaments are formed and produce unwanted droplets [1]. Also, it is one of the fundamental processes in spray generation since atomization of a liquid surface most often implies the formation of liquid ligaments which are stretched out of the surface, elongate and break-up into single droplets [2]. The dynamics and fragmentation mechanisms of those stretched ligaments are investigated experimentally in this work. The experiments are performed with pure Newtonian liquids and suspensions of spherical glass particles (3-20 µm) in water with mass fractions up to 0.8.

Methodologies

Single liquid ligaments were produced and stretched using the experimental setup shown in Fig. 1. It consists of two parallel glass plates, from which one is mounted at a fixed position while the other one, movable, is connected with a linear driving system. The motion of the movable plate is accurately controlled. By using a displacement pipette a defined amount of liquid is deposited onto the movable plate which then drives towards the fixed plate until a liquid bridge is formed in the gap between the plates. After this a ligament is created stretching the bridge uniaxially by driving the movable plate apart from the other. Two high-speed camera systems allow observation of stretching and break-up processes from a top-view and side-view.

The main advantage of using a controllable linear motor for stretching the liquid bridge is the accuracy and reproducibility of the experiments. Furthermore it is possible to investigate the influence of velocity and acceleration of the movable plate on ligament stretching and break-up in detail.

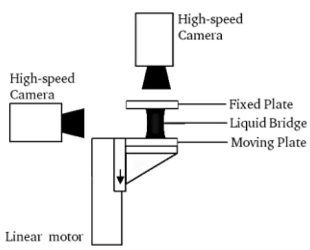


FIG. 1 Experimental setup.

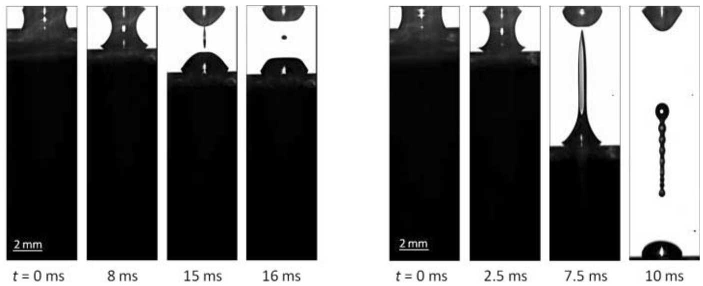


FIG. 2 Side view on ligament stretching and breakup at two different magnitudes of the plate acceleration: a = 10 m/s^2 (left) and 150 m/s^2 (right).

Sample results

In series of experiments the acceleration was varied from a = 10 to 150 m/s^2. Some examples of ligament breakups at various accelerations are shown in Fig. 2. These images are used for estimation of the kinematics of the ligament stretching (variation of its diameter and length), measurements of the typical breakup time and evaluation of the distribution of the fragment size.

The influence of accelerations of the movable plate as well as liquid properties and particle concentration on the kinematics of ligament stretching and breakup were investigated in this work.

References

[1] James, D. F. and Pouran, M., *Rheol Acta* 48: 611 (2009).
[2] Marmottant, P. and Villermaux, E., *J. Fluid Mech.* 16: 2732 (2004).

* Corresponding author: weickgenannt@csi.tu-darmstadt.de

Kinetics of Liquid Film Disintegration at "Bag" and "Claviform" Modes of Drop Breakup

A. G. Girin[1*], Ye. A. Ivanchenko[2]

[1]Odessa National Maritime University, [2]Odessa State Institute of Finance, Odessa, Ukraine

[1]club21@ukr.net [2]jeki1@rumbler.ru

Abstract

The process of "bag" film disintegration at "bag" and "claviform" modes of drop breakup is connected with mechanism of hydrodynamic instability of Kelvine – Helmholtz type under action of air stream, and theoretical model for liquid film breakup is elaborated. The investigation of hydrodynamic instability of drop surface in gas flow allowed to explain [1] the mechanism of blowing off of a thin wide film of bag from parent drop after extreem drop deformation. It helps us to estimate the important values of the liquid mass in the ring and the thickness of the bag film, which are necessary to evaluate sizes of daughter droplets.

The model of liquid film disintegration is developed, which describes two stages of the process – initial perforation of the film and further dynamics of the expanding hole, which is accompanied by film splashings. The process of liquid film fragmentation itself is connected in our study also with mechanism of hydrodynamic instability of film in air stream.

After initial film rupture, caused by instability, behavior of liquid film near the hole edge is governed by two factors: liquid gathering there into moving toroidal roller by the action of surface tension forces and further action of instability mechanism near the streamlined film edge. When characteristic time of first process is smaller, than of the second one, the surface tension grasps significant amount of liquid into roller before unstable disturbance shakes it off from the edge. Therefore, the diameter of the roller and the sizes of shaken-off droplets are greater than thickness of the film.

Solution of a simplest problem about instability of liquid film in gas flow was applied to determine the characteristic size and time of dominant unstable disturbance. The comparison of the former with experimentally observed [2] values of perturbations on film, which are probably produce initial rupture, showed a qualitative agreement.

At simplest assumptions (media are ideal, film is plane, liquid roller has a form of torus) the ordinary differential equation of torus motion under action of surface tension forces and reactive forces is derived and laws of torus motion (for mass center of its cross section) and history of its radius are calculated. Taking into account simultaneously both mechanisms of unstable shaking-off and of the liquid roller growth, the main regularities of film disintegration kinetics have been calculated: histories of roller radius and of quantity of shaken-off droplets, sizes and moments of their generation and the period of the entire film disintegration.

The comparison of computed values with experimentally observed [3] values of daughter droplet sizes, dispersity, film thickness and volume of liquid ring showed a good agreement.

References

[1] Girin A. G., *12th ICLASS 2012*, Heidelberg, Germany, Sept. 2-6, 2012, # 1225

[2] Hanson A.R., Domich E.Y. & Adams H.S. *Phys. Fluids* **8**: 1070–1080 (1963)

[3] Chou W.-H., Hsiang L.-P. & Faeth G. M. *Dynamics of drop deformation and formation during secondary breakup in the bag breakup regime,* AIAA Paper: 97-0797 (1997)

* Corresponding author: club21@ukr.net

Turbulent secondary atomization of non-evaporating dilute spray jets

A. Kourmatzis*, A. R. Masri
Aerospace Mechanical and Mechatronic Engineering, The University of Sydney, NSW 2006, Australia
akourmatzis@sydney.edu.au and assaad.masri@sydney.edu.au

Abstract

The secondary atomization characteristics of dilute spray jets of mineral turpentine with varying levels of turbulence are investigated using phase Doppler anemometry (PDA). The choice of mineral turpentine as the injected liquid ensures no evaporation at room temperature and a dilute spray is utilized to avoid droplet-droplet interactions. The spray is formed upstream of a pipe and is carried with air to the jet exit plane. The influence of turbulence on secondary atomization is studied in this paper via presentation of the Sauter mean diameter (SMD), droplet diameter probability density functions (PDFs), and scatter plots of the droplet Weber (We_d) vs. Ohnesorge (Oh_d) number. A range of Reynolds numbers from ∼12,000-37,000 are tested with tube lengths varying from 4.7 to 43 jet diameters. The focus is on data from measurements taken at the tube exit planes where the effects of dispersion are minimal. All of the aforementioned simplifications allow for the authors to attribute changes in droplet diameter predominantly to secondary atomization. Scatter plots of the droplet We_d vs. Oh_d reveal that in the investigated geometry, $We_d << 10$ for $Oh_d < 0.1$ due to the low droplet slip velocity, indicating that only droplet deformation would be occuring in an analogous non-turbulent gas flow. However, it is found that the SMD decreases by as much as $20\mu m$ with increasing Reynolds number and by $10\mu m$ with increasing tube length. Increasing the tube length from 4.7 to 43 diameters whilst keeping the Reynolds number constant results in a different flow profile at the exit plane, varying from under-developed, to transitional, and finally to a fully developed turbulent flow. This increase in tube length leads to a consistent decrease in the SMD of the dilute spray, acting as evidence of turbulence enhanced secondary atomization.

Keywords: Secondary atomization, Turbulent Atomization, Dilute Spray, PDA

*Corresponding author: akourmatzis@sydney.edu.au

Acceleration Effects on Instability of High-Pressure Fuel Jets

D. Jarrahbashi*, W. A. Sirignano
University of California, Irvine, USA
djarrahb@uci.edu and sirignan@uci.edu

Abstract

The transient behavior of the jet during the start-up and shut-down portion of the injection at very high Reynolds and Weber numbers is addressed in the present work. The acceleration of the liquid during start-up is about 10^6 m/s^2 at the orifice exit for high Reynolds numbers. The influence of acceleration on the dynamics of jets has never been fully considered previously. When the jet emerges from the orifice, drag forces due to the dense ambient air cause a deceleration. Also, the dynamic protrusions from the jet surface created by Kelvin-Helmholtz (KH) instability are subject to local accelerations. The Rayleigh-Taylor (RT) instabilities are driven by acceleration when the liquid accelerates away from the gas locally. With this instability, the waves corrugate at the free surface during acceleration. Ultimately, these waves will finger into the liquid, causing it to break up. The effects of the RT and KH instabilities on jet break-up during start-up and shut-down transient are considered in this research. During start-up and shut-down, the jet exit velocity varies producing an acceleration which enters into the equations of motion transferring from the laboratory frame to an accelerating frame fixed to the liquid mass center as a generalized body force. In addition, the fingers have an accelerating motion, even during steady injection. At very high Reynolds and Weber numbers, calculations show that the unstable wavelengths could be as small as a few microns. To tackle the resolution problem and capture the shortest unstable surface wavelengths, we examine stream-wise segments of the jet, treating these segments as ballistic slugs coming from the orifice. This slug or liquid section of the jet deforms and exchanges both mass and momentum with the surrounding gas during start-up and shut-down. Use has been made of the unsteady multidimensional code with a finite-volume solver of the Navier-Stokes equations for liquid streams and adjacent gas, a boundary-fitted-gridding scheme, and a level-set method for gas/liquid interface tracking. We also simulate the full transient jet during the transients. These results have been very fruitful for an estimate of the acceleration and also for implementing the proper boundary conditions in the liquid-section model. The effects of the acceleration on the surface instability and the range of unstable wavelengths in the liquid-section model have been compared with the classical instability theories.

* Corresponding author: djarrahb@uci.edu

Breakup Behavior and Encapsulation Regime of a Non-Newtonian Viscous Compound Liquid Jet

T.Yoshinaga* and K.Matsumoto
Department of Mechanical Science, Osaka University, Osaka 560-8531, Japan
yoshinag@me.es.osaka-u.ac.jp

Abstract

We analytically examine breakup and encapsulation phenomena of a viscous compound liquid jet which consists of core and surrounding annular phases. Considering the non-Newtonian viscosity described by the Carreau model, a set of reduced nonlinear jet equations is analytically derived by means of a long wave approximation. The equations are numerically solved when the jet is semi-infinite and sinusoidal disturbances are applied at a nozzle exit of the jet. Typical breakup profiles are shown for the surface tension ratios, Reynolds and Weber numbers when the liquid viscosity is pseudo-plastic, Newtonian and dilatant. It is found that the non-Newtonian effects are salient for low Reynolds numbers when the surface tension ratio is not sufficiently small. Then, there is a tendency that the jet breaks up through disintegration of the annular phase for the pseudo-plastic, while the breakup appears through ballooning or closing of the annular phase for the Newtonian and dilatant, where the closing which leads to the encapsulation is more dominant for the dilatant. It is also found that influence of the non-Newtonian viscosity significantly appears near the breakup of the jet.

---

*Corresponding author: yoshinag@me.es.osaka-u.ac.jp

Assessment of an Eulerian atomization model on diesel spray CFD simulations

J.M. García-Oliver, J.M. Pastor*, A. Pandal
CMT-Motores Térmicos, Universitat Politècnica de València, Spain
jgarciao@mot.upv.es, jopasen@mot.upv.es, adrianpandal26@gmail.com

N. Trask, D. P. Schmidt
Departament of Mechanical and Industrial Enginnering, University of Massachusetts, Armhest, USA
nat.trask@gmail.com, schmidt@acad.umass.edu

Abstract

Diesel spray numerical modeling is still a challenging task due to the complex phenomena taking place and the wide range of spatial and time scales required to capture the physics involved. The aim of this work is to evaluate the so-called *Σ-Y* Eulerian atomization model, originally proposed by Vallet & Borghi, for CFD simulations of diesel engine sprays. This model has emerged as an alternative to the traditional discrete particle methods, widely employed on practical design applications for more than twenty years. Those Lagrangian liquid spray models are not well suited for the description of the primary atomization and the dense two-phase flow occurring at the near field of diesel sprays, where basic model hypothesis cannot be fully accomplished.

The *Σ-Y* model was developed on the basis of an Eulerian representation of the liquid/gas mixture by means of a single-fluid variable density turbulent flow. It is assumed that under high Reynolds and Weber numbers, large scale flow features are independent of viscosity and surface tension; but they affect the smallest scale flow, i.e., the size of the liquid fragments. From this assumption, spray liquid dispersion into the gas phase could be evaluated as the turbulent mixing of a variable density flow**.** The extent of the atomization process is computed from an interphase surface density equation (*Σ*), and then it is not required to presume any particular shape for liquid fragments. Liquid dispersion is calculated by means of liquid mass fraction (*Y*) transport equation using a traditional turbulent diffusion flux closure.

In this work a finite-volume solver for *Σ-Y* model equations has been written using the open-source CFD C++ library OpenFOAM. Model predictions have been compared to experimental data from free diesel sprays injected into a non-vaporizing quiescent vessel with ambient density conditions typical of current automotive DI diesel engines. Spray macroscopic characteristics were obtained by means of high-speed imaging. In order to evaluate droplet velocity field and size, Phase Doppler particle analyser (PDPA) technique measurements were also performed at different spray axial and radial locations. Single-hole conical nozzles were used in those experiments. Nozzle flow characterization has been obtained from mass flow rate and momentum flux measurements.

Different numerical and physical sub-model parameters effects on spray modeling results have been evaluated on a reference test case. The model set-up study indicates that accurate predictions of spray penetration as well on axial and radial velocity profiles can be simultaneously achieved. This result is obtained when κ-ε turbulence model constant $C_{1\varepsilon}$ for dissipation equation is set to typical values employed for round jet simulations. Further parametrical studies indicate that the proposed *Σ-Y* model approach and computational set-up remains valid for a broad range of injection pressure (30 to 130 MPa) and ambient density conditions (10 to 40 kg/m^3). Model accuracy is worse for low ambient density and injection pressure conditions. It is proposed that under these particular conditions, the slip between phases becomes more significant and the single velocity field assumption is less appropriate.

* Corresponding author: jopasen@mot.upv.es

Analysis of Disintegration of Planar Liquid Sheet Sandwiched between Gas Streams with Unequal Velocities and Resulting Spray Formation

Sujit Nath[1], Achintya Mukhopadhyay[1,*], Amitava Datta[2], Swarnendu Sen[1]
[1]Mechanical Engineering Department, Jadavpur University, Kolkata – 700032, India
[2]Power Engineering Department, Jadavpur University, Kolkata – 700098, India
sn_ju_mech@yahoo.com, achintya.mukho@gmail.com, sen.swarnendu@gmail.com, amdatta_ju@yahoo.com

Abstract

A temporal stability analysis was carried out for a planar liquid sheet sandwiched between two gas streams moving with unequal velocities. A planar sheet is an idealized representation of an annular liquid sheet emanating from an injector owing to the high ratio of the radius of the liquid sheet to its thickness. To account for the nonlinearity of the breakup process, regular perturbation analysis was carried out up to second order using the initial amplitude of disturbance of the liquid sheet as the perturbation parameter. Unlike the case of planar sheets subjected to gas streams moving with equal velocity on either side, the present problem leads to para-sinuous and para-varicose modes giving phase differences between the two interfaces close to 0 and π respectively. In the present work, both these modes were investigated to determine the dominant break-up mechanism. On account of the high velocities typically encountered in injectors like air blast atomizers, both the liquid and the gas flows are assumed inviscid and irrotational. This enables use of velocity potentials for both the phases. The two phases are coupled through kinematic and dynamic boundary conditions at the interface. Assumption of normal modes for the perturbed interface and use of the kinematic and dynamic boundary conditions lead to dispersion relations at each order in the form of ordinary differential equations. These equations are solved using Laplace transform. The nonlinear stability analysis gives the ligament area and the breakup time corresponding to the most unstable wave numbers in para-sinous and para-varicose modes. Gaster transformation is used to obtain the breakup length from the breakup time. Secondary breakup of the ligament into droplets was modeled following Rayleigh mechanism.

Apart from modeling the breakup of the liquid sheet, droplet size and velocity distributions were predicted using Maximum Entropy Formulation (MEF). In the MEF approach, the joint size and velocity probability distribution that maximizes the information entropy subjected to certain constraints was considered the most probable distribution. In the present work, the constraints were based on the conservation of mass, momentum and energy of the liquid drops. The droplet diameter predicted by the breakup analysis is used as the mass mean diameter in the MEF and the breakup lengths obtained from the stability analysis are used in the constraints relations.

The results show that the asymmetry in the gas velocity significantly affects both the first order and the second order results. The liquid sheet profile shows considerably higher asymmetry before breakup compared to that observed with equal velocities. The range of unstable wavelengths, the maximum growth rate, the breakup length and the mean droplet diameters were found to be determined through a complex interaction of the gas phase velocities and their differences. In general, it is found that increase in difference the gas velocities on two sides of the liquid sheet leads to a significantly narrower size distribution in the spray though the velocity distribution is affected much less.

---

* Corresponding author: achintya.mukho@gmail.com

A comparison of non-reactive fuel sprays under realistic but quiescent engine conditions for SGDI

P. Marti-Aldaravi[*,1], T. Rogers[2], E. de Benito[3], S. Hemdal[3], P. Dahlander[3]
[1]CMT - Motores Térmicos, Universidad Politécnica de Valencia, Valencia, Spain
[2]School of Aerospace, Mechanical & Manufacturing Engineering, RMIT University, Melbourne, Australia
[3]Department of Applied Mechanics, Chalmers University of Technology, Gothenburg, Sweden

Abstract

A comparative study on the two most commonly found gasoline direct injectors is presented, where a solenoid driven multi-hole (6 horseshoe hole) and piezo driven outward opening injector were evaluated on liquid penetration, spray width and spray structure within a constant volume chamber. These three variables have been investigated for three typical fuels (iso-octane, gasoline 95 and e100 ethanol) at a fixed calorific delivery value of 389.4 J, typical combustion required value for stratified road-load operation. A first series of tests allowed correlating mass flow and injection duration for each injector and fuel. The chemical properties of the three fuels were used to calculate the injection duration for the target calorific value. This energy value was determined previously by tests in a single cylinder research engine at stratified operation. The non-impinging, non-combusting spray was visualized using back-lit high speed photography. The pressure and temperature values set on the chamber correspond to SOI of 20, 30 and 40 CAD bTDC during engine testing at 2000 rpm and an IMEP of 2.5 bar for an overall lean operation of $\lambda = 4$. The spray visualization was also carried out at ambient conditions (25°C of temperature and 1 and 6 bar of back pressure). The results show that the penetration length is function of ambient temperature and pressure, fuel and injector type. The solenoid driven multi-hole injector produces longer penetration lengths and at a faster rate than the piezo unit under all test parameters. Moreover, there is greater variance in penetration curves for the fuels tested using the solenoid multi-hole injector compared to the piezo actuated outward opening injector. Despite wall-wetting aspects have not been tested, larger variance in penetration curves for different fuels in the solenoid multi-hole injector indicate that it is less suitable for bi-fuel engines. Thus, spray targeting differences will lead to potential increase in combustion stability (COVimep), increase in emissions and increase in wetting of surfaces i.e. sparks plug, bore or piston.

---

[*]Corresponding author: pedmar15@mot.upv.es

Pulsation dampers for combustion engines

F.Durst, V. Madila, A.Handtmann, M.Zeilmann[1]
FMP TECHNOLOGY GMBH, Albert Handtmann Holding GmbH & Co. KG
info@fmp-technology.com, v.madila@fmp-technology.com, info@handtmann.de,
m.zeilmann@fmp-technology.com

Abstract

Fast operating injectors produce pressure pulsations because of the opening and closing of the valve. These pressure waves run through the whole injection system which causes pressure differences of $\pm 20\%$ of the mean system pressure. Therefore, the mass injection into the combustion chamber is also uneven which leads to an uncertainty in the combustion. This mismatch of the air-fuel ratio affects the performance of the combustion process and increases the emissions.

Starting with Navier-Stokes equations one can show by neglecting gravitational and viscosity influence that a change in mass flow causes a pressure wave.

$$\rho \frac{DU_j}{Dt} = \frac{\partial(\rho U_j)}{\partial t} = -\frac{\partial P}{\partial x_j} = -\frac{1}{const.}\frac{\partial P}{\partial t}$$

A typical pressure profile for an Otto injector shows exactly the above described behavior that a pressure wave runs through the entire injection system and influences the pressure in the common rail and the connected injectors. By using the commercial software AMESim, a theoretical model can be setup for a simple injection system. Afterwards one can investigate the influence of different parameters like system pressure, injection time, mass flow rate, etc. to make an analytical approach for a damped system.

$$\Delta P = \frac{8\,\mu\,\dot{V}}{\pi\,r^4}\,l$$

$$\Delta P \approx 2\,P_0\,cos\left(2\,\pi\,f\,\left(t-\frac{l}{2\,c}\right)\right)\,sin\left(2\,\pi\,f\,\frac{l}{2\,c}\right)$$

Based on these formulae one can design damper elements which can be tested afterwards. The used test-rig for the experimental studies consists of a common rail with two connected injectors and pressure sensors. It can be demonstrated that the pressure pulsations are removed completely in the common rail and in the passive injector and also mostly in the active injector. Therefore each injector can be decoupled by using these pulsation dampers because an interaction is prevented. Thus only minor pressure pulsation occur at the active injector. Furthermore, it can be shown numerically with AMESim that a dying-out time of the pressure pulsations of less than $0{,}30\ ms$ is possible. Also, different damper materials and damper configurations have been investigated numerically and experimentally to show their potential for applications and the possibility to optimize these damper elements regarding the pressure drop or the dying out time.

---

[1] Corresponding author: m.zeilmann@fmp-technology.com

Investigations of Twin-Jet Sprays for DISI Engine Conditions

M. Heldmann[1,*], M. Wensing[1]
S. Wanner[2,*], S. Franz[2], S. Schneider[2], A. Handtmann[2]

[1]: Dept. Engineering Thermodynamics and Erlangen Graduate School in Advanced Optical Technologies (SAOT), FAU Erlangen-Nuremberg, Germany

[2]: Handtmann Systemtechnik GmbH & Co. KG, Arthur-Handtmann-Str. 7-1, D-88400 Biberach/Riss, Germany

Abstract

A well atomized spray enhances the evaporation process in gasoline engines. This work studies a sheet break-up mechanism formed by two impinging jets as a more effective alternative to the commonly used pressure driven break-up in multi-hole nozzles.

Figure 2 shows exemplarily three different shadowgraphy single shot images (view: max. radial expansion) for three different nozzle geometries, for which the bore-hole diameter (I → II) and the impinging angle (II → III) of the jets were changed.

Fig. 2 Single shot shadowgraphy-images for different nozzle design parameter acquired 450μs after visible start of injection;
Parameters: p_{amb}=0.1MPa, p_{fuel}=20MPa, T_{fuel}=350K, T_{amb}=350K, injection time: 1ms

To summarize, the penetration depth and the cone angle can be adjusted in combination with nozzle-hole diameter and impinging angle, which allows the adjustment of both, mass flow and spray shape. A bigger bore-hole diameter increases both, the spray cone angle and the penetration depth; a bigger impinging angle of the two liquid jets results in an increase of spray cone angle, but decreases the penetration depth.

In order to quantify the quality of the atomization process the drop size distribution was determined spatially and temporally with Phase-Doppler-Anemometry. In Table 1 the measured mean drop sizes for both injectors are shown for three different injection pressures for a fully opened injector. Here the small pressure dependency of the mean drop size of the twin-jet injector gets clear. Considering a pressure change from 20MPa to 5MPa, an increase of 16 percent (1.2μm) for the twin-jet injector is accompanied by an increase of 46 percent (4.5μm) for the multi-hole injector.

Tab. 1: Evaluation of the measured mean diameter during full needle lift of the injector in a distance of 30mm to nozzle tip

	Bosch HDEV 5.2 min. mean diameter [µm]	Twin-jet Injector min. mean diameter [µm]	Reduction
5 MPa	14.2	8.8	38.0 %
10 MPa	11.8	8.4	28.8 %
20 MPa	9.7	7.6	21.2 %

At full needle lift the measured mean drop size for the twin-jet injector compared to the multi-hole injector decreases drastically –depending on injection pressure– 21.2...38 percent. Furthermore, the level of 8.8μm mean droplet diameter of the twin-jet injector found for an injection pressure of 5MPa is not reached from the multi-hole injector even for a pressure of 20MPa.

* Corresponding author: markus.heldmann@ltt.uni-erlangen.de, stephan.wanner@handtmann.de

Characterization of the Mixture Formation Process in a GDI Engine Operating under Stratified Mode

L. Allocca*, M. Costa, A. Montanaro, P. Sementa, U. Sorge, B.M. Vaglieco
CNR - Istituto Motori, Naples, Italy
l.allocca@im.cnr.it, m.costa@im.cnr.it, a.montanaro@im.cnr.it,
p.sementa@im.cnr.it, u.sorge@im.cnr.it, b.m.vaglieco@im.cnr.it

Abstract

The paper is intended at an experimental and a numerical study of the mixture formation process in a GDI engine equipped with a high-pressure seven-hole injector. One of the 4 engine cylinders is modified to allow two optical accesses that leave unvaried the combustion chamber configuration, hence the performance and emissions of the real engine. A first optical access is through the piston head and a second one is through an endoscopic fiber probe inserted in the cylinder head. Under a medium-speed, medium-load working condition, the gasoline injection phase is experimentally characterized by image acquisition with high temporal resolution.

The considered injector is also preliminary experimentally characterized as delivering gasoline into an optically accessible confined vessel under various injection strategies. Spray images are collected to derive information relevant to penetration length and cone angle. The issuing mass flow rate is measured by means of a Bosch tube. The measurements data base is used to develop a 3D numerical model of the spray dynamics, whose validation is performed through an automatic procedure. The spray model is included within a 3D model of the in-cylinder processes to realize a reliable description of mixture formation and combustion.

---

* corresponding author: l.allocca@im.cnr.it

LES Study on the Poly-disperse Particle Deposition in the Human Upper Airway

X. G. Cui*, E. Gutheil
Interdisziplinäres Zentrum für Wissenschaftliches Rechnen
Ruprecht-Karls-Universität Heidelberg, Germany
xinguang.cui@iwr.uni-heidelberg.de and gutheil@iwr.uni-heidelberg.de

Abstract

Aerosol particle application of therapeutic agents into the deep lung represents an essential treatment of asthma and other lung diseases. The advantage of pulmonary drug delivery through inhalation is the topical treatment of specific lung conditions with limited whole-body effects. Aerosol particle deposition in this region has important impact in drug delivery efficiency, and it is highly desirable to study particle deposition in the human upper airway. Although significant investigations have been performed in this field, little research has focused on the study of poly-disperse particle deposition in the human respiratory system, considering a realistic drug dose. In the present study, a poly-disperse particle distribution as shown in Fig. 1 from a dry powder inhaler [1] is adopted using a realistic drug dose of 200 μg, which is introduced into the human upper respiratory system through the mouth. The mouth-throat configuration is constructed based on cast. Ansys ICEM-CFD 11.0 is used to generate the numerical grid. Two-way coupling is implemented to model the two phase flow. Large eddy simulation (LES) with the Smagorinsky sub-grid model is used to simulate the transitional laminar–turbulent gas flow, and the method is combined with a model for Lagrangian particle motion. The open source software of OpenFOAM 1.5 is adopted to solve the governing equations, where a new solver has been constructed to account for the particle motion using a Lagrangian tracking method within the LES formulation for the flow field.

The numerical results show that the particle deposition in the human mouth-throat is dominated by particle distribution and dispersion. The contribution of particle deposition is related to both the initial particle size and the geometric region. It is found that particles in the initial size range of 1–5 μm contribute least to the particle deposition as can be seen from Fig. 2, although the major part of injected mass consists of these particles, c.f. Fig. 1. The total mass fraction of particles in the size ranges 0.35–1 μm, 1–5 μm, and 5–23.5 μm is 26.3%, 63.5%, and 10.2% respectively, whereas the corresponding particle deposition efficiencies are 1.79%, 0.59%, and 3.99%. It is found that the particle deposition in the trachea is mainly caused by particles less than 1 μm, in the pharynx and larynx by particles larger than 5 μm, whereas in the mouth cavity, contributions of both particle size ranges deposit.

Thus, poly-disperse particle size distribution greatly influences particle deposition in the human upper airway, and particles in the size range of 1–5 μm are most likely to reach the deep lung. The present method is suitable to study particle deposition in more realistic mouth-throat models based on computed tomography (CT) scans.

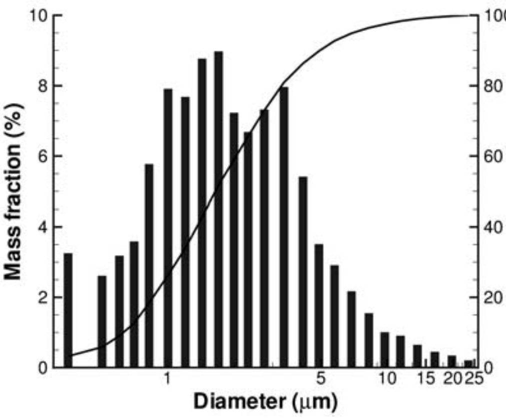


Figure 1. Initial poly-disperse particle size distribution measured from a metered-dose inhaler [1].

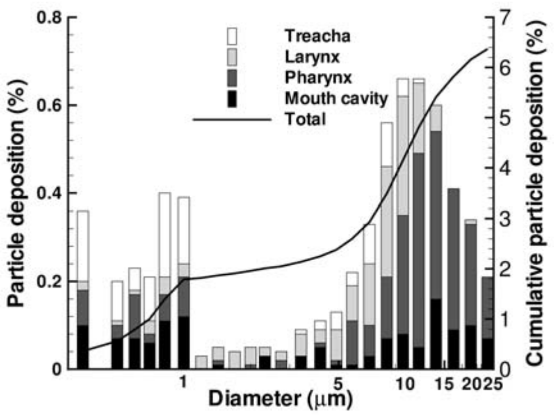


Figure 2. Particle deposition in different regions of the mouth-throat.

[1] Littringer, E. M., Mescher, A., Schroettner, H., Walzel, L. A. P. and Urbanetz, N. A., *European Journal of Pharmaceutical Sciences* (2011), accepted.

*Corresponding author: xinguang.cui@iwr.uni-heidelberg.de

Optimal Parameters for Pulmonary Particle Deposition as Function of Age

Renee Worden, Lisa Weber, Corinne S. Lengsfeld
[1]Department of Mechanical and Materials Engineering, University of Denver, Denver, Colorado, USA
Corinne.Lengsfeld@du.edu

Abstract

Background: As a result of dissimilarity in lung morphometry and physiological conditions, therapeutic aerosol particles deposit differently in humans of various ages and body weights. Recently, clinical findings observed low inter- and intra- patient reliability in dosing, suggesting that a single aerosol device and particle size distribution is ineffective. Methods: This work conducts a formal optimization study to determine the optimal particle size distribution for maximum drug delivery for a 2, 9 and 21 year old individual over a range of breath rates. The work also explores the optimal particle size distribution for the therapeutic aerosol to provide the same dose volume regardless of breath rate. This study utilizes classical, static statistical, probabilistic models in conjunction with lung cast data to calculate deposition and imparts population variation within normal distributions. Results: This study finds that both optimal particle size for maximum volume weighted deposition efficiencies are age/weight dependent as well as breath rate dependent. A monodisperse particle size is obtained even when a polydisperse spray is allowed for optimization when investigating a single condition (i.e., age or flow rate). The optimal particle size increases from 2 to 5 μm for toddler to adults, while the optimal size decreases from 5 to 3 μm for a 21 year old adult with a flow rate increasing from 13 to 60 L/min. Optimizing the particle size distribution for reliable dosing of a 21 year old adult was dependent on the minimum dose volume constraint applied. As the dose volume constraint increased from 0.1% to 5% a single peak at 0.5 micron was replace by a bimodal distribution and the mean size grew.

Protein Aggregation during Ultrasonic Nebulization

John A. Giarratano[1], John Carpenter[2], Corinne S. Lengsfeld[1]
[1]Department of Mechanical and Materials Engineering, University of Denver, Denver, Colorado, USA Corinne.Lengsfeld@du.edu
[2]Department of Pharmaceutical Biotechnology, University of Colorado, Denver, Colorado, USA

Abstract

Therapeutic proteins represent an essential piece of a health management plan for diseases such as diabetes, cancer, hemophilia and myocardial infarction. These proteins, however, must be maintained in their correct, biologically active conformation throughout processing, transportation, and delivery. This requirement poses serious engineering challenges because of a protiens susceptibility to thermodynamic instabilities resulting from the weak bonds driving the teriarty structure of the molecule. A particularly problematic type of protein degradation is aggregation. Administration of aggregated proteins, a particularly problematic degradation form, can have dire consequences, including blocking a patient's responsiveness to therapy, inducing immunogenicity, and even anaphylactic shock and death. Normal shipping and delivery methodologies are suspected of causing protein aggregation. This work investigates the effect of ultrasonic nebulization on protein aggregation as a function of impurity level, gas-liquid surface to value ratios, protein concentration, solution viscosity, and nebulization time. A 0.2M and pH of 4.2 Glycine buffer solution was utilized with IVIg protein at 0.5, 1, 5, and 10 mg/ml and 20ºC. Protein aggregates were characterized using Microflow imaging. Transient cavitation and formation of radicals was monitored using classical iodine assays. Higher protein aggregation is observed in solutions that initially contain greater amounts of impurities or have a larger contact area with the gas interface. Monitoring of the formation of I_{3-} from iodine as a function of nebulization time shows increasing production or radicals. All this supports the hypothesis that ultrasonic pressure waves in protein solutions cause transient cavitation which upon bubble implosion release hydroxyl radicals that can attact the protein in solution. Aggregate production does not continually increase with protein concentration, rather falls at higher concentrations. We have demonstrated this increase in viscosity inhibits cavitation by elevating the lowest pressure region based on a specified pressure drop.

Comparison of numerical simulation to experiments for a jet nebulizer

N. Lelong * (1), L. Vecellio (1), A. Junqua-Moullet (2), Y. Sommer de Gelicourt (3),
C. Tanguy (4), P. Diot (5), H. Suzanne (2)
(1) – DTF Aerodrug, Faculté de Médecine, Tours, France
(2) – CEA Le Ripault, Monts, France
(3) – ANSYS France, Montigny le Bretonneux, France
(4) – Dantec Dynamics, Nozay, France
(5) – INSERM U1100 – EA 6305, Faculté de Médecine, Tours, France
nicolas.lelong@etu.univ-tours.fr, vecellio@med.univ-tours.fr,
alexandra.junquamoullet@cea.fr, yann.sommerdegelicourt@ansys.com,
christian.tanguy@dantecdynamics.com, diot@med.univ-tours.fr, herve.suzanne@cea.fr

Abstract

The performances of jet nebulizers for medical purposes are limited compared to the new mesh nebulizers, regarding output rate and drug loss. Their experimental development has reached a plateau, however the improvement of the device, cheaper and able to produce smaller droplets, stays an important challenge. The purpose of this study is to design a numerical model simulating the nebulization process and to compare it with experimental data obtained from various measurement methods. Such a model allows a better understanding of the atomization process and a determination of the relevant physical parameters influencing the nebulizer output.

The Updraft nebulizer (Hudson) was chosen to set geometric dimensions of the model and was designed with ANSYS Workbench. The simulation domain covers a wide area with a 4 µm mesh in the center of the device, where the liquid atomization takes place. Two inlet air flow rate were considered: 2 L/min (Re = 4000) and 8 L/min (Re = 16000). Boundary conditions were set with experimental data and the relevant model parameters were determined through a 2D axial simulation. Large Eddy Simulation has been used as the turbulence model to account for the wide range of Reynolds regimes in the process. The transient 3D CFD calculations were then run on a 15° angular sector representing 4 million cells with ANSYS Fluent. Droplet formation times and locations, along with diameters and airflow velocity, were determined thanks to user-defined functions in Fluent. The experimental study consists in characterizing the spray output. Particle size measurements were performed with a laser diffractometer Spraytec (Malvern), particle size and velocity were measured with a Phase Doppler Anemometer (Dantec Dynamics) and aerosol output was determined by a gravimetric method. The behavior of the droplets was visualized with a Fastcam SA1 CCD Camera (Photron).

Observations made with CCD camera showed similar patterns as numerical results. At 2 L/min the liquid periodically formed a film which blew up in drops with various sizes. The larger ones impinged and spread out on the sphere while the faster ones rebounded and split in smaller particles. At 8 L/min, images showed a continuous production of small droplets and the progressive formation of a liquid ring on the sphere surface. For both flow rates, the simulation and camera particle size distributions were similar, especially at 8 L/min. The difference between the mean diameters was about 20 % (26 µm vs. 21 µm; 17 µm vs. 14 µm), which is acceptable given mesh size and image resolution. PDA and laser diffractometry, which take into account spherical droplets with a diameter less than 5 µm, provided smaller values.

CFD predicted atomization phenomena and provided a good estimation for droplet size in the nebulizer compared to CCD camera images. By showing the dynamics in the liquid phase during the fragmentation step, CFD helps understanding the physical processes governing the fragmentation of droplets over 5 µm, which could be extrapolated to micrometric particles obeying to the same dynamics. This model could help predicting nebulizer output with defined geometrical and physical parameters.

---

* Corresponding author: nicolas.lelong@etu.univ-tours.fr

Quantitative three-dimensional imaging using computed tomography and structured illumination

E. Kristensson*, E. Berrocal, and M. Aldén
Division of Combustion Physics, Lund University, Sweden

Abstract

An imaging technique capable of measuring the extinction coefficient in three dimensions is presented and demonstrated on two different atomizing spray systems with promising results. The approach is able to suppress unwanted effects due to both multiple scattering and light extinction, which, for spray imaging, seriously hampers the performance of conventional imaging techniques. The main concept consists in illuminating the sample of interest with a light source that is spatially modulated and to measure the transmission in two dimensions at several viewing angles. This, in turn, allows the local extinction coefficient to be calculated in three dimensions by means of standard computed tomography algorithms, in this case filtered-back projection. These algorithms are based on the Beer-Lambert law and it is therefore essential to detect only the unperturbed (ballistic) light. All extraneous light must be suppressed, hence the use of structured illumination. To improve the filtering capabilities even further, a novel "crossed" structured illumination approach is tested and implemented. With this approach the sample is illuminated with a light source that is spatially modulated both in the vertical and horizontal direction. In the paper, the accuracy and limitation of this new method is first evaluated by probing several homogeneous milk solutions at various levels of turbidity, where the opacity is altered through dilution. The presented approach shows good agreement with theory up to an estimated optical depth of ∼5, whereas conventional transmission imaging shows discrepancies already at $OD > 1$. Interestingly, this value represents the limit of the single scattering regime ($OD < 1$). In the single scattering regime, most of the existing laser-based diagnostics are unaffected by errors introduced by multiple light scattering. The results presented in this paper demonstrate the potential of exceeding this limit for transmission imaging by the implementation of structured illumination. To test the method two different optically dense aerated spray systems are probed; one transient GDI nozzle injected with iso-octane and one quasi-steady state multi-hole nozzle injected with water. In order to temporally freeze the transient GDI spray pulsed lasers (Nd:YAG) are implemented. The resulting 3D renditions show almost no sign of image artefacts that may occur due to errors in the reconstruction process or if the number of viewing angles are too few. In addition, any skewness in the spatial distribution of the droplets, ordinarily caused by extinction, is avoided with the presented method. The performance of the filtered-back projection algorithm is evaluated by comparing the experimentally measured optical depth with the opacity calculated from the reconstructed 3D result. Due to the limited dynamic range of the camera (14-bit), the technique is limited to regions within the sprays where the optical depth is reduced below ∼6. However, the method is compatible with other filtering approaches, such as temporal-, spatial- and polarization filtering (as is utilized for Ballistic imaging), which potentially could increase the range of applicability even further. Perhaps one of the main benefits with the presented approach concerns its relatively low experimental cost, the technique can for instance be used to study non-reacting, optically dense sprays running in steady-state operation using a simple cw laser combined with an inexpensive non-gated camera.

---

*Corresponding author: elias.kristensson@forbrf.lth.se

Correlating Results from Numerical Simulation to SLIPI-based Measurements for a non-combusting Diesel Spray

H. Grosshans* [1], E. Berrocal [2], E. Kristensson [2], Robert-Zoltán Szász [1] and Laszlo Fuchs [1]
1: Division of Fluid Mechanics, Lund University, Sweden
2: Division of Combustion Physics, Lund University, Sweden

Abstract

The validation of numerical methods for the study of highly atomizing transient sprays (e.g. Diesel sprays) is a nontrivial task. One important issue concerns the fact that such sprays are optically dense making the measurement particularly difficult, especially when high accuracy is required (which is the case for model validation purposes). It has recently been demonstrated that SLIPI-based (Structured Laser Illumination Planar Imaging) techniques are able to provide either two or three-dimensional quantitative results of the extinction coefficient even in challenging situations. More specifically, the extinction coefficient of a non-combusting Diesel sprays injected at 1100 bar in a chamber pressurized at 18.6 bar could be extracted in two-dimension at late time (2000 μs) after injection start. In this article, these experimental results are correlated with numerical data. The numerical calculations are based on Large-Eddy Simulation (LES) combined with Lagrangian Particle Tracking (LPT). The simulation includes secondary droplet break-up models, gas-liquid two-way coupling and an evaporation model. The main purpose of the work presented here is, then, to numerically deduce the extinction coefficient field which can can be directly compared to the one obtained experimentally. The extinction coefficient is related to the droplet number density times the extinction cross-section which is calculated, for each droplet size, based on the Lorenz-Mie theory. We show, that by extracting the extinction coefficient from numerical simulation, comparisons between simulated and SLIPI-based experimental results are becoming possible even in optically dense sprays. Thus, the presented approach is a step towards in providing strong evidence of similar or different structures between the experimental and simulated description of atomizing spray systems. The experimental results used in this paper have been published in [1]. The extinction coefficient field obtained numerically is shown for three instances in time in Fig. 1, where $t = 0$ corresponds to the beginning of injection of the liquid spray. The shown planes are also cuts through the spray including the injection nozzle, which is located at the bottom of the figures.

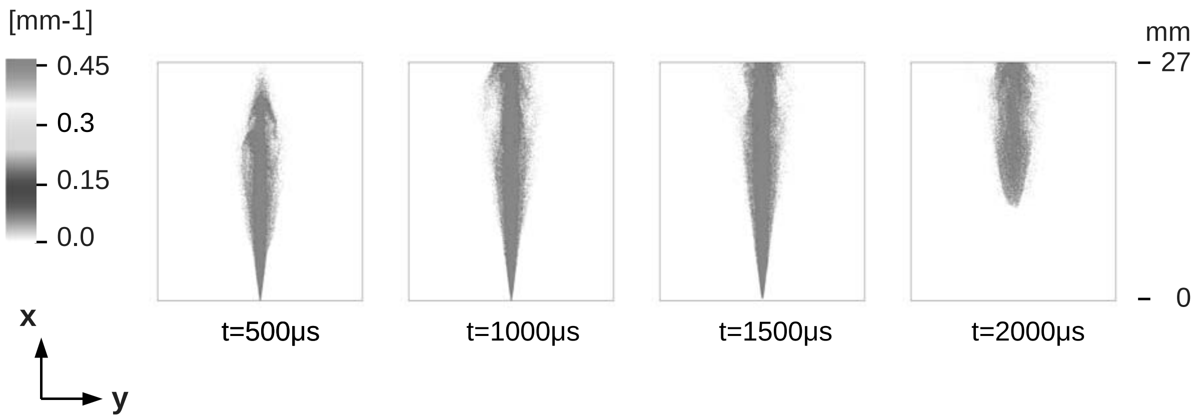


Figure 1. Extinction coefficient fields numerically simulated for $t = 500\ \mu s, 1000\ \mu s, 1500\ \mu s$ and $2000\ \mu s$ (from left to right).

References

[1] E. Berrocal, E. Kristensson, P. Hottenbach, M. Alden, and G. Gruenefeld. Quantitative imaging of a non-combusting diesel spray using structured laser illumination planar imaging. *Appl. Phys. B*, to be published.

*Corresponding author: holger.grosshans@energy.lth.se

Development of future spray imaging techniques using General Purpose GPU accelerated Monte Carlo simulation

E. Berrocal* and J. Jönsson
Combustion Physics, Lund University, Sweden
edouard.berrocal@forbrf.lth.se

Abstract

The past decade has seen the development and application of emerging laser techniques for spray imaging. Two noticeable examples are Ballistic Imaging (BI) and Structured Laser Illumination Planar Imaging (SLIPI). The main motivation in developing such novel techniques was to filter out the blurring effects introduced by multiple light scattering in order to obtain reliable two-dimensional qualitative and quantitative spray information. In parallel to this experimental development, Monte Carlo (MC) simulation of light propagation and scattering through spray systems has also been initiated. While the MC simulation is a powerful and versatile tool for modeling various spray geometries and detection schemes on a single computer, its main drawback remains its long computational time. However, since 2007 a programming approach, named Compute Unified Device Architecture (CUDA), has been created for performing general purpose calculations on Graphics Processing Unit (GPU), as a data-parallel computing device. Thanks to the continuously increased number of cores in combination with larger memory bandwidth, recent GPUs offer considerable extended resources for general purpose computing. In this article, we describe an accelerated version of a validated MC model (originally presented at ICLASS-2006), for the simulation of laser light propagation in sprays. The code is now capable of running the calculations on a modern GPU card, showing a ~100x increase in simulation speed compared to the original version of the code. Thanks to these new possibilities, the MC model presented in this article allows detailed performance analysis of various laser imaging techniques. This is demonstrated for BI where a time-gating approach is used and for SLIPI where a modulation-based filtering is employed in the spatial domain.

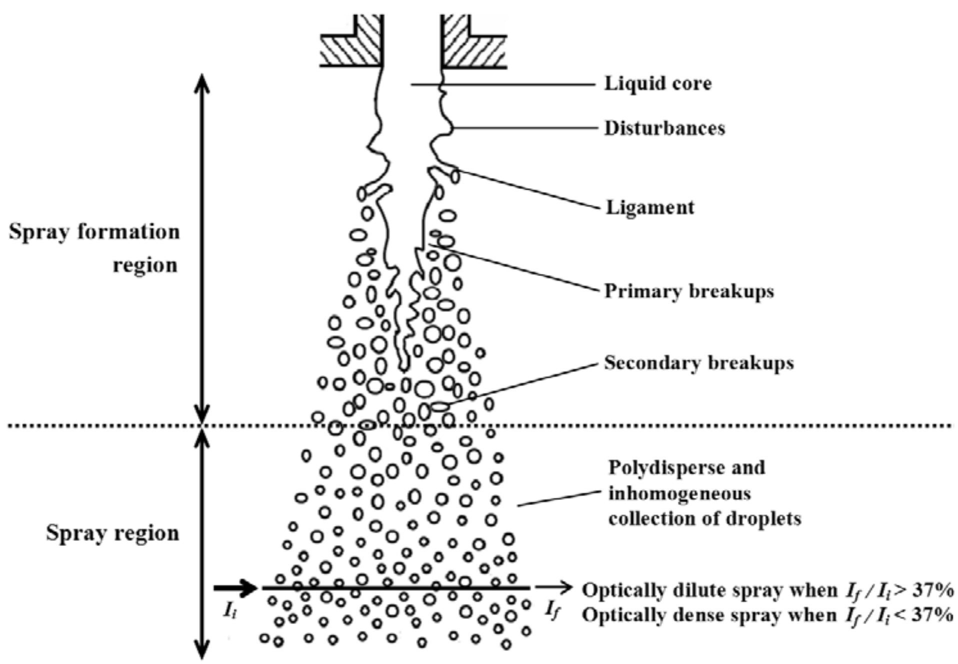


Figure: Illustration of the structure of an atomizing spray. The spray region can either be optically dilute or optically dense depending on the droplets number density, the droplets size and the illumination wavelength. As indicated in the figure a measure of light transmission I_f/I_i allows defining if the spray is optically dense or dilute for a given wavelength. The GPU-MC model presented here allows the simulation of a laser beam propagating through the spray region.

* Corresponding author: edouard.berrocal@forbrf.lth.se

Spray resulting from High Pressure Atomization with low L over D Multihole Injectors and the role of the Cavitation

S. Makhlouf*[1], J. Hélie[1], O. Grimoux[1], J. Cousin[2], L. Gestri[3], A. Wood[4], G. Wigley[4].
samir.makhlouf@continental-corporation.com
[1]Continental Automotive SAS, Toulouse, France
[2]CORIA laboratory, Rouen, France
[3]Continental Automotive, Pisa, Italy
[4]Loughborough University, England.

Abstract

Two different low L/D nozzle designs are tested: one version with a sharp entrance (normal injector) and a second version submitted to hydro-grinding to smooth out the nozzle entrance rounding radius in order to counteract the cavitation trend. The resulting spray is analyzed in a close view by direct imaging shadowgraphy and by PDA (Phase Doppler Anemometry).

The comparisons presented, due to the reduction in cavitation on one side of the nozzle in the hydro-ground injector, show that the overall spray morphology is different in comparison to the normal injector. Asymmetric cavitation causes the spray, especially its dense region, to tend towards the injector axis. As the cavitation at nozzle exit is reduced, the angle of the dense spray region is increased and the spray width is reduced.

Having cavitation on one side of the nozzle changes the velocity field of the flow, hence the jet experiences a transverse motion. Further analysis shows that at high pressures the mixing of hydro-ground injector's spray takes place further downstream than for the normal injector. PDA results also do agree with these findings, showing increased droplet axial velocity for the hydro-ground injector, and also a reduction in the range of droplet flow angles, which results from the reduced levels of cavitation. Therefore, cavitation inside the nozzle improves the dispersion of the spray because as soon as the bubbles emerge from the nozzle they break the liquid structures and increase the injection angle. Droplet sizing were applied and surprisingly both sprays lead to close droplet diameters; which means that the result of the overall atomization process is not dramatically influenced by cavitation. These features behaviours are discussed in the paper by analyzing the resulting sprays from the two injectors.

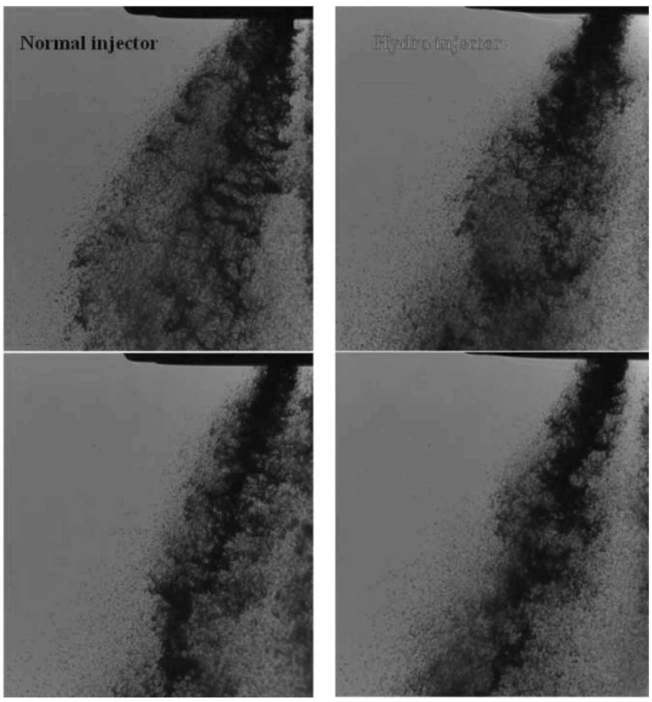


Figure 1: Samples of jet images at 60bar.

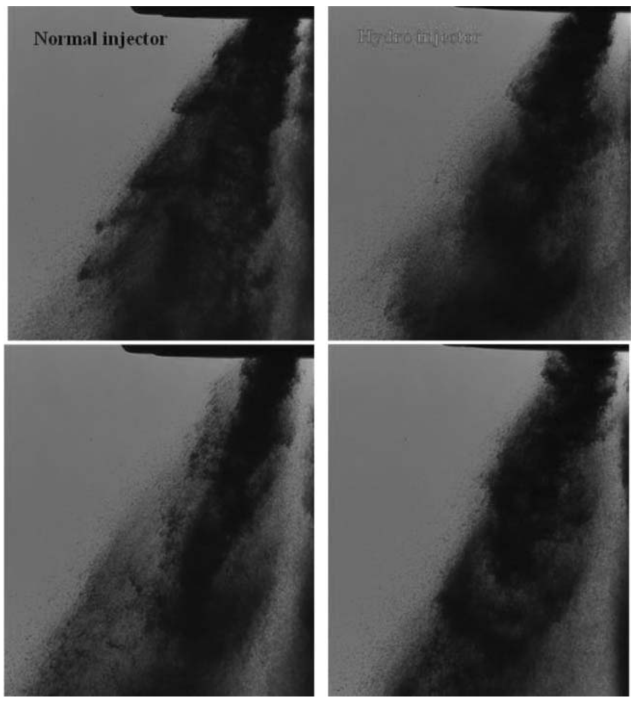


Figure 2: Samples of jet images at 130bar.

* Corresponding author: samir.makhlouf@continental-corporation.com

Thursday

September 6, 2012

Contents

New Developments I

Spray Impact III

Cavitation

Spray Impact IV

Exploration of Aerated-Liquid Jets Using X-Ray Phase Contrast Imaging and X-Ray Radiography

Kuo-Cheng Lin[1*], Campbell Carter[2], Alan Kastengren[3] and Kamel Fezzaa[3]
[1]Taitech, Inc., Beavercreek, Ohio 45430, USA
[2]Air Force Research Laboratory, Wright-Patterson AFB, Ohio 45433, USA
[3]Argonne National Laboratory, Chicago, IL 60439, USA

Abstract

Liquid jet atomization plays an important role in establishing stable and efficient combustion inside the combustor of a liquid-fueled air-breathing propulsion system. For applications requiring both deep fuel penetration into high-speed crossflows for broader fuel spreading and smaller droplets in the liquid spray for faster evaporation, a superior liquid injection scheme is sought. Among the possible candidates, aerated-liquid (or effervescent, or barbotage) jets have been explored extensively. It has been shown that the liquid aeration technique can generate a spray that penetrates well into the flow and produces a large fuel plume containing a large number of small droplets. The required amount of aerating gas and delivery pressure are practically obtainable in a high-speed air-breathing propulsion system. The utilization of aerated-liquid jets has led to successful combustion in a liquid-fueled high-speed air-breathing combustor. While macroscopic and far-field features of the aerated-liquid jets have been extensively examined, detailed near-field spray structures cannot easily be explored.

In the present study, two X-ray diagnostics, one is the X-ray phase contrast imaging (PCI) technique and the other one is the X-ray radiography, available at the Advanced Photon Source (APS) of the Argonne National Laboratory, were successfully utilized to characterize the dense near field of aerated-liquid jets injected into a quiescent environment. Water and nitrogen were used as the injectant and aerating gas, respectively. An axisymmetric aerated-liquid injector equipped with an exit adaptor was utilized for the investigation of external spray structures. A total of three adaptors with various internal configurations and a throat diameter of 1.0 mm were selected for testing. The major motivation of this study was to obtain a better understanding of the near-field structures of aerated-liquid jets discharged from the contoured adaptors.

It was found that the measurements from the X-ray PCI technique give both a qualitative understanding of microscopic structures, such the existence of small droplets, ligaments, and even bubbles, and also quantitative size distributions of the disintegrated small objects within the peripheries of aerated-liquid jets. The phase contrast images were captured with a CCD camera by converting the transmitted X-rays into visible light via a fast scintillator crystal. Each image has a field of view of 1.5×1.9 mm^2. The actual exposure time, determined by the X-ray pulse, is 150 ps (FWHM). Size measurement of microscopic structures was achieved, using an in-house code to remove the undesirable background noise with an enhanced contrast ratio for each X-ray image and then to identify objects of interest, such as droplets or gas bubbles, for measurement. The measured Sauter mean diameter (SMD), bubble SMD and bubble film thickness are on the order of 20, 40 and 10 μm, respectively in the present study. This technique, however, cannot depict either the spray structure within the core region of the spray or the liquid mass distribution within the plume, due to the line-of-sight feature of the X-ray PCI.

The X-ray radiography was utilized by the present study to supplement X-ray PCI technique. Notably, X-ray radiography provides quantitative liquid mass distribution profiles or even contours within the dense near-field region of an aerated-liquid jet. This was achieved by converting the measured X-ray extinction to the equivalent path length (EPL) via the Beer's law. The EPL is the thickness of pure water required for the transmitting X-ray to generate the same amount of extinction as that generated from the dispersed spray at the same X-ray energy level. The value of the EPL can, therefore, be related to the local density of the liquid/air mixture or local liquid mass fraction. For the present study, only the two-dimensional line-of-sight EPL is presented to depict the spray structure. The focused X-ray beam is approximately 5 (V) × 6 (H) μm FWHM. The utilization of X-ray radiography also helps to illuminate the two-phase structures inside the injector and within the near field of the spray plume. For the present injector design and operating conditions, it was found that an annular-like liquid distribution exists inside the injector and within the discharged near-field jet at low liquid flow rates, even at high aeration levels. The present study demonstrates that a combination of both X-ray diagnostics provides valuable insights into the understanding of aerated-liquid jets or potentially other optically dense sprays.

* Corresponding author: Kuo-Cheng.Lin.ctr@wpafb.af.mil

The effect of initial fuel temperature on vaporization in aeroengine combustors with prevaporization

Christoph Hassa, Pierre-Fréderic Wiesmath
German Aerospace Center
51170 Cologne, Germany

In gas turbines, the temperature of liquid fuels influences atomization and evaporation and thus the position of the flame front which in turn influences almost every aspect of the combustion. Nevertheless little is known about the actual fuel temperature at the place of initial contact with the air. The paper describes an experimental investigation of atomization and evaporation of Kerosene sprays emitted from a jet in cross flow at operating conditions similar to an aero engine cruise and a variation of Kerosene temperature between 40 and 150°C. Phase Doppler Anemometry was used to measure drop size and velocity and to infer vaporization rates of the spray. A noticeable influence of the Kerosene temperature was measured. For the isothermal atomization with identical air and liquid temperatures and a variation between 40 and 150 °C, the Sauter diameter drops from 20 to 16 µm. For a residence time of 0.5 ms at 750 K air temperature, 9 bar air pressure and 120 m/s air velocity, the fuel temperature difference of 110 °C is responsible for 26 % higher vaporization. Hence the initial fuel temperature can affect combustion stability and efficiency at critical conditions, which makes it attractive to investigate the effect in realistic premixing combustors.

The goal of this contribution is to provide information on the magnitude of the effect of fuel preheat in gas turbine combustors with partial prevaporization and premixing. Due to the high Weber number of the atomization process and the complexity of the fuel, experimental information is needed. Since the magnitude of the effect is closely coupled to droplet size, realistic operating conditions with respect to the parameters influencing atomization are required. The extent to which this is also the case for other parameters influencing evaporation, namely temperature and fuel was one of the questions to be answered by this study and hence the influence had to be reproduced in the experiment. The resulting environment poses challenges to the use of optical techniques, the number density of droplets of the resulting fine spray for the application of PDA and the ill-definedness of Kerosene for the application of the vapor absorption techniques. Nevertheless, the experimental techniques were clearly able to identify the trends in a meaningful manner. To the author's knowledge, this is the first study to expose them at realistic operating conditions, specifically at a typical aero engine cruise condition.

With respect to the influence of fuel preheat on drop size, the capability of the vaporizing duct to independently vary air pressure as well as fuel and air temperature enabled to investigate the influence of temperature isolated from density and heat transfer effects. It could be concluded, that for Kerosene with the experimental uncertainties connected to drop size correlations, the influence is indeed sufficiently described by surface tension. However, the variation of fuel temperature and constant high and low air temperature shows that the real question is how to define that surface tension and no easy answer is at hand. Instead it has to be concluded, that heat transfer from the hot air to the liquid prior to and during atomization always influences atomization in a way not to be neglected as demonstrated by the large difference of the drop sizes for high and low air preheat at constant air density and velocity, which surpasses the influence of the fuel preheat. Obviously this has implications for the detailed modeling of atomization processes.

The influence of the 110 °C fuel preheat on drop size at high air temperature was smaller than the difference effected by the air velocity variation from 94 to 120 m/s. However the reverse is observed for the influence on evaporated mass at the measurement section of 60 mm, where the fuel preheat led to 26 % more evaporated fuel. This is obviously due to the reduced enthalpy needed for evaporation and is a sizeable effect compared with the difference in pressure loss needed to accelerate from the lower to the higher velocity. The higher vaporization rate will influence the flame position in real combustors and will be especially important in situations when flame stability or combustion efficiency is marginal.

The enrichment of fuel vapor observed in the wake of the jet for the lower penetration case is certainly tied to the occurrence of special conditions which are not always present, but it shows that the fuel preheat can give rise to a behavior which could influence flame stabilization in a nonlinear way as in this case where especially the amount of vapor produced early in the vaporization process is influenced. This is not necessarily limited to shear breakup of jets in cross flow, as an analog situation occurs for the atomization from the wave crests of prefilming atomizers.

Generally the results show that the effects of increased initial fuel temperature are big enough to justify an investigation in a more realistic environment with combustion where the changes in combustion behavior and performance could be studied.

Spray Characterization of Gas-to-Liquid Synthetic Aviation Fuels

Kumaran Kannaiyan, Reza Sadr*
Department of Mechanical Engineering, Texas A&M University at Qatar, Qatar
kumaran.kannaiyan@qatar.tamu.edu and reza.sadr@qatar.tamu.edu

Abstract

Development of alternative aviation (jet) fuels is gaining importance in recent years to meet growing energy demand of the world and to reduce the environmental impact of aviation fuel combustion. Alternative fuels need to match the energy density as that of the conventional fuels and need to possess vital qualities such as rapid atomization and vaporization, quick re-ignition at high altitude and acceptable emission level. Gas-to-Liquid (GtL) synthetic paraffinic kerosene obtained from Fischer-Tropsch synthesis has grabbed global attention for its cleaner combustion characteristics due to the near absence of aromatics and Sulphur content in the fuel composition. Characterizing the atomization of the alternative liquid fuels is necessary as it affects evaporation process and mixing with air which, in turn affect combustion and emission characteristics of the fuel. As a part of an ongoing joint research effort between Texas A&M University at Qatar (TAMUQ), Rolls-Royce (UK), and German Aerospace Laboratory (DLR), an experimental facility is built at TAMUQ to study the spray characteristics of different GtL blends.

The main objective of this work is to investigate the influence of the change in fuel composition on the spray characteristics at different injection pressures using a pressure (simplex) nozzle. The GtL blends used in this work consist of varying degree of cyclic carbon content and iso-to-normal paraffin ratio that fit in between the commercial GtL kerosene and the commercial paraffinic solvents. A planar optical diagnostic technique, Global Sizing Velocimetry is used in this work to study the spray characteristics, such as droplet size, distribution and velocity, of three different GtL blends and the results are compared with that of the conventional Jet A-1 fuel. The droplet size distributions highlight the influence of fuel composition on the mean droplet size, distribution and spray structure among different blends.

* Corresponding author: reza.sadr@qatar.tamu.edu

Towards fuel spray ignition in aircraft engine

A. Bruyat*, G. Linassier*, P. Villedieu*, N. Bertier**, O. Rouzaud*, G. Lavergne*
*ONERA-The French Aerospace Lab, F31055 Toulouse
**ONERA-The French Aerospace Lab, F92322 Châtillon
anne.bruyat@onera.fr

Abstract

In order to certify turbojet combustors, motorists need to ensure security criteria. One of them is the possibility to relight the combustor in case of in-flight extinction were the conditions are critical (low temperature and low pressure).
The most reliable and common ignition mode of air-breathing engines is electrical spark discharges. Heat deposition leads to the formation of an ignition kernel : the spark discharge deposites energy, and spherical flame expands in a small-scale way. The kernel then propagates in two different length scales : the kernel can first be transported into a recirculation zone and, finally, the whole chamber is ignited.
The certification of new combustors needs numerous and expensive ignition tests, for a wide range of conditions. For the preliminary phase, and in order to improve the design of industrial combustors, numerical tools are used. This is why accurate numerical models are needed to correctly reproduce the two-phase flow inside the combustion chambers before, during, and after the ignition process.

An ignition kernel model [1] has been developed and combined with a CFD code in two ways. Firstly, the ignition model is used to build the ignition probability map of a combustor. Secondly, a local ignition simulation is introduced as initial condition in an unsteady simulation to model the complete flame propagation.
These models are applied to numerical simulations for a mono-sector combustor, experimentally investigated on the MERCATO test-rig (ONERA-Mauzac). Velocity fields of the two-phase flow and the droplet size fields are provided by LDA and PDA measurements [2][3].

The unstructured reactive compressible code CEDRE is used, in which a lagrangian approach is developped to model spray [4][5]. The RANS and LES simulations are compared to the experimental measurements, and validated. More, the precessing vortex core, an hydrodynamic instability of the gas flow, is correctly captured by LES. From the two-phase mean flow field, the ignition model is first used to get an ignition probability map. Secondly, the local ignition simulation model is applied to instantaneous LES flow fields in order to get the ignition statistics. Results are then compared to the experimental statistics of the combustor for given localisation of the spark, and given operating conditions.

References:

[1] Garca Rosa, N., *Phenomenes d'allumage d'un foyer de turbomachine en conditions de haute altitude,* Universite de Toulouse, PhD Thesis.
[2] Lecourt, R., Linassier, G., Lavergne, G., Proceedings of ASME Turbo Expo 2010, Vancouver, Canada, June 2011.
[3] Lang, A., Lecourt, R., and Guliani., F., Proceedings of ASME Turbo Expo 2010, Glasgow, June 2010.
[4] Garca Rosa, N., Linassier, G., Lecourt, R., Villedieu, P., Lavergne, G., Heat Transfer Engineering, Xi'an, 2009.
[5] Linassier, G., Lecourt, R., Villedieu, P., Lavergne, G., and Verdier, H., 23rd European Conference on Liquid Atomization

Fuel Jet in Cross Flow: Experimental Study of Spray Trajectories at Elevated Pressures and Temperatures

E. Lubarsky*, D. Shcherbik, O. Bibik, J. Bennewitz and B. T. Zinn

School of Aerospace Engineering
Georgia Institute of Technology
Atlanta, Georgia 30332-0150 USA
el45@mail.gatech.edu

N. Patel and M. Benjamin

GE Aviation
General Electric Company
Cincinnati, Ohio 45215 USA

Abstract

This paper describes an experimental investigation of the spray created by Jet-A fuel injection into the cross flow of air at elevated pressures and temperatures that are encountered in modern gas turbines. Fuel was injected from a 0.671 mm diam. orifice with flow coefficient C_D=0.683. The orifice was incorporated into the wall of a rectangular air channel (25.4×31.75 mm). In some experiments the orifice was installed flush with the wall of the channel and in others in a cavity recessed in the wall. The pressure of air in the channel was P~200KPa and temperature was *T*=590K and *T*=700K. The aerodynamically shaped design of the air channel created a uniform trapeze-shape velocity profile with turbulence level in the core of about ~2%, while thickness of the boundary layer was ~3mm. The momentum flux ratio of the fuel jet to the crossing air was kept in a range between *J*=5 to *J*=40. Shadowgraph spray images were captured using a high speed camera at a rate of 24,000fps. The length of the record was typically 8000 frames. Exposure time was minimized by using short flashes (30ns) of a copper-vapor laser synchronized with the camera shatter. This methodology allowed obtaining statistically relevant maximum, mean and fluctuating characteristics of spray penetration into the cross-flow. Two image sizes were captured: a general view image that covered 22.5mm×50.5mm and a zoomed view of 10.7mm×10.6mm. This corresponds to the downstream distances from the orifice *z/d*~70 and *z/d*~15, respectively. Simple correlations for spray trajectories were obtained using only two empirical coefficients for each flow condition or injector mounting. Spray penetration was found to be proportional to the square root of momentum flux ratio and independent of the Weber (*We*) number in the range investigated (*We*=400….800). The spray shape was well determined with the logarithmic function. More sophisticated correlations with up to four empirical constants were attempted. These included use of momentum flux ratio in powers that differs from ½, dependence of penetration on *We*, and use of power functions instead of logarithmic functions to determine spray trajectory. All these attempts did not improve the correlations significantly.

* Corresponding author: el45@mail.gatech.edu

Modelling of Single Droplet Drying and Morphology Evolution using Meshfree Simulation Methods

W. Säckel*, F. Keller, U. Nieken
Institute of Chemical Process Engineering, University of Stuttgart, Germany
saeckel@icvt.uni-stuttgart.de and nieken@icvt.uni-stuttgart.de

Abstract

Spray drying is a widely used industrial process. However, in many cases process parameters need to be evaluated empirically in order to achieve the desired product properties. Common drying models of single droplets typically picture a spherical symmetric geometry and thus model radial distributions of a quantity. These are calculated according to effective transport parameters, which need to be fitted to experiments. The substructure and its morphology and properties inducing the transport behaviour cannot be predicted. However a more detailed insight into the drying process is very desirable.

The present contribution introduces a novel approach for single droplet drying models based on the meshfree method Smoothed Particle Hydrodynamics (SPH). In this technique the continuum is approximated using interpolation points, so called particles, which can move independently from each other according to a Lagrangian viewpoint. Due to the particle-based point of view meshfree methods are capable of handling phenomena occurring in morphology evolution like large material deformations and fractures.

In a first model the Smoothed Particle Hydrodynamics approach is validated in comparison to a common grid-based model using finite differences. The implementation of Nusselt and Sherwood boundary conditions requires more effort in the SPH model, when surfaces of arbitrary shapes need to be regarded. The conversion of area based fluxes into volume/mass based ones can be undertaken using the CSF approach. In a next step the model was enhanced to fluid dynamics incorporating the motion of an incompressible Newtonian liquid using a predictor-corrector projection method. Solids are modelled as a different particle/point class using a rigid body approach. Surface tension and wetting are implemented using forces between interpolation points. By applying different forces between different phases a wetting behaviour can be parameterised. Using this technique the first drying period regarding a suspension can be modelled in principle. In order to simulate the second drying period, where transport through a porous crust is taking place, diffusive vapour transport through the gas phase was implemented and tested for a porous structure containing fixed solids, a liquid and a gas phase.

Numerical results showed that the SPH approach is able to achieve the same results as grid based methods, when Nusselt and Sherwood boundaries are applied and radial distributions are calculated. Moreover the geometries can be arbitrarily shaped due to the CSF approach. Simulations for the drying of a suspension have been undertaken in a two-dimensional study. A variation of physical properties, the wetting behaviour and the size of solid particles, leads to different morphologies inside the droplet. Subsequently simulations of diffusive vapour transport and the receding liquid interface inside a fixed, porous structure were undertaken. The approach is capable of modelling the second drying period, too, so that a future drying model of a droplet can regard both drying periods on a detailed length scale.

The approach shows reasonable results and underlines the methods ability to study morphology evolution. However, the method is relatively new for this application, thus requiring further research. In the near future studies shall be undertaken using a finer resolution. A future perspective lies in multi-scale simulations, which may combine the potential of detailed approaches like the one presented here with the computational efficiency of coarser approximations.

* Corresponding author: saeckel@icvt.uni-stuttgart.de

Production of fat-based emulsion powder by prilling process using twin-fluid atomizer for controlled release of iron

Bipro N. Dubey*, Erich J. Windhab
Lab. of Food Process Engineering, IFNH, D-HEST, ETH Zurich, Switzerland
bipro.dubey@hest.ethz.ch and erich.windhab@hest.ethz.ch

Abstract

Encapsulation of iron is necessary to supply bioavailable iron to large number of population possess iron deficiency. In the present study, we dispersed the iron solution in a fat matrix of palm stearin, and prepared the simple emulsion (water-in-oil) at $60\,^{\circ}\mathrm{C}$, where fat was a continuous phase. Using that emulsion, we produced fat based emulsion particles through prilling (spray + chilling) process using twin fluid atomizers (internal mixing). We characterized the particle in terms of size and size distribution, and investigated the internal structure of the fat-particles by cryogenic scanning electron microscopy (cryo-SEM) for observing the distribution or homogeneity of dispersed phase. Present study includes mainly the iron release kinetics through the fat matrix of the emulsion particle in an in-vitro gastric system ($pH \approx 2.0$) as a function of (a) particle size of prills, (b) thickener concentration (polyethylene glycol, PEG) in dispersed phase, (c) droplet size of dispersed phase, (d) mixing properties (Reynolds number, *Re*), and (e) shelf-life of particles. The release kinetics was explained by the second order kinetics, where we estimated the release kinetic constant, and co-related with the viscosity ratio of dispersed phase to continuous phase, mean particle size of emulsion, and shelf-life of particles. The result showed that the control of the release properties can be obtained by choosing particle size and thickener concentration.

---

*Corresponding author: bipro.dubey@hest.ethz.ch

Spray drying behaviour and functionality of beta-lactoglobulin-/pectin-stabilized o/w-emulsions

Y. Serfert[1*], J. Laackmann[2], A. Mescher[3], J. Schröder[4], S. Drusch[1], K. Schwarz[5]

[1]Technical University of Berlin, Berlin/Germany

[2]University of Hamburg, Hamburg/Germany

[3]University of Dortmund, Dortmund/Germany

[4]Karlsruhe Institute of Technology (KIT), Karlsruhe/Germany

[5]University of Kiel, Kiel/Germany

Abstract

During microencapsulation of sensitive (food) ingredients by spray drying, one important step is the spray drying process itself, which affects the emulsion droplet size distribution, the emulsion stability with regard to the oil droplet size as well as the structural integrity of the interface. The latter has a strong impact on the encapsulation efficiency and the physical stability of reconstituted emulsions. By interfacial engineering the barrier properties and the packing density at the oil/water (o/w) interface can be modified. The interface can be engineered by e.g. layer-by-layer-technology that involves sequential layering of polymers at the o/w-interface of an emulsion via electrostatic interactions. By that means so-called multilayer emulsions can be obtained. Studies on the characterization of these multilayer emulsions in relation to the spray drying process, especially with respect to the type of atomization and the drying behavior are lacking and literature on the oxidative stability of spray-dried multilayer emulsions is scarce.

Scope

Aim of the present study was to investigate the impact of atomization and drying on the functionality of emulsions with a modified o/w interface consisting of beta-lactoglobulin and pectins with varying degree of methylation.

Experimental Approach

Beta-lactoglobulin and pectin (low and high methylated) were used for preparation of single and multilayer-emulsions containing fish oil. Electrostatic interactions were analyzed via zeta potential measurements. Elasticity of emulsion interfaces was analyzed by droplet shape analysis. The single droplet drying behavior was characterized by acoustic levitation. Spray droplet and oil droplet size distribution were measured by laser diffraction after two-fluid-nozzle and rotary atomization at different energy inputs. Spray drying was carried out at 180/70 °C inlet/outlet temperature. The morphology of the particles was analyzed by scanning electron microscopy. The oxidative stability of the fish oil was monitored via hydroperoxide and propanal content.

Major results

The single droplet drying behavior of the differently stabilized emulsions was similar as examined by levitation. With regard to the atomization process, the emulsion spray droplet size generally decreased with increasing energy input to the atomizer. In two-fluid nozzle atomization but not in rotary atomization, the spray droplet size distribution was markedly influenced by the differences in emulsion viscosity. The spray droplet size distribution was narrower with rotary atomization compared with two-fluid nozzle atomization. The oil droplet size of the emulsions was only slightly affected by the different energy inputs during atomization. However, in the reconstituted state, the oil droplet size was higher than in the original emulsions, which can be attributed to coalescence.

The oxidative stability of the oil was influenced by both the physical state of the emulsions and the different constituents at the o/w-interface. For instance, in the liquid state the oxidative stability was higher in the original emulsions when compared to the reconstituted emulsions. Furthermore, the oxidative stability was affected by the type of pectin and thus the intensity of the oil droplet charge. It can be assumed that in liquid and spray-dried emulsions, not only the particle charge but also the constitution of the interface itself influences the oxidative stability of encapsulated oil.

* Corresponding author: yvonne.serfert@tu-berlin.de

Investigation of polymerization of NVP and drying of PVP in an acoustic levitator using a smart camera for online process measurement

J. Laackmann[1*], S. Ahmed[2], R. Sedelmayer[1], M. Klaiber[2], W. Pauer[1], S. Simon[2], H.-U. Moritz[1]
[1]Institute for Chemical Engineering, University of Hamburg, Hamburg, Germany
[2]Institute of Parallel and Distributed Systems, University of Stuttgart, Stuttgart, Germany
julian.laackmann@chemie.uni-hamburg.de and silvia.ahmed@ipvs.uni-stuttgart.de

Abstract

Spray processes are widely used in the chemical, pharmaceutical and food industries for the drying of products. The high surface to volume ratio of sprayed droplets allows high mass and heat flows under moderate conditions which predestines these processes for the treatment of sensitive substances. Furthermore the spray processes can be applied to reactive solutions where the synthesis and the drying of the product are combined. While the properties of the sprayed solution as well as the dried product are easily accessible via offline techniques the processes during the fall of the droplets are hard to measure. However these processes are important especially for the formation of the morphology of the dried particles. In order to investigate the occurrences during the drying process acoustic levitation can be employed for the contactless handling of droplets. Without the drawbacks of analyzing mass transfers on scales or on filaments the acoustic levitation is an ideal model system for falling droplets. Because of the fixed position of the droplet in the acoustic field droplets can be observed under conditions similar to those in a spray tower making the processes during the fall accessible to online analytics.

In this study the synthesis and drying of polyvinylpyrrolidone (PVP) in an acoustic levitator is investigated. The concentration of the monomer in the droplet is reduced by evaporation and polymerization. The competition between these partially parallel processes is monitored on the one hand by imaging to determine the loss of solvent and monomer due to evaporation and on the other hand by Raman spectroscopy to determine the consumption of monomer by polymerization. Combining Raman spectroscopy with principal component analysis demonstrated the feasibility of observing evaporation as well polymerization by Raman spectroscopy. This allows for investigation of both competing processes with the help of only one analytical method. In this context a smart camera system with integrated processing capabilities is introduced to compute droplet characteristics from the images online. The required algorithms were implemented in parallel on the field-programmable gate array (FPGA) of the smart camera to demonstrate the performance of smart camera systems in general. It was shown that the image processing algorithms for extracting relevant features of the objects can be implemented as single pass algorithms. With this kind of algorithms pixel data can be processed line-by-line in the same order the information is provided by the image sensor of the camera. Thus there is no need to store full images but only a single line. This is an important advantage when frame rates increase to several hundred frames per second or more as memory and bandwidth in most hardware architectures are limited. Since the pixel stream is processed in the camera and only relevant information is passed on to an external device, online measurements with high frame rates are achievable. This may be interesting for further applications in spray diagnostics.

---

*Corresponding author: julian.laackmann@chemie.uni-hamburg.de

Analysis of confined spray processes for powder production

A. Lampa*, U. Fritsching
Particles and Process Engineering, University of Bremen, Germany
lampa@iwt.uni-bremen.de and ufri@iwt.uni-bremen.de

Abstract

Solid particles or powders may be produced in spray processes like spray drying or spray polymerization. The tailoring of the particle and powder properties that are produced within spray processes is influenced by various unsteady transport processes in the dispersed multiphase spray flow in a confined spray chamber. In this context differently scaled spray structures in a confined spray environment have been analyzed. The basic setup of the study consists of a twin-fluid atomizer central top-spraying in a confined spray chamber. The atomizer gas can be heated up to 500°C, providing enhanced disintegration and drying conditions. In contrast to a traditional spray drying process by using heated atomization gas in the twin-fluid atomizers no superimposed massflow of hot dry air is applied. Thus energy may be saved. Mixing phenomena of momentum, energy and species in the near region of the nozzle play an even bigger role for this hot-gas atomization and drying process than in a conventional spray dryer. Large scale spray structures in enclosures like:

- the oscillation of the spray plume,
- the shear flow instabilities at the spray edge,
- gas entrainment and recirculation zones and
- droplet cluster formation

greatly influence the heat and mass transfer for individual droplets within the spray cone. These spray structures are significantly influenced by the nozzle's operational parameters (air-liquid ratio, atomizer gas pressure, feed fluid, etc.). Another process parameter which has been scarcely investigated so far is the geometry of the spray chamber. Therefore differently shaped conical spray chamber designs with square cross sections have been analyzed experimentally and numerically.

Numerical simulations of the bounded spray flow have been carried out using Reynolds-Averaged-Navier-Stokes models (RANS) and Large-Eddy-Simulation (LES) models with lagrangian particle tracking for the droplet motion. The trajectory of droplets/particles in the gas has been tracked with respect to their environment to estimate the drying conditions. Especially the local droplet concentration and the turbulent motion characteristics of the gas phase and its interaction with the droplets have been analyzed. While the spray structures outside the spray cone flow like recirculation and entrainment zones of gas can be well predicted by RANS models, the prediction of large scale turbulent structures within the spray cone is realistic to be realized only with Detached-Eddy-Simulation (DES) and LES models that are more complex due to the modeling and computation of the initialization and propagation of vortex patterns. Synthetic perturbation generators for the inflow conditions have been applied to induce realistic vortex patterns and to promote the unsteadiness of the spray flow. In the focus of the LES investigations is the spatial distribution of droplets within the spray. The specific process shows significant clustering of droplets, which is induced by the coherent flow structures within the spray. The numerical results show good agreement with Particle-Image-Velocimetry (PIV) measurements in a lab scale process under isothermal conditions. The Influence of the spray chamber design on the droplet/particle-gas interactions will be analyzed. The project is part of the SPP 1423 Prozess-Spray: „Herstellen funktionaler Feststoffpartikeln in Sprühverfahren: Von den Anforderungen an das Pulver und an seine Eigenschaften zum geeigneten Prozess“ funded by the German Research Foundation (DFG).

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

Nozzle Geometry Effects on Primary Atomization

M. Arienti
Sandia National Laboratories, Livermore, CA 94550, USA
marient@sandia.gov

M. Sussman
Florida State University, Tallahassee, FL 32306, USA
sussman@math.fsu.edu

Abstract

The increasing demand for predictive simulations of primary atomization (the complex process of spray formation right after the injection of liquid fuel) suggests pushing the inlet boundary condition upstream of the injection hole, so that effects due to the nozzle geometry can be properly included. In this work, we address the coupling between internal (with respect to the injection hole) and external flow, in relation to the spray characteristics of primary atomization. A simulation capability is presented where internal and external flow can be seamlessly calculated across the injection hole. The injection is assumed to be sub-critical, so that the external liquid-gas interface can be handled by the combined level-set volume-of-fluid (CLSVOF) method. The solid wall boundary is represented by a second level-set function ψ on the same Cartesian, block-structured grid. A grid cell belongs to the physical flow domain if $\psi > 0$. Cells that are near the wall boundary, but outside of the flow domain, form a narrow band of ghost cells where velocity boundary conditions are assigned. Assuming no cavitation, results from the calculation of a scaled-up, transparent, six-hole Diesel nozzle are compared to experimental observations. Flow field features appearing just before the liquid is injected are highlighted, and their impact on spray characteristics is briefly discussed.

Large Eddy Simulation-Probability Density Function modelling of nucleation and condensation of DBP droplets in a turbulent jet

N. Seubert, A. Kronenburg *, O. T. Stein

Institut für Technische Verbrennung, University of Stuttgart, Herdweg 51, 70174 Stuttgart, Germany

Y. Ge, M.J. Cleary **

The University of Queensland, School of Mechanical and Mining Engineering, QLD 4072, Australia

Abstract

Homogeneous nucleation and growth of dibutyl phthalate (DBP) droplets forming in a turbulent jet are modelled using LES combined with a probability density function/Monte Carlo method. LES resolves the largest turbulence structures but small scale processes like droplet nucleation and growth require closure. The Monte Carlo particles represent the gas temperature and species mass fractions, as well as the particle size distribution (PSD) of the DBP droplets. Thus all nucleation and growth terms are in closed form and the Interaction by Exchange with the Mean (IEM)-mixing model is used to account for the scalar mixing of the particles within each LES cell. The (physical) particle ensemble is represented by discrete size bins on each (stochastic) Monte Carlo particle. The LES-PDF method reproduces the experimental data well, and an analysis of the nucleation and growth terms demonstrates that the correlations of the large turbulent scales must not be ignored when modelling the averaged nucleation rate. In contrast, the large scale correlations do not significantly affect the particle growth, but the small scale, LES-subgrid term notably reduces particle growth for a wide particle size range.

---

*Corresponding author: kronenburg@itv.uni-stuttgart.de
**Current address: University of Sydney, NSW 2006, Australia

Multiphase flow simulations using the meshfree Smoothed Particle Hydrodynamics method

C. Hoefler*, S. Braun, R. Koch, H.-J. Bauer
Institute of Thermal Turbomachinery, Karlsruhe Institute of Technology, Germany
corina.hoefler@kit.edu

Abstract

In order to numerically model primary atomization a fully three-dimensional code based on the meshless Lagrangian Smoothed Particle Hydrodynamics method has been developed. The present paper focuses on the capability of our code to accurately capture multiphase flow phenomena.

Within the scope of this work a fully three-dimensional code based on the meshless Lagrangian Smoothed Particle Hydrodynamics (SPH) method [1][2] has been developed and validated for various test cases. The long-term objective is the development of a so called virtual atomizer test rig in order to avoid the cumbersome, time-consuming and costly iterative process of nozzle design and repeated manufacturing. This SPH approach was chosen to overcome the difficulties inherent to meshbased techniques such as the Volume of Fluid or Level Set method. Those methods suffer from complex and often inaccurate reconstruction of three-dimensional interfaces and mass loss, respectively. Inaccurate reconstruction of the interface causes deficiencies of the predicted interface curvature, which in turn results into erroneous surface tension forces and thus inaccurate modeling of the primary atomization process. In addition, gridbased methods require enormous computational resources to accurately capture the phenomena occurring during primary atomization.

As a promising alternative the Smoothed Particle Hydrodynamics method is based on the Lagrangian description of the governing equations at movable spatial discretization points. The flow quantities at a certain location in space are interpolated from the neighboring discretization points via a weighing function. Different phases can be modeled by assigning an appropriate marker function and the according fluid properties to the corresponding discretization points. A huge benefit is the inherent advection of the phase interfaces, as the discretization points or so called particles move with the velocity field of the fluid flow. Thus, there is no need for any interface tracking or capturing techniques. The SPH method has already been successfully validated for shear-driven single phase flows and a qualitative comparison between the modeling of free surface flows and experiments has been conducted [3]. Within the present paper we focus on the capability of the method concerning multiphase flow simulations with particular emphasis on surface tension.

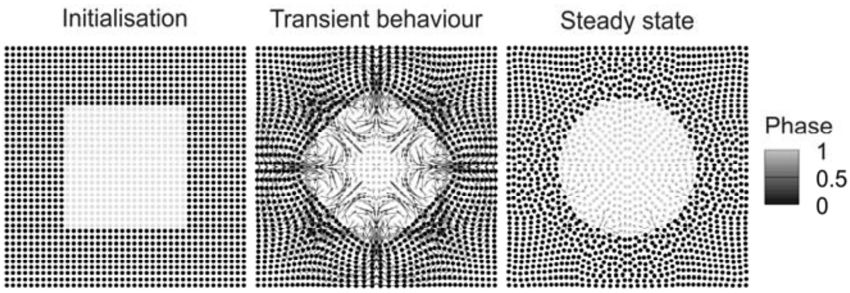


Fig.1: Droplet formation

As depicted in Fig. 1, the formation of a droplet from non-equilibrium state and its transient behaviour could be modeled successfully. The pressure jump across the curved interface of the droplet depending on the surface tension coefficient has been reproduced in excellent agreement with theory. Moreover, the promising first step towards the modeling of planar sheet disintegration at an airblast atomizer edge is presented. Test cases performed so far demonstrate the capabilities of the Smoothed Particle Hydrodynamics method for simulating multiphase flows and the initial stage of numerically predicting primary atomization.

* Corresponding author: corina.hoefler@kit.edu

[1] Gingold, R. and Monaghan, J. J., *Monthly Notices of the Royal Astronomical Society*, 181, 375 – 389 (1977)
[2] Lucy, L., *The Astronomical Journal*, 82, 1013 – 1024 (1977)
[3] Hoefler, C., Braun, S., Koch, R. and Bauer, H.-J., *ILASS Europe – 24th European Conference on Liquid Atomization and Spray Systems*, Estoril, Portugal, 2011

A DNS database to improve atomization modeling

B. Duret, T. Menard, J. Reveillon and F.X. Demoulin*
UMR6614-CORIA, Technopôle du Madrillet, BP 12, Avenue de l'Université, 76801
Saint-Etienne-du-Rouvray Cedex, France

Abstract

Recent advances in interface tracking method allows us to use them as a tool to study primary atomization. Some recent works have performed accurate Direct Numerical Simulation of turbulent liquid jets which are very promising [1] [2]. Unfortunately, these simulations have shown that, even with a high resolved DNS of 6 billions points [2], the finest scales of the flow are not resolved. This issue shows that subgrid models are necessary to modelize the physics under this particular scale.

In order to improve modeling, it is useful to study simpler configurations which allows us to solve all relevant scales of the flow. In single-phase flows, DNS of Homogeneous Isotropic Turbulence (HIT) is still used to study scalar mixing and the kolmogorov theory. Extension of this configuration to two-phase flows to characterize important processes (for example turbulence/interface interactions, vaporization) is emerging in recent DNS studies. Concerning vaporization, few studies have emerged using complex numerical methods but are limited to single droplets or laminar jet, because of their high computational cost. Duret et al. [3] have also studied vaporization but in the context of turbulent atomization (high Weber and Reynolds number). A passive scalar is used to represent the evaporation and mixing process in a two-phase turbulent flow, as a first approach.

The aim of this work is to pursue these studies by analyzing the interface behaviour and the vaporization of two-phase flows in a forced HIT. This analysis is performed over a large range of liquid volume fractions. From these results we can extract useful informations in order to improve modeling of primary atomization. The liquid gas interface is represented in Figure 1 to illustrate the numerical configuration for a liquid volume fraction of 5 %.

Both phase are resolved in DNS, the interface tracking method used is a coupled Level Set/VOF method. High density ratio between the two phases is chosen to simulate realistic engine conditions. Quantitative and qualitative aspects are analyzed, with a main focus on interfacial quantities (equilibrium interface density and Weber number), which are key parameters for primary atomization modeling.

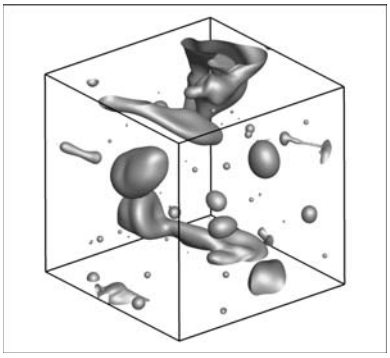


Figure 1. Interface visualization, liquid volume fraction $\phi = 5\%$

References

[1] T. Menard, S. Tanguy, and A. Berlemont, "Coupling level set/vof/ghost fluid methods: Validation and application to 3d simulation of the primary break-up of a liquid jet," *International Journal of Multiphase Flow*, vol. 33, no. 5, pp. 510 – 524, 2007.

[2] J. Shinjo and A. Umemura, "Simulation of liquid jet primary breakup: Dynamics of ligament and droplet formation," *International Journal of Multiphase Flow*, vol. 36, no. 7, pp. 513 – 532, 2010.

[3] B. Duret, G. Luret, J. Reveillon, T. Menard, A. Berlemont, and F.-X. Demoulin, "Dns analysis of turbulent mixing in two-phase flows," *International Journal of Multiphase Flow*, vol. 40, no. 0, pp. 93 – 105, 2012.

*Corresponding author: demoulin@coria.fr

Application of the immersed boundary method to simulate flows inside and outside the nozzles

E. Noël, A. Berlemont, J. Cousin*, T. Ménard
UMR 6614 – CORIA, France
emeline.noel@coria.fr, berlemont@coria.fr, cousin@coria.fr, thibault.menard@coria.fr

Abstract

DNS of two phase flows remains a major topic of research. One of the main challenges is to capture the interface behavior in the nozzle vicinity with enough accuracy in order to well describe the primary breakup. A major task to achieve such a simulation is to handle jump conditions at the liquid-gas interface without artificial smoothing; this is made possible thanks to the Ghost Fluid method. The Level Set method is used to represent the interface and is coupled with the Volume of Fluid method to ensure mass conservation. This coupling already showed a good behavior for the simulation of primary jet breakups (Ménard et al [1], Shinjo and Umemura [2] for instance). In such situations, boundary conditions strongly influence the perturbation initiation on the liquid-gas interfaces. For instance, Berlemont et al [3] proposed a coupling between simulations of flows inside and outside the nozzle using different solvers where calculated velocity profiles at the nozzle outlet were used as initial condition for the jet primary breakup. Results were encouraging despite the fact that some information can be lost during the coupling.

The objective of the present work consists in a strong coupling between internal and external flows by keeping the same numerical tools. Here the walls which delimit the fluid volume inside the nozzle have to be accounted for. As the walls can have complex shapes, the chosen methodology needs to be able to perform simulations within the scope of regular Cartesian grids. The chosen approach is based on a discrete forcing immersed boundary method similar as the original Mohd Yusof's proposal [4]. The solid-fluid boundaries are represented by a level set function identical as the one already used to take into account the liquid-gas interfaces. The major difference concerns the physical conditions to be written at the solid-fluid boundary. As far as numerics are concerned, a projection method is used to solve incompressible Navier Stokes equations. A uniform staggered grid is used, spatial discretization is carried out with a WENO scheme for convective terms, second order Runge Kutta method is used for temporal derivatives. Poisson equation is discretized with a second order central scheme and solved with a BICGSTAB method. Finally, specific care has been carried out to improve simulation capabilities with MPI parallelization.

In a first part of the paper, we present in details the methodology to impose physical conditions on the immersed body. The second part of the paper presents the validation of the method on two 2D test cases. First the equilibrium of a droplet on a solid surface is investigated and validated thanks to analytical results. Second, the simulation of an injection event is presented and it is showed that the velocity fluctuations created inside the nozzle directly affects the liquid/gas interface downstream the injector tip.

References

[1] T. Ménard, S. Tanguy and A. Berlemont. Coupling level set/VOF/ghost fluid methods: Validation and application to 3D simulation of the primary breakup of a liquid jet. Int. J. of Multiphase Flow, 33(5): 510-524, 2007.
[2] J. Shinjo and A. Umemura. Simulation of liquid jet primary breakup: Dynamics of ligament and droplet formation. Int. J. of Multiphase Flow, 36(7): 513-532, 2010.
[3] A. Berlemont, J. Cousin, M. Doring and T. Ménard. A coupling between internal flow and primary breakup simulations – case of a compound nozzle. Proc. of the 11th ICLASS, 2009.
[4] J. Mohd Yusof. Combined Immersed-Boundary/ B-spline methods for simulations of flow in complex geometries. CTR Annual Research Briefs. Standford CA, 1997.
[5] W. Sander and B. Weigand. Direct numerical simulation and analysis enhancing parameters in liquid sheets at moderate Reynolds numbers, Phys. Fluids 20, 053301, 2008.

* Corresponding author: cousin@coria.fr

Drying kinetics and packing of particles of silica-water nanofluid droplets dried in an acoustic levitator

R. Mondragón[1], J. C. Jarque[1], J. E. Juliá[2*], L. Hernández[2]
[1]Instituto de Tecnología Cerámica, Universitat Jaume I.
[2]Departamento de Ingeniería Mecánica y Construcción, Universitat Jaume I.
Campus de Riu Sec, 12071 – Castellón de la Plana. Spain
rosa.mondragon@itc.uji.es, juancarlos.jarque@itc.uji.es, bolivar@emc.uji.es and lhernand@emc.uji.es

Abstract

In many industrial applications such as ceramics, food products, detergents and pharmaceuticals, the spray drying process is used to produce powders with different characteristics. Drying models predicting the drying kinetics of single droplets can be used to relate the final powder properties (such as the final grain diameter, mean porosity, compacity, morphology, microstructure, etc.), with the spray dryer design and process parameters.

The drying models available can be classified in different categories. Drying models based on reaction engineering approach (REA) have been found promising due to its simplicity and high accuracy at different drying conditions. However, the model applicability is limited by the range of materials whose drying behavior has already been experimentally studied.

In this work single droplets of silica-water nanofluids (Aerosil 200) were dried in an acoustic levitator under different experimental conditions of initial solid mass load (0.02 w/w < Y_S < 0.20 w/w), *pH* value (2 < *pH* < 10), salt concentration (0 M < *[NaCl]* < 0.05 M), air temperature (80ºC < *T* < 120ºC), and initial droplet volume (0.3 µl < V_0 < 0.8 µl). The drying curves (*X*=f(t)) were experimentally obtained for each test conducted and the REA model was used to model the experimental data. To do this, correlations for the activation energy were obtained for this system at different drying conditions. These correlations enlarge the database existent which only contains activation energies for materials used in the food industry. Theoretical curves show a good agreement with the experimental data.

For each experiment the critical moisture content was obtained experimentally and theoretically from the drying curves, showing a good agreement. The final grain diameter can be related with the critical moisture content by means of a mathematical equation. The results obtained were compared with those resulting from the image processing of the videos recorded for each experimental test. The theoretical results show a good agreement with the experimental ones.

Finally, the packing of particles inside the droplet has been checked to be constant for a particular system and equal to the random close packing. The packing fraction in independent on the solid content, the drying temperature, the *pH* of the suspension, the salt content and the initial droplet volume. This packing can be obtained from the modelling of the viscosity data to the Quemada equation. For nanoparticles the value of the packing fraction is lower than for hard microparticles due to the presence of cohesive forces that difficult the movement and rearrangement of particles.

The final diameter and the packing fraction were used to calculate the shell length and the degree of hollowness of the grains. Experimental and theoretical results for the shell length and the ratio shell length/grain diameter show a good agreement.

---

* Corresponding author: bolivar@emc.uji.es

Analysis of the fuel adhering to a model engine cylinder by using time series LIF methods

Shinya Okamoto*, Hisanobu Kawashima, Tsuneaki Ishima and Kenjiro Nakama
Gunma University, Japan
t10802201@gunma-u.ac.jp, hkawa@gunma-u.ac.jp, ishima@gunma-u.ac.jp and knakama@hhq.suzuki.co.jp

Abstract

In the gasoline engine, the adhesion of the gasoline droplets to the wall is an unexpected phenomenon that is the reason of a unburnt HC and oil dilution. It is important to analyze the behavior of the fuel adhesion to the wall because of improving the efficiency and decrease in the negative environmental impact material from the engine. The purpose of this research is to know an absolute measurement of the fuel adhesion thickness and amount by fuel spray in the model engine. In this study, for improving the accuracy of thickness measurement, using the ratio of fluorescence intensities is proposed. An improved LIF (Laser Induced Fluorescence) method, that is used the ratio of fluorescent intensity I/I_{ref} (I: fluorescence intensity, I_{ref}: fluorescent intensity of the fluorescent plate), is proposed and tried to measure. This technique is able to correct the influence of light source irregularity.

To examine the relation between the ratio of fluorescence intensity I/I_{ref} and the thickness t, the calibration experiment is carried out. Perylene ($C_{20}H_{12}$) is used for a fluorescent dye. It is dissolved by the toluene, and it mixed with n-heptane for surrogate fuel. UVLED (the emission peak wavelength: 375 nm) is used for the light source. As the result of this experiment, it is shown that there is a quadratic function relation to thickness and I/I_{ref}.

The fuel adhesion measurement using a model engine is performed by the improved LIF method. The improved LIF method and a high speed video camera are combined. A light source and surrogate fuel are same as a calibration experiment. The experiment is performed in the flow field of the model engine cylinder. The pictures are able to be converted to the fuel thickness distribution by an improved LIF method. The amount of fuel adhesion is calculable in thickness. As a result, the behavior of the fuel adhesion is able to be measured. The fuel adhesion thickness distribution and the amount by the difference of the injection conditions are presented in time series.

---

* Corresponding author: t10802201@gunma-u.ac.jp

Infrared thermography based fuel film investigations

F. Schulz*, J. Schmidt,
Institute of Fluid Dynamics and Thermodynamics
Otto-von-Guericke University Magdeburg
PO 4120, 39106 Magdeburg, Germany
florian.schulz@ovgu.de and juergen.schmidt@ovgu.de

Abstract

In this paper we present an infrared thermography based method for systematic studies of the temporal and spatially resolved heat fluxes during the whole process of spray impingement to film evaporation. This is mainly possible due to new developments in infrared camera technology. The influence of three test parameters was examined - the injector position, the injection pressure and the wall temperature.

In DI-engines fuel wall films are an important origin of soot particles. In order to reduce fuel wetting a detailed analysis of wall film forming is necessary. The strong influence of the surface temperature on the droplet-wall interaction and the resulting liquid deposition is known. But the quantitative forecast of wall films caused by dense sprays is usually poor because of the various parameters influencing the occurring heat fluxes which reduce the surface temperature during the injection.

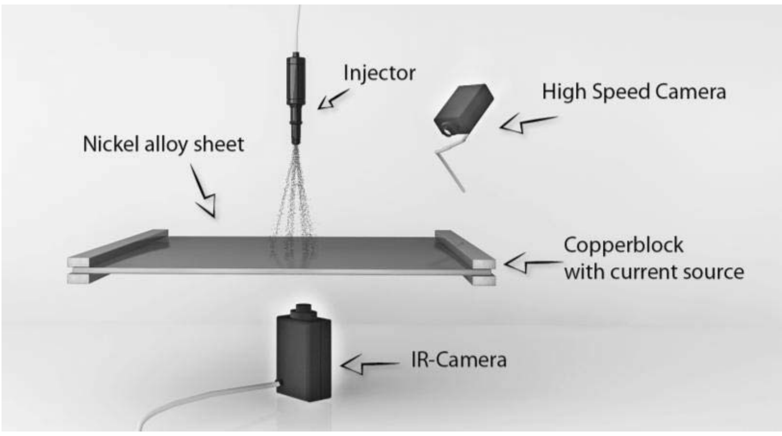


Figure 1 Schematic experimental setup

The maximum temperature decrease is sensitive to all three test parameters. The overall heat inserted into the fuel is almost independent of the distance between injector and wall. Contrary to the first guess that the spray impact takes more energy out of the wall than the following film evaporation, it can be shown that underneath the Nukiyama-Temperature the film evaporation needs up to 15 times the heat used during the spray-wall interaction.

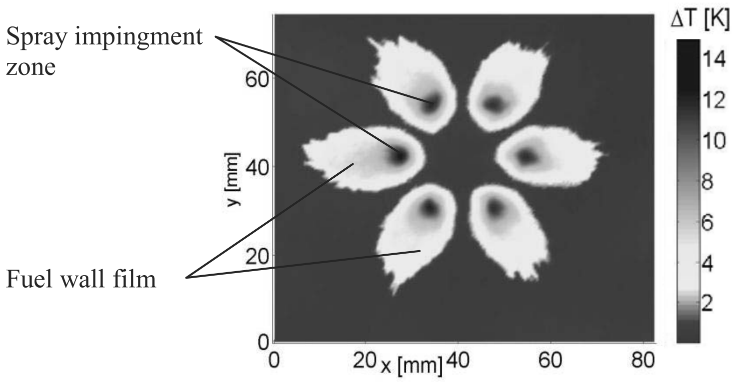


Figure 2 Field of temperature-differences of the fuel wall film caused by a six hole nozzle

* Corresponding author: florian.schulz@ovgu.de

Thickness imaging of evaporating liquid water films by simultaneous Tracer LIF, Raman imaging and Diode Laser Absorption Spectroscopy

D. Greszik, H. Yang, T. Dreier[1*], C. Schulz
Institut für Verbrennung und Gasdynamik (IVG), and Center for Nanointegration (CENIDE)
University of Duisburg-Essen, Germany

Abstract

Knowledge about the thickness of liquid films is important, e.g., in flash boiling of fuel spray impingement on cylinder walls in internal combustion engines or of aqueous urea solutions and subsequent evaporation on exhaust pipes of Diesel engines during exhaust gas after-treatment in the selective catalytic reduction (SCR) of nitrogen oxides. For purposes of process optimization and for providing validation data in CFD-based modeling of these processes, non-intrusive, quantitative liquid film thickness measurement techniques are desired, which potentially provide two-dimensional imaging capabilities and/or high data rates for temporally resolved measurements of the liquid film thickness is required. In this work we present the application of two imaging-based laser-diagnostic techniques – laser-induced fluorescence (LIF) and spontaneous Raman scattering (RS) – as well as a point measurement method – near-infrared diode laser absorption spectroscopy (NIR-DLAS) – for the simultaneous measurement of liquid film thickness during the time-varying evolution of spray-deposited liquid water droplets on transparent surfaces, with specific applications in an air-fed flow duct.

* Corresponding author: thomas.dreier@uni-due.de

Fine Spray Generation for Single-Wall Carbon Nanotubes (SWCNT) Production

M. B. Abuhesa*, G. G. Nasr, and G. C. Enyi
Spray Research Group (SRG), University of Salford, Manchester, UK
said64msa@yahoo.com and G.C.Enyi@edu.salford.ac.uk

Abstract

Gas flaring is the burning of natural gas, which cannot be processed or sold during oil and gas production and processing operations. In past decades, gas flaring was believed to be environmentally tolerable. On the contrary the flaring of gas has been found to be an impediment to the environment; this has led to attempting to tackle the problem of gas flaring to advance it to an acceptable level worldwide.

There are currently over 700,000 gas wells worldwide and according to World Bank about 110 billion cubic meters (bcm) of natural gas are flared annually. If all the flared gas is stopped and instead converted to hydrogen (H_2) and carbon (C) nanotubes, the reduction of CO_2 emissions which stands at 400 million metric tonnes per annum could be drastically reduced. The hydrogen component produced from the reaction could then be used for power generation and the irregular carbon nanotubes as composite materials.

The main aim of this investigation was to develop an alternative approach to continuous gas flaring in oil and gas industry. Sprays and atomisation techniques were experimentally employed as a promising option for the production of Single-Walled Carbon Nanotubes (SWCNT).

Laboratory experiments were performed to test the concept of using this technique to study the effects of the related parameters on its behaviour by spraying simulated catalyst solution (i.e. water) droplets into a hydrocarbon gas stream (methane as a carbon source) using a specially designed "atomiser device" that incorporates a number of pressure swirl atomisers.

A furnace was installed underneath of the "atomiser device" and the stream of droplet particles fell down through the furnace (400 - 800^o C). Reactions which took place inside the furnace produced the Single-Wall Carbon Nanotubes (SWCNT) material from natural gas stream. The effect of water flow rate (0.001- 0.005 l/min) and water supply pressure (≤12MPa) as well as the gas flow rate (0.3-0.4 l/min) together with the downstream distance of the corresponding atomiser device on the droplet size distribution (≤ 5μm) were also characterised. The qualitative and quantitative analysis of the results obtained from the series of trials demonstrated that the production of SWCNT is certainly possible by using a combination of pressure swirl atomisers.

* Corresponding author: said64msa@yahoo.com

Experimental and Numerical Study of Flow and Heat Transfer due to Intermittent Impinging Mist Jets

A. D. Nazarov*, M. A. Pakhomov, A. F. Serov, V. I. Terekhov
Kutateladze Institute of Thermophysics, Russian Academy of Sciences, Siberian Branch, 1, Acad. Lavrent'ev Avenue, Novosibirsk, 630090, Russia.
E-mails: nazarov@itp.nsc.ru, serov@itp.nsc.ru, pakhomov@ngs.ru, terekhov@itp.nsc.ru

Abstract

Impinging jet is employed in many industrial applications (for example drying of sheets of various materials, cooling of electronic devices and GT blades, printing processes, etc) due to heat and mass transfer enhancement around of stagnation point. In many technical applications intermittent (pulsed) flow occurs due to moving parts, by vibrations or flow oscillations. Studies performed show a complex physical situation that in some cases leads to significant heat transfer enhancement, however, also decrease of heat transfer can occur. Spray impingement on hot surfaces has capability of removing large amounts of heat due to use of the latent heat of evaporation. The impingement of pulsed mist flow on hot surfaces also occurs in practical situations, which require a comprehensive knowledge of the flow patterns and the interaction of the spray with the impinging surface.

The aim of the present work is experimental and numerical simulation of the effect of droplets evaporation on the flow and heat transfer in turbulent impinging pulsed mist jets.

The experimental set-up consists of the heat exchanger with the original digital calorimeter and pulse spray source. The heat exchanger is made of high heat-conducting copper with the plane dimensions of 140 × 140 mm and thickness of 25 mm. The air-droplets flow is formed by the pulse spray source from 16 sprayers (matrix of 4 × 4) and 25 air nozzles (matrix of 5 × 5). The source generates pulses of droplets (duration t_{on} from 2 ms to 10 ms, frequency f_i from 1 Hz to 50 Hz and with a velocity – from 0 ms^{-1} to 20 ms^{-1}) that move to the surface of the heat exchanger in a constant cocurrent gas flow (speed U_{L1} – from 0 to 25 ms^{-1}). The main studies have been carried out, when the distance between the heat exchanger and source was L = 230 mm. At this position on a cross-section of heat exchanger surface the source of pulse gas-droplet flow forms a two-phase flow with the area of 300 × 300 mm^2. Near the plate surface (L ~ 200 mm) the jets are split into separate droplets of two main sizes: large (0.12 ÷ 0.15) mm and small ones (0.045 ÷ 0.05) mm.

The numerical model based on the axisymmetrical Euler/Euler approach. For the gas phase were used the set of non-steady-state RANS equations. In the study is considered for the gas phase the second moment closure by Craft and Launder (1992). For the dispersed phase velocity fluctuations were used kinetic stresses equations, turbulent heat flux and temperature fluctuations equations by Zaichik (1999). The two-way coupling model was used along with the particulate feedback onto the mean distribution of the gas phase.

The comparison between our predictions and experiments for pulsed mist jet measurements was provided. These results are presented in the Fig. Agreement between our computation results and numerical and measurements data was rather good especially for the small value of the time of the pulse t_{on} (line *1*) when the wall surface in the experiments is dry. The increase of t_{on} leads to the formation of liquid film and spots on target plate (observed in the measurements) and the residual between our simulations and experiments is rise (line *2*). We did not take into account the film formation on the wall from deposited droplets. We assumed droplets deposited onto the impinging surface momentarily evaporate (solid curves). The predictions without taking into account the heat spend the droplets heat up and evaporation q_{WL} (dashed curve) agreed better with measurements.

This work was partially supported by the Russian Foundation for Basic Research (Projects No. 12-08-00504-a).

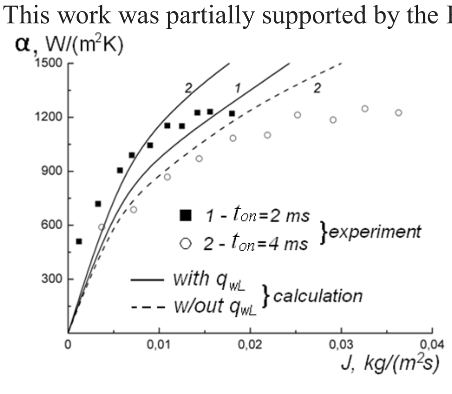


Fig. Heat transfer coefficient in intermittent mist impinging jet. Symbols are experimental results, curves are computations. H = 230 mm, time of the pulse t_{on} = 2 and 4 ms, f_i = 1 – 50 Hz, $T = T_\infty$ = 295 K, T_W = 373 K, U_{L1} = 14.8 m/s. *1* – t_{on} = 2 ms, *2* – t_{on} = 4 ms; solid curves – instant evaporation of droplets on the surface of the heat exchanger (with q_{wL}); dashed curve – w/out q_{wL}.

* Corresponding author: nazarov@itp.nsc.ru

Numerical and experimental study of spray cooling of a heated metal surface

M.R. Guechi*, P. Desevaux, P. Baucour
FEMTO-ST Institute, Energy Department, 2 avenue Jean Moulin, F-90000 Belfort, France
mguechi@univ-fcomte.fr, pdesevau@univ-fcomte.fr and philippe.baucour@univ-fcomte.fr

Abstract

The spraying of an impinging jet is an effective way to cool heated surfaces. The objective of this study is to develop a numerical model to predict the heat transfer with phase change between a hot plate surface and a two-phase impinging jet. Different two-phase modeling approaches (Lagrangian and Eulerian methods) are compared. The influence of the spray nozzle operating conditions (pressure, flow rate, droplets size) and of the distance between the nozzle exit and the surface impact is analyzed. The numerical results are compared with measurements obtained on an experimental test bench. The confrontation numerical/experimental is carried out by comparing the distribution of temperature at the surface of the plate and the heat transfer coefficient. This comparison shows that it is the Eulerian model which seems most capable to take into account the evaporation of the droplets in contact with the heated plate and consequently, which gives results more in agreement with the experiments. However, the simulation performed with this model show a strong dependence of the results to the turbulence model used.

Introduction

This work falls under a research project aiming at improving the cooling of electric motors of great power using an impinging jet spray. A computational approach by CFD is planned in order to evaluate the effectiveness of the spray cooling and to dimension and optimize the cooling system. For that, it is necessary to develop a numerical model making it possible to predict the heat transfer with phase change between a heated surface and a two-phase impinging jet and to validate the CFD results against experiments.

Experimental Setup

A test bench designed to enable the experimental study of the spray cooling of a heated metal surface has been constructed. It consists of a heating system, a spray nozzle and a data acquisition system. The spraying surface corresponds to the top surface of a copper block cylinder heated by means of a 400 W cartridge heater. 12 J-type thermocouples were embedded at various depths below the heater surface to provide the temperature gradient and temperature profile within the copper block cylinder. The test bench enables the investigation of the spray cooling process for various operating conditions (different heat fluxes, different distances between the spray nozzle and the hot surface).

Numerical approach

The numerical model was implemented in a commercial CFD code. Different two-phase modeling approaches (Lagrangian and Eulerian methods) and several RANS turbulence models were tested.

Results and Discussion

The results obtained relate to the evolution of the local heat transfer coefficient and of the surface temperature along the plate. The computational results more in agreement with the experiments were obtained with the two-phase Eulerian model associated with the Realizable k-epsilon turbulence model. The CFD model was then used under various operating conditions of the spray system (water pressure, flow rate, droplets size) and for various conditions of heating. The influence of the distance between the nozzle exit and the surface impact was also analyzed. The results obtained show strong variations of the surface cooling with the spraying system operating conditions and the comparison between simulations and experiments shows the capability of the model to correctly predict the phase change heat transfer during the spray cooling. The model developed must allow, in a short time, the optimization of the spraying system before its implementation on an electric motor.

*Corresponding author : mguechi@univ-fcomte.fr

Liquid Film Dynamic on the Spray Impingement Modelling

C. Rodrigues, J. Barata, A. Silva*
Dept. of Aerospace Science, University of Beira Interior, Covilhã, Portugal

Abstract

The dynamic of droplets impacting onto a dry or wet solid surface plays an important role in a wide variety of fields, such as in ink-jet printing technologies, spray painting and liquid-fuelled combustors. However, it has only been in the near decade that a major scientific effort has been pursued in order to acquire a comprehensive knowledge about the mechanisms underlying the spray impingement process. Yet, this complete physical understanding of the phenomena found during the spray/wall interaction is still lacking mainly due to the host of parameters that influence the outcome. One of those parameters, which curiously is often neglected in spray impingement models, is the liquid film accumulated on the wall due to the deposition of the impinging drops.

The present work aims at developing and integrating the liquid film formation in a multiphase computational model. This paper follows on from a set of previous studies [1-3] that seek to refine a flexible dispersion model in some aspects that would allow converging towards the best computational solution through the use of adapted and more suitable empirical correlations that fit specific configurations in order to preserve a close relation to the physical understanding of the phenomena involved in the spray impingement process with the presence of a crossflow. Therefore, the liquid film formation is developed by considering some basic principles (conservation of mass and volume between impinging and adhered parcels) but also an empirical correlation deduced from experimental data [4] for the average film thickness which allows a connection to the phenomenological experience that can easily be fitted to specific settings. This model is incorporated into the code originally proposed by Bai et al. [5] and the performance is recognized through the comparison between the numerical results – with and without the liquid film sub-model – and measurements for two crossflow rates. In addition, the integration of this newly developed computational extension with the spread/splash transition criterion used in this work is also evaluated by comparing the prediction results obtained using two different splashing thresholds: one that takes into account the effect of the film thickness on the disintegration criterion against another that does not.

Figure 1 illustrates the distribution of the relative liquid film thickness over half the impingement wall (a symmetry plane is considered at X=0). The injector is located 0.05m from the inlet with an inclination of 20º, in relation to the vertical plane, in the downstream sense. The figure shows that a thin liquid film forms along the surface but has a particular incidence in the region below the nozzle, which is where the maximum thickness is found. The reduction of the air flow velocity to 5 m/s leads to the thickening of the liquid film and the reduction of the number of the secondary droplets as also reported in the literature.

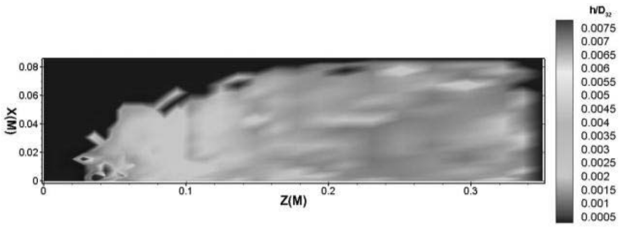


Figure 1: Distribution of the relative liquid film thickness over the impingement wall for a crossflow of 15 m/s.

The use of a transition criterion that does not take into account the effect of the liquid layer does not improve the simulation results. On the other hand, integrating the effect of the liquid film and the transition criterion that considers this effect on the splash threshold clearly enables the enhancement of the model performance.

References

[1]Silva, A., Barata, J., Rodrigues, C., "Influence of Spread/Splash Transition Criterion on the Spray Impingement Modelling", *ILASS-Europe 2011,* Estoril, Portugal*,* September, 2011.

[2]Rodrigues, C., Silva, A., and Barata, J., "Influence of the Energy Dissipation in the Spray Impingement Modeling", *50th AIAA Aerospace Sciences Meeting,* Nashville, Tennessee*,* January, 2012.

[3]Rodrigues, C., Barata, J., and Silva, A., "Dissipative Energy Loss and Influence of an Enhanced Near-Wall Treatment", *11th Int. Conference on Combustion and Energy Utilization*, pp. 1-16, 2012.

[4]Kalantari, D., and Tropea, C., "Spray impact onto flat and rigid walls: Empirical Characterization and Modelling", *International Journal of Multiphase Flow*, pp. 525-544 (2007).

[5]Bai, C.X., Rusche, H., and Gosman, A.D., "Modelling of Gasoline Spray Impingement", *Atomization and Sprays*, Vol. 12, No. 1-3, pp. 1-27 (2002).

* Corresponding author: andre@ubi.pt

Measurement and Simulation of DI Spray Impingements and Film Characteristics

Yi Zheng*, Xingbin Xie, Ming-Chia Lai
Wayne State University, USA
dz7403@wayne.edu

Brad VanDerWege
Ford Motor Co., USA
bvanderw@ford.com

Abstract

Spray impingement of liquid fuel on the combustion chamber wall and piston head in the direct injection engine is difficult to avoid and mostly undesirable, because it delays the gas-phase fuel-air mixture preparation processes and is a possible source for unburned hydrocarbons and soot emissions. This work investigated wall impingement and surface fuel film formation by sprays emerging from a side-mounted six-hole gasoline injector, one of the most dominant direct injection (DI) gasoline engine combustion configuration used today. In the meanwhile, numerical study was conducted for the same operating conditions as the experiments to study the spray behavior and fuel film characteristics.

The spray wetted area, fuel film thickness, and the resultant footprint mass were derived using the Refractive Index Matching (RIM) technique. In this method, the difference in index of refraction between the impinging surface and air results in the scattering of light off the roughened surface, which is modified by the presence of a liquid that closely match the index of refraction of the impingement window. A flat optical ground glass diffuser with polished top surface was placed in the pressurized camber horizontally. The six-hole injector was mounted on the cylinder wall of the chamber with angle of 25 deg to replicate the piston injector orientation of a side-mounted DI Engine. Calibration experiments were carried out to obtain the correlation between fuel film thickness and variation of reflection. A liquid mixture of iso-octane and dodecane with known deposit volume was used for calibration procedure. The fuel was injected on the rough flat window surface at various ambient conditions, injection conditions and distance between the injector tip and window, using the same optical setup. Multi-dimensional computational fluid dynamics (CFD) simulation with selected models of spray validated first for its transport in the air is used to compare the impingement models with the experimental measurements. The Kelvin-Helmholtz/Rayleigh-Taylor (KH-RT) breakup model was used to predict the spray behavior of multi-hole injector. The interaction of liquid drops and solid surfaces is molded using wall film model, which is a hybrid model of assuming individual particle-based quantities and film-based quantities. The numerical simulation of characteristics of spray impingement and fuel film were compared with the experimental results of back-lighting visualizations and RIM method.

The Refractive Index Matching technique was investigated to measure the fuel film thickness of sprays of side-mounted multi-hole injector. As expected, the deposit area of each spray plume is effected by the injection pressure and distance between injector tip and window. The effect of ambient temperature on fuel film thickness is significantly, and the film evaporation rate is strongly affected by the ambient pressure especially at lower temperature. Higher pressure at the same fuel amount tends to reduce film thickness. The CFD simulation models of spray and wall impingement were validated by compared with spray visualization on the free spray transport. The numerical investigation of spray behavior and film characteristics agrees in general with the experimental observations in terms of overall spray shape, tip penetration and wall impingement pattern, and the maximum fuel film thickness.

---

* Corresponding author: dz7403@wayne.edu

Overspray Characteristics and Droplet Density Distribution of Low Pressure Shear Coaxial Injector

I. C. Lee[1], D. H. Kim[2] and J. Y. Koo[*]
Graduate Student, Korea Aerospace University[1,2]
School of Aerospace and Mechanical Engineering,
Korea Aerospace University, Republic of Korea[*]
leeincc@kau.ac.kr, teddyper@kau.ac.kr and jykoo@kau.ac.kr

Abstract

An HVLP (high-velocity-low-pressure) injector configured with a liquid-center and gas-outer nozzle can be extensively used in many industries, including automobile refinishing, wood furniture finishing and so on. The main problem of spray painting is overspray. Overspray produces much VOC (volatile organic compounds) which can cause bronchitis in an operator and waste paint or coating fluids. So, the aim of the present work is to reduce overspray and increase transfer efficiency. An HVLP injector with a gas post of 2-stages are designed and built to conduct spray characteristics as well as to increase transfer efficiency. This efficiency is critical to the spraying and coating from both a cost and an environmental regulatory standpoint. Droplet diameters are measured by laser diffraction methods considering the ratio of refractive indices inside and outside of the liquid sphere.

Experimental studies have been carried out to use the HVLP injector under various momentum ratios and Weber numbers. A higher momentum ratio and Weber number can achieve better atomization characteristics, but it can cause the super-pulsating phenomena which offers bad performance to painting process with film thickness. The experiment results show that the main effect of liquid jet break-up is governed with gas jets of the inner-stage. Additionally, gas jets of the outer-stage also contribute atomization at the far-field spray region and droplet transportation to the object. As gas jets increase, droplet mean diameters decreased and transfer efficiency increased due to the outer gas post which makes an air-curtain near the spray jets so that impinged droplet cannot overspray to the outer region of the spray jets. However, atomized droplets are dropped down to the bottom region due to the gravitational force. As expected, there is a strong dependence of outer gas injection to reduce overspray to the far field. So, the transfer efficiency increases with the effect of the air-curtain.

Keywords: Overspray, HVLP, SMD, Impinging, Transfer efficiency

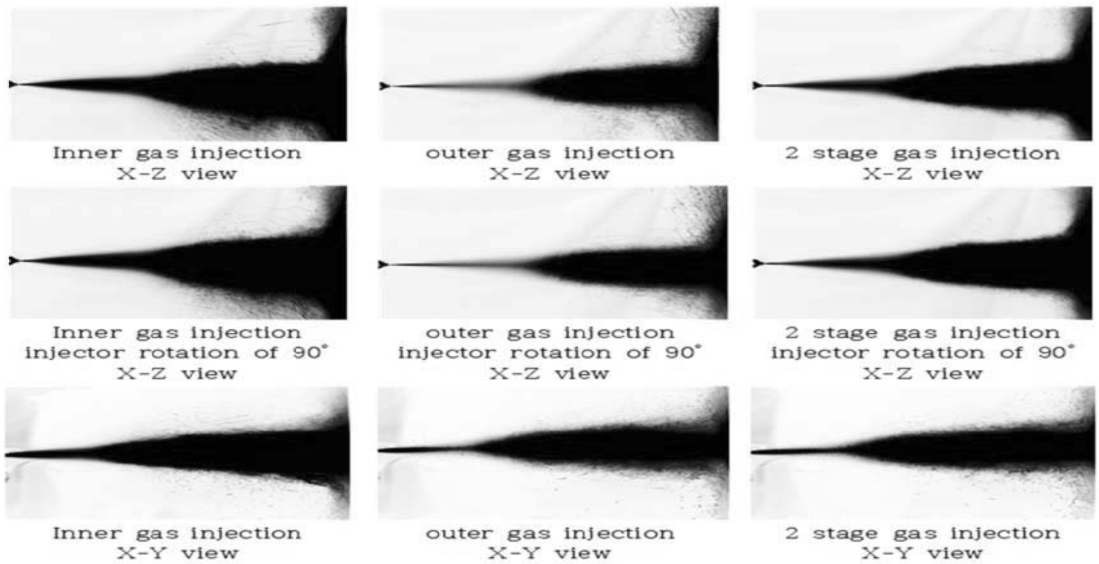


Figure Cross-sectional spray images of inner-stage, outer-stage and 2-stage gas injection

Cavitation Modelling

Y. Meslem[†] , S. Honnet[‡], W. Schwarz[*], J. Reveillon[†] and F.X. Demoulin[†]

[†]UMR6614-CORIA, Technopole du Madrillet, BP 12, Avenue de l'Universite, 76801 Saint-Etienne-du-Rouvray Cedex, France
[‡]PSA Peugeot Citroen, Centre Technique de Velizy, Route de Gisy 78943 Velizy-Villacoublay, France
[*] AVL France, 2-4 rue Hans List F-78290 Croissy sur Seine, France

Abstract

Introduction

During the last decade, the combustion efficiency in Diesel engines has risen by the improvement of injection systems. The increase of injection pressure has been one of the main strategy to reach that goal. Nevertheless, strong local pressure drops have been observed in many Diesel injectors and as a result, cavitation occurs. According to He and Ruiz [2] the dense zone of the spray is influenced by the in-injector flow and particularly by the cavitation phenomenon. Some studies show that the occurrence of cavitation modifies basic characteristics of the spray, which play a major role in the Diesel combustion.

Computational methods

A cavitation model has been developed. As surface density and liquid and gas fractions are the solved variables in the model, this model is able to deal with non spherical gaseous structures, like huge gaseous cavities in Diesel injectors. In case of large gas cavities, a modelling proposal has been suggested, close to the term using Rayleigh-Plesset equation. Only driving pressure term is kept. Indeed, the other terms are linked to bubble sphericity assumption and lose their sense in case of larger gaseous structure or are hard to estimate. Another distinctive feature of the model is to transport interface velocity in order to keep second order time derivative of the bubble radius in the Rayleigh-Plesset equation so as to improve accuracy. Reference conditions issue for cavitation nuclei, that is to say estimation of residual gas pressure and radius of the nuclei, has been raised and a procedure has been proposed. The first step is to identify two zones, "pre-cavitating zone" and cavitating zone, on a first calculation without cavitation modelling. The "pre-cavitating zone" is the zone usptream the cavitating zone and the zone where nuclei are considered to exist. Reference conditions are chosen according to comparisons between critical values identified by Franc [1] and order of magnitude of pressure for the two zones.

Results and Discussion

First results of the model have been presented. Calculations are based on pressure history extracted from a 3D calculation with the CFD solver AVL Fire along a bubble streamline. Response to the pressure drop from gas volume fraction and surface density looks reasonably good. Results are hopeful but will have to be confirmed with 3D reference calculations with an implemented version of the model in the solver AVL Fire.

References

[1] J.P. Franc. The rayleigh-plesset equation: a simple and powerful tool to understand various aspects of cavitation. *In Fluid Dynamics of Cavitation and Cavitating Turbopumps, CISM Courses and Lectures*, 496, 2007.
[2] Lu He and Francisco Ruiz. Effect of cavitation on flow and turbulence in plain orifices for high-speed atomization. *Atomization and Sprays*, 5(6):569–584, 1995.

[*]Corresponding author: yann.meslem@coria.fr

Influence of the needle lift motion on cavitating flow inside Diesel injector

R. Marcer*, C. Audiffren, B. Yerly
[1] Principia, Z.I. Athélia, 13705 La Ciotat FRANCE
richard.marcer@principia.fr, charles.audiffren@principia.fr, benoit.yerly@principia.fr

Abstract

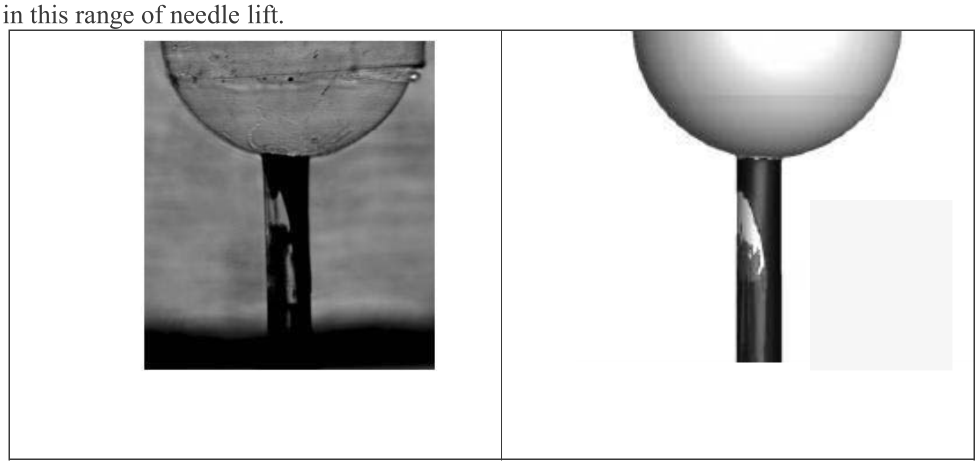
In Diesel injection the flow in the injector, and especially the cavitation development and its influence on the flow rate, is led by the needle lift according to an opening/closing motion cycle. The simulation of that kind of unsteady cavitating flow can be simulated with a CFD model considering a time-varying needle lift. However in this unsteady approach computational time may be large because of the small time steps that are required to represent the cavitation dynamics in the nozzle, especially if a complete needle lift cycle is considered. An interesting possible alternative may be the steady needle lift approach, meaning that the lift is considered as fixed for a given calculation. This approach helps to reduce the simulation time because it allows providing at convergence all the features of flow (including flow rate) for the considered lift, without having to simulate the previous transient opening phase of the lift motion (up to the considered lift). The question is whether the steady approach allows to reproduce the same results as unsteady approach, i.e. for a given needle lift, if simulated flow (flow rate, cavitation) is the same for the steady and unsteady approaches considering that the last one takes into account the dynamic aspects related to the lift motion.

In this study a one nozzle injector with an eccentricity of the hole is concerned. Numerical simulations are performed with both unsteady and steady approaches for the needle lift modeling. Validations of the numerical model based on comparisons with measurements of cavitation carried out at the LMFA [1], are presented.

Figure hereafter shows examples of comparisons of the cavitation pattern issued from experiments and from the computations, at high needle lift. In both cases the cavitation extents all along the hole but in a dissymmetrical way due to the eccentricity of the hole. It has been noted that at high needle lift (> 15% of the maximal lift), the steady approach gives very similar results according to the unsteady approach, for cavitation development as well as for the flow rate. So, at high lift, the motion of the needle has no influence on the flow.

However at lower needle lift (<15% of the maximal lift) cavitation and flow rate obtained with the two methods are significantly different (figure 3). The variable needle lift calculation shows a peak of the flow rate linked with a dynamic pulse of cavitation in the hole, responsible of a peak of liquid velocity in the contracted section. In this range of lift (5%<lift<15%), the steady approach is not able to put into evidence this transient development of the cavitating flow.

Moreover for very small lift (< 5% of the maximal lift) the steady method tends to overpredict the flow rate because dynamic effects related to the needle motion, which are important at such low lift, are not taken into account. Errors up to 20% on flow rate values are thus given by steady approach at low lift which turns out to be inaccurate in this range of needle lift.



Comparison LMFA measurements (left) and CFD (right) of the cavitation (in black) in the hole at high needle lift

[1] Loïc Mées, Marc Michard. Visualisation of cavitation in a transparent diesel injector nozzle. Laboratoire de mécanique des fluides et d'acoustique (LMFA), Ecole Centrale Lyon, CNRS, Université Claude Bernard and INSA Lyon. Private communication.

* richard.marcer@principia.fr

Cavitation Flow in Nozzle of Liquid Injector

Akira Sou*, Raditya Hendra Pratama, Tsuyoshi Tomisaka, Yusuke Kibayashi
Kobe University, Japan
sou@maritime.kobe-u.ac.jp

Abstract

It has been known that cavitation takes place in a nozzle of fuel injectors for internal combustion engines and strongly affects the characteristics of discharged fuel spray. Hence, a large effort has been paid to understand cavitation flow in single nozzles with a symmetric inflow. Cavitation in a nozzle is affected by liquid velocity, geometry of the injector, needle lift, length-to-diameter ratio of the nozzle, curvature of the inlet edge, liquid property, dissolved gases in a liquid, system pressure, and so on. In this study, cavitation flow in various nozzles of 4 mm in width W with an asymmetric inflow, different needle lifts Z (Z/W = 1, 2 and 4), nozzle angles θ (θ = -30, -20, 0, 20, 30 deg.) and various water temperatures T_L (T_L = 285, 305, 326 K) are visualized to clarify the effects of an asymmetric inflow, needle lift, nozzle angle, and fluid property on cavitation flow in a nozzle of liquid injectors.

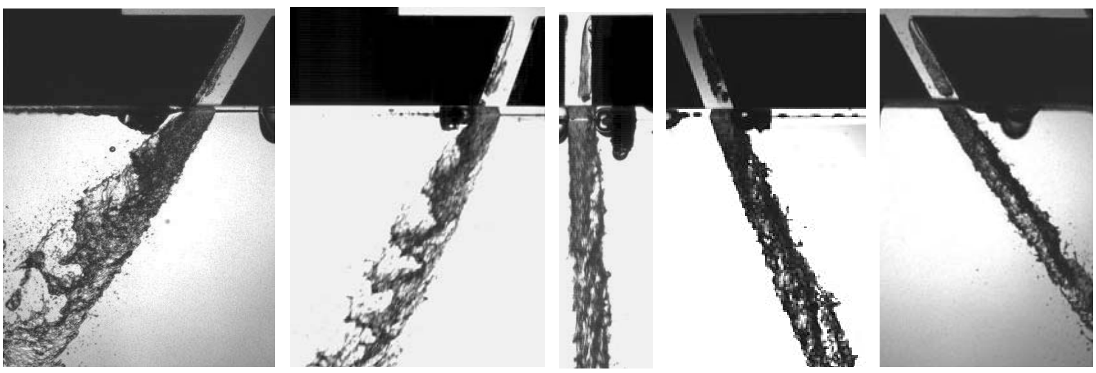
Various two-dimensional (2D) nozzles are manufactured and used to measure the length and thickness of cavitation appearing in the nozzles. Transient behavior of cavitation in the nozzles and the deforming liquid jets are visualized using a high-speed cameras and a metal-halide lamp. The ratio of the length L and the width W of the nozzles are 4.

Figure 1 shows the images of cavitation in tilted nozzles with various nozzle angles θ. Due to the asymmetric inflow mainly from upstream right into the nozzle, a thick separated boundary layer and cavitation appears and grows asymmetrically mainly along the right wall even for θ = 0 degree. The asymmetric and thick cavitation induces an asymmetric and large-scaled deformation of the discharged liquid jet. We also found that the decrease in needle lift Z results in a thicker and longer cavitation as well as a larger deformation of the liquid jet. An acute angle of the nozzle inlet edge is found to enhance the cavitation thickness and its development as well as a large deformation of the discharged liquid jet.

Through the investigation of cavitation flow with different water temperature T_L, we confirmed that cavitation does not depend on the Reynolds number Re in the range of Re > 12500 but strongly on the modified cavitation number σ_C, which is based on local pressure at vena contracta.

Finally, we found that cavitation taking place in a nozzle of liquid injectors is strongly affected by the profile of the separated boundary layer, which depends not on the Reynolds number Re but strongly on an asymmetric inflow, needle lift Z and nozzle angle θ.

These results suggest that by knowing the contraction coefficient C_C we can quantitatively estimate the inception and development of separation-induced-cavitation even for various injector geometries, needle lifts, and fluid properties.



(a) θ = 30 deg (b) θ = 20 deg. (c) θ = 0 deg. (d) θ = -20 deg. (e) θ = -30 deg.

Figure 1 Cavitation in tilted nozzles with various nozzle angles θ and liquid jets (Z/W = 4, W = 4mm).

* Corresponding author: sou@maritime.kobe-u.ac.jp

Study of the influence of internal flow on the spray behavior under cavitating conditions using a transparent nozzle

R. Payri, J. Gimeno*, P. Marti-Aldaravi, O. Venegas
CMT-Motores Térmicos, Universidad Politécnica de Valencia, Spain
rpayri@mot.upv.es, jaigigar@mot.upv.es,
pedmar15@mot.upv.es, and ovenega@mot.upv.es

Abstract

One of the areas of interest in diesel engines is the study of nozzle flow behavior in fuel injectors. As it is well known through previous studies the flow behavior inside the orifice is strongly related to the behavior of the spray at the exit and its interaction with the air, affecting the mixing and combustion process.

In the present work, behavior of the internal flow under cavitating conditions and its influence on the spray behavior is studied. For this purpose, a transparent nozzle plate with a cylindrical orifice is used. The transparent nozzle, made of fused silica SiO2, has an exit diameter of 0.51mm and 1mm length and it is installed in a pressurized rig with fuel (commercial diesel) in order to measure the mass flow and observe the flow inside the orifice, as well as the spray at the exit, using a special visualization technique. The rig consists of a stainless steel vessel, including two opposed optical windows to perform backlighting visualization.

The optical system developed for visualization includes light source with optical fibers, a PCO SensiCam CCD camera and one optical lens, aligned between each other in order to reach high zoom levels. Additionally, an optical diffuser is placed after the light flash exit. This element diffuses the flash light and produces a uniform illumination in the chamber region under study. All images are obtained at a resolution of 1280X1024 pixels and 3mm of window size.

During the initial phase of the work, the hydraulic characterization of the nozzle is done by measuring the mass flow of the nozzle injecting fuel at stationary conditions. For this purpose, the test rig is filled with fuel and the nozzle injects in continuous flow and measuring the mass that escapes the test rig (with a gravimetric balance) for a given time, the stationary mass flow can be calculated. Mass flow is measured for different conditions of injection and discharge pressures. Simultaneously, the visualization of the orifice is carried out in order to observe the cavitation inside the orifice and the bubbles in the spray.

Additional measurements are carried out in the rig pressurizing with nitrogen to observe the spray at the outlet orifice using the same measurement conditions and the same visualization technique. This configuration allows determining the correspondence between the behavior of the flow inside the orifice and the spray at the exit.

The results from the hydraulic characterization show that the beginning of the cavitation (incipient cavitation) occurs before the mass flow collapse. Furthermore, the presence of bubbles in the spray before the mass choke indicates the existence of cavitation.

Finally, the images obtained with the rig pressurized with fuel and with nitrogen are compared to observe the influence of cavitating flow inside the orifice on the development of the spray. It is observed that the spray experiences a significant change in the cone angle once the cavitation is fully developed through the whole orifice.

* Corresponding author: jaigigar@mot.upv.es

Investigations on superheated atomization

A. Günther*, K.-E. Wirth
Institute of Particle Technology
Friedrich Alexander University Erlangen-Nuremberg,
a.guenther@lfg.uni-erlangen.de and k.e.wirth@lfg.uni-erlangen.de

Abstract

Superheated atomization utilizes thermal energy in addition to low pressure energy in order to enhance the disintegration of fluid media. The dispersity of the spray is among other things greatly influenced by evaporation effects. An optical access into the nozzle capillary where a partial evaporation takes place is given by glass nozzles. A shadowgraphy method is applied to analyze the phenomena taking place inside the nozzle. In addition the generated spray is analyzed regarding droplet velocity (Particle Image Velocimetry (PIV)), morphology (Shadowgraphy) and pulsation (acoustical measurements).

Introduction

The additional energy component applied in this process (thermal energy) enhances the dispersity of the product spray in comparison to a pure pressure atomization. [1, 2] Evaporation processes, nozzle geometry, entrainment of surrounding gas phase and process parameters are among the influencing parameters. The resulting spray offers a fine droplet size distribution and moderate droplet velocities without having to apply an additional gas phase. The scope of this work includes the combined investigation of flow phenomenon (mass flux measurements) with evaporation events (shadowgraphy) and the analysis of the resulting process spray (PIV).

Materials and Methods

The superheating of the applied process media is generated in a pressure vessel via heating jackets. The pressure is introduced by pressurized air. The investigated nozzle is located in the lower part of the vessel. The spraying process is initiated by opening a hand valve. The mass flux is continuously measured by a weighing method. The optical analysis of the spray is carried out using PIV and Shadowgraphy. The last named includes the application of a Long-Range-Microscope in order to allow the investigation of micro-processes. Acoustic measurements are done by applying a microphone.

Results and Discussion

Shadowgraphy pictures of the flow capillary of glass nozzles have been captured and analyzed regarding the location of the first evaporation effects. The higher the process temperature the sooner the evaporation starts since the driving force (vapor pressure) for vapor generation is enhanced. The increase of pressure yields a reversed trend. The more pressure energy is applied the later vapor bubbles are generated. The retardation time inside the nozzle capillary is reduced since most of the pressure energy is converted into kinetic energy resulting in a lower growth time for vapor bubbles. In addition, the possibility to reach the vapor pressure inside the nozzle capillary and by this allowing evaporation to happen is reduced. The generated spray morphology is mostly dependent on the applied temperature rather than the pressure. By enhancing the thermal energy input the spray angle is widening, the droplet velocity profile flattens and smaller droplets are produced. An important topic regarding spray events is pulsation. In this work inhomogeneities in the spray over time are investigated mainly by applying an acoustic system. An enhanced process temperature yields a higher pulsation frequency since more vapor bubbles are contained in the liquid which burst when they come into contact with the surrounding.

Acknowledgement

Special thanks go to the Deutschen Forschungsgesellschaft (DFG) for the financial support of the research project within the priority program “Process sprays”.

References

[1] Sher, E., Bar-Kohany, T., Rashkovan, A., *Progress in Energy and Combustion Science*, 34: 417-439 (2008)
[2] Razzaghi M., *Nuclear Engineering and Design,* 114: 115-124 (1989)

* Corresponding author: a.guenther@lfg.uni-erlangen.de

TiO_2 Nanoparticle Production with Flame Synthesis Method by using Flash Boiling Spray - Relation between Injection Conditions and Nanoparticle Physical Properties -

M.Oshima[*1], M. Matsushita[2], H. Miyamoto[2], D. Fukushima[2], J. Senda[3], K. Ishida[4]
1: Fkui University of Technology, 3-6-1, Gakuen, Fukui city 910-8505, Japan
2: Doshisha University Graduate School, 1-3, Miyakodani, Tatara, Kyotanabe city 610-0321, Japan
3: Doshisha University, 1-3, Miyakodani, Tatara, Kyotanabe city 610-0321, Japan
4: HORIBA, Ltd., 2 Miyanohigashi, Kissyoin, Minami-ku, Kyoto city 601-8510, Japan
m-oshima@fukui-ut.ac.jp, dtl0354@mail4.doshisha.ac.jp, dtk0337@mail4.doshisha.ac.jp, jsenda@mail.doshisha.ac.jp and kozo.ishida@horiba.com

Abstract

The authors proposed a novel nano size particle production system, Flashing Spray Flame Nanoparticle Synthesis Method, by using flash boiling spray. In this method, the mixed solution is used to improve the vaporization characteristics of the precursor. The mixed solution is supplied directly to the chamber as flash boiling spray by an injector. The nanoparticles are produced by flame thermal energy. In this paper, we investigated the relation between TiO_2 nanoparticle physical properties and injection conditions. As a result, TiO_2 particle was agglomerated by decreasing injection frequency.

* Corresponding author: m-oshima@fukui-ut.ac.jp

Use of computational modelling for investigation the Effect of melt delivery nozzle tip length on gas flow separation in supersonic gas atomization

Shahed Motaman *, Andy Mullis, Duncon Borman, Bob Cochrane
Institute of Materials Reserch (IMR), University of Leeds, United kingdom,
pmsmo@leeds.ac.edu and a.m.mullis@leeds.ac.uk

Abstract

In this paper, the effect of different melt delivery nozzle tip length on gas flow separation for annular-slit gas atomization, in gas-only flow, was numerically simulated by solving the compressible Navier-Stokes equations. Gas flow separation in the vicinity of the melt delivery nozzle during operation of the supersonic gas atomizer causes a back-flow of melt from the melt delivery nozzle tip along its outer surface, leading to very poor atomization performance and finally will result in aborting of the run. The melt delivery nozzle tip length plays a crucial role in preventing this problem during operation. Four different melt delivery nozzles with the tip length of, 10, 8, 7 and 3mm were numerically modeled in the confined-feed annular slit geometry with gas pressures of 0.5, 1, 1.5, 2, 2.5,3 and 4MPa. The results indicate that the nozzles with 8 and 10mm melt tip length are very sensitive to flow separation even in at a low gas pressure of 1MPa. With increasing atomization gas pressure the flow separation moved forward to the melt nozzle tip and at the gas pressures of 3MPa and above, flow separation was completely suppressed. In addition, no flow separation was seen on the two other melt nozzles at any gas pressure. These results specify that the flow separation occurrence is a function of melt delivery nozzle tip length and atomization gas pressure.

---

* Corresponding author: pmsmo@leeds.ac.uk

Numerical modelling of liquid jets atomisation due to leakage of liquefied gas storage

Cedric LEMOFACK[a,b,*], J.M. LACOME[b], J. REVEILLON[a], F.X. DEMOULIN[a]
[a] CNRS, UMR6614-CORIA, Technopole du Madrillet, BP 12, Avenue de l'Universite, 76801 Saint-Etienne-du-Rouvray Cedex, France
[b]INERIS, Parc Technologique ALATA, B.P. N°2, 60550 Verneuil-en-Halatte, France
lemofacc@coria.fr, demoulin@coria.fr, Jean-Marc.LACOME@ineris.fr, Julien.Reveillon@coria.fr

Abstract

The objective of this work is to develop a numerical model with the aim of simulating a two phase jet resulting from a breach or leakage in a vessel or pipe containing liquefied gas. This jet consists in three areas: the expansion zone (flash boiling, atomization), the entrainment zone (secondary break-up, droplets evaporation) and the final dispersion zone.

Among the complex phenomena involved during the injection, this work focus on the effect of the vaporization and boiling process in the jet. To represent the flash boiling phenomena that occurs at the exit of the injector up to the end of expansion zone, the Homogeneous Equilibrium Model (HEM) have been used. Consequently, gas and liquid velocities are identical at the beginning of the entrainment zone. An issue of this approach is the determination of the velocity induced by the pressure drop inside the injector and by the flash effect. The maximum kinetic energy can be estimated by the pressure drop and the variation of thermodynamic energy in the flow. Here, the velocity is considered to be driven mainly by the pressure drop, thus it is determined by the Bernoulli law. After the expansion zone, the spray is supposed to be at boiling temperature with a gas environment composed only by vapour. Thus, any increase of temperature will promotes boiling. But if some air diffuses inside the spray, a non classical vaporization process is expected. Thermodynamic conditions are at these boundaries in the beginning of a region called entrainment zone. To determine the evolution of the spray, vaporization or boiling, a special model is requested.

A thermodynamic equilibrium vaporization model were developed. During a time step, it is considered a thermodynamic system composed by a liquid droplet film and surrounding gas film. We consider that this film reaches the thermodynamic equilibrium at the end of the time step. Finally, the remaining liquid in the film at the equilibrium temperature is mixed with the droplet part not used in the film. The same is done with the gas phase. This procedure ensure a realisable thermodynamic state after the complex vaporization process.

To test the model, an atmospheric two-phase jet of butane, emanating from a circular orifice is considered. The Euler-Lagrange approach has been used for this two-phase simulation. A co-flow of air surrounds the spray injection. The mass fraction of vapour and liquid is given by the HEM model. Finally, the turbulence is modelled with the $k - \epsilon$ model.

It is complicated to have experimental results on the types of jets described above. But the INERIS were able to have more reliable measurements in temperature. The spray is surrounded by a hot gas environment. However, the temperature of the spray decreases up to a certain distance. Due to the initial droplet boiling temperature, close to the injection, the vaporization process dominates the flow. Since evaporation is an endothermic phenomenon, the spray jet cools down until there is no droplet enough. Thus the spray temperature rises only once the liquid vaporization does not have enough influence in the flow.

The modelling results show that the calculated temperature behaviour in the spray jet by comparison with the observations is generally satisfactory. This result cannot be obtained with classical vaporization model. A perspective to this work can be to add more accurate models to the internal flow of material in vessel and to the flash boiling and atomization phenomena in the expansion zone.

---

*Corresponding author: lemofacc@coria.fr

A steady-state Eulerian-Lagrangian solver for non-reactive sprays

A. Andreini, C. Bianchini, B. Facchini, A. Giusti*
Dipartimento di Energetica Sergio Stecco, University of Florence, Italy
andrea.giusti@htc.de.unifi.it

Abstract

A steady-state RANS solver for sprays, based on fully coupled Eulerian-Lagrangian approach, was implemented in the framework of the open source CFD code called OpenFOAM. Because of the intrinsic unsteady nature of the Lagrangian tracking, the most straightforward approach to solve for droplet motion in continuous media is the fully unsteady method. However, since for many industrial purposes the focus is on the time averaged solution, steady state acceleration techniques are of primary interest.

The computational strategy, implemented to handle typical configurations encountered in liquid-fueled gas turbine combustors such as pressure swirler atomizers, consists in separately resolving the gas and the liquid phases exploiting respectively a standard steady-state pressure based solver and a pseudo-transient Lagrangian approach. The coupling between the two phases is achieved by means of time averaged source terms in the Eulerian conservation equations and by including gas-droplet interaction in the Lagrangian tracking to account for evaporation, drag, break-up and turbulent dispersion. The Lagrangian tracking is performed using a statistically representative number of parcels which are tracked over their entire history from injection up to either evaporation or computational domain exiting. Pseudo time integration employs a user-defined time-step chosen to obtain an optimal fuel mass injection to suitably represent droplet distribution. The obtained instantaneous solutions are then summed up to compute the Lagrangian solution and hence the steady-state spray-gas coupling source terms. In such a way the number of parcels necessary to describe the spray evolution, as well as the computational time to resolve the Lagrangian tracking of the liquid particles, is strongly reduced compared to standard unsteady approaches. In order to improve the stability of the coupling source terms, an averaging procedure among successive Lagrangian solutions is proposed instead of the classical approach which only considers the last available Lagrangian solution. Such averaging is performed using a moving average approach which only accounts for a given number of Lagrangian solutions. This technique allows us to increase the number of parcels considered in source term computation without increasing the computational efforts and to smooth out possible oscillations of the Lagrangian solution.

This paper describes the derivation and implementation of such numerical methods. Solver validation was performed against experimental data available from two well known literature test cases. The first one consists of an isothermal swirled flow with solid particles axially injected at the center of the vortical structure. In this case, the Eulerian-Lagrangian coupling is reduced to flow field interactions, thus only drag, particle dispersion caused by turbulence and turbulence generated by particle motion are to be considered. The solution obtained with the proposed coupling approach was compared with experimental measurements and numerical results obtained using classical approaches: detailed results in terms of gas phase velocity field, particle velocity and particle size-velocity correlation are presented. The second test case is an isopropyl alcohol spray generated by a hollow cone injector issuing into a co-flowing heated air stream. In this case the Eulerian-Lagrangian coupling is also extended to the mass and energy conservation equations and the convergence capabilities of the different approaches are further tested. Comparisons with experiments for both continuous and disperse phase are reported.

The newly implemented solver showed faster convergence rates compared to numerical approaches already available in the code, maintaining equivalent capabilities for mean flow field and particle distribution prediction.

---

*Corresponding author: andrea.giusti@htc.de.unifi.it

A High Resolution Study of Non-Reacting Fuel Sprays using Large-Eddy Simulations

A. Wehrfritz*, V. Vuorinen, O. Kaario, M. Larmi
Department of Energy Technology, Aalto University, Finland
armin.wehrfritz@aalto.fi

Abstract

Introduction The present study deals with Large-Eddy Simulations (LES) of non-reacting fuel sprays in internal combustion engine like conditions. The motivation for the research is to gain a deeper understanding of high resolution spray simulations using the implicit Large-Eddy Simulation (iLES) approach in combination with Lagrangian Particle Tracking (LPT). The main objective is to investigate the influence of the grid size on droplet breakup and evaporation in conjunction with the implicit LES method for a well defined reference case. The chosen spray case is corresponding to the “Spray A” conditions specified within the Engine Combustion Network (ECN). This test case is experimentally investigated by several research institutes and the data is freely available through the ECN. The ambient gas density and temperature for Spray A are 22.8 kg/m3 and 900 K, respectively. The injected fuel is n-dodecane and the injection pressure is 150 MPa. The nominal nozzle diameter of the injector is 90 μm, which leads to an averaged injection velocity of approximately 590 m/s.

Methods The simulations are carried out with the open source CFD tool box OpenFOAM. A compressible flow solver is used and the pressure treatment is handled by the PISO-algorithm. The accuracy is formally second order in space and an implicit, 2nd order accurate time integration is used. For the turbulence modeling the implicit LES approach is applied, hence no explicit sub-grid scale model for the unresolved turbulent scales is used. Within this method the grid, or more precisely the numerical discretisation scheme, functions as an implicit low-pass filter and it is assumed that sub-filter scales will dissipate in the same manner as the numerical scheme. The liquid phase is modeled by droplets and hence LPT is used to compute the droplet motion. The box sampling method is used to determine the initial droplet location and the initial droplet size is obtained from a Rosin-Rammler distribution. Heat transfer and evaporation are modeled according to Ranz/ Marshall and Frössling correlations. Two droplet breakup models are compared in this study: the Enhanced Taylor Analogy Breakup (ETAB) and the Kelvin-Helmholtz Rayleigh-Taylor model as proposed by Reitz (1988).

Computational domain A fully hexahedral base mesh is used, which is refined by a 2:1 cell-splitting approach in the spray region in order to obtain appropriate cell sizes for iLES. The smallest cell size in this study is 62.5 μm, i.e. approximately $\frac{2}{3}$ of the nozzle diameter. The time step is in the order of 10^{-8}s due to the small cell size and droplet time scale.

Results and Conclusion Visualizations of the flow field show that the methods used are suitable to resolve the characteristic flow structure (e.g. Kelvin-Helmholtz vortices). Comparisons to the experimental data show a reasonable agreement with respect to liquid and vapor penetration for the two finest mesh resolutions. Concerning the mesh resolution it was found that a cell size of 125 μm was sufficient to capture the turbulent motion of the spray. Both investigated breakup models performed reasonable well, even though differences in predicted droplet size were found. The differences in droplet size affect again the global spray quantities (liquid/ vapor penetration). Hence, a further investigation and tuning of the breakup model parameter is suggested for the chosen LES approach.

---

*Corresponding author: armin.wehrfritz@aalto.fi

Numerical and experimental investigation of the optical connectivity technique in cross flow atomization

G. Charalampous, C. Hadjiyiannis, Y. Hardalupas*
Mechanical Engineering Department, Imperial College London, London, SW7 2AZ, UK
georgios.charalampous@imperial.ac.uk, c.hadjiyiannis09@imperial.ac.uk and
y.hardalupas@imperial.ac.uk

Abstract

The optical connectivity technique has been proposed for the characterization of the morphology of continuous liquid jets before breakup. The technique is based on internal illumination of a liquid jet by a laser beam through the spray nozzle. The liquid jet acts as a light guide and the laser beam propagates along the length of the jet in the same way that light travels along the length of an optical fibre. The laser beam excites a fluorescent dye that is dissolved in the liquid jet, making the volume of the liquid jet luminous. However, unlike an optical fibre, there are laser beam intensity losses along the length of the liquid stream due to refraction at the liquid/gas interface and due to absorption by the fluorescence dye. While the technique has been shown to work well in 'straight' jets, for liquid jets exposed to a cross stream of air, where the liquid jet becomes gradually inclined relative to the axis of the jet exit, laser light losses due to refraction through the liquid interface may increase and lead to limitations of the technique. A numerical and experimental investigation of the performance of the optical connectivity technique is conducted for liquid jets exposed to a cross stream of air. The numerical investigation revealed that the fluorescent intensity profiles along the liquid column length are highly sensitive to the divergence of the illuminating laser beam. In addition, the wavelength and deflection of the jet had a significant effect on the fluorescent intensity profiles only for a collimated illuminating laser beam. The experimental investigation showed that the fluorescent intensity is not uniformly distributed throughout the volume of the jet but a fluorescent intensity maximum exists at the point of maximum jet inflection. The rate of decay of the fluorescent intensity along the length of the jet is similar among the jets regardless of the length of the jet, which suggests that after the jet maximum inflection, there are considerable losses of the illuminating laser beam intensity due to refraction.

Key words: Optical Connectivity, Diagnostics, Laser-induced Fluorescence, Geometrical optics

* Corresponding author: y.hardalupas@imperial.ac.uk,

Application of Detached Eddy Simulation to Lagrangian Spray Simulations

P. Pischke*, D. Christ, D. Cordes, R. Kneer
Institute of Heat and Mass Transfer, RWTH Aachen University, Germany
pischke@wsa.rwth-aachen.de and kneer@wsa.rwth-aachen.de

Abstract

Lagrangian simulations of liquid sprays involve models for many small scale processes, including primary and secondary breakup or droplet collisions in the liquid phase, and turbulence in the gas phase. For turbulence modelling, two classes of models have evolved, first, turbulence models based on Reynolds Averaging of the Navier Stokes equations (RANS models), and more recently spatially averaging Large Eddy Simulations (LES). Most common RANS models are k-epsilon- and k-omega-models, which describe turbulence in terms of turbulent kinetic energy and the (specific) turbulent dissipation rate. LES models are distinguished by the subgrid-scale-model (SGS model) involved. Studies by other authors have shown that the choice of the turbulence model has major influence on the accuracy of Lagrangian spray simulations.

While most liquid phase models are based on a RANS description of the gas phase, RANS turbulence models are not capable to capture vortex creation and break down in highly unsteady free jets. In turn, LES can describe free jet turbulence very well, but does not link to many liquid phase models properly, as it does not provide time-averaged quantities. This trade-off can be overcome by hybrid turbulence models, such as Detached Eddy Simulation (DES). Contrary to classical hybrid LES-RANS models, which decompose the domain into dedicated LES and RANS regions, DES is basically a LES approach with RANS for subgrid-scale-modelling, i.e. the turbulent transport equations are solved for the entire domain. When combined with a k-epsilon-model, DES can provide the turbulent quantities required by the liquid phase models, while maintaining LES behavior in the free flow.

In this study, the application of DES to Lagrangian spray simulations is assessed by example of a hollow-cone spray as is found in gasoline direct injection engines. The DES model is coupled to a realizable k-epsilon model, and compared to LES and RANS simulations of the same case (including standard k-epsilon, RNG-k-epsilon and realizable k-epsilon models). For all turbulence models, a mesh sensitivity analysis is performed. The influence of turbulence modeling on the spray simulation is evaluated by macroscopic properties such as liquid penetration, vortices formed etc., and by microscopic properties and turbulent quantities, such as turbulent kinetic energy (for the DES and RANS models) and dissipation rates.

Preliminary results indicate that DES can handle free jet turbulence better than the RANS models tested. The DES approach with realizable k-epsilon model shows low diffusivity of turbulent quantities (in particular when compared to the realizable k-epsilon RANS simulation, which is known for its overly diffusive behaviour), and turbulent dissipation is reduced. For the spray simulation, this leads to increased liquid phase penetration, and to a more destinct prediction of vortex structures, which are in many cases dissipated by RANS models. In contrast to the LES simulation, turbulent fluctuations can be derived from turbulent kinetic energy, leading to a more realistic prediction of turbulent interaction and turbulent dispersion. The prediction of turbulent dispersion is crucial to the formation of droplet recirculation regions, which is a known issue when coupling Lagrangian spray simulations to LES simulations. As assumed, the DES simulation appears to combine the benefits of LES and RANS simulations.

Despite the good results, the combination of DES with Lagrangian spray simulation requires further validation by synthetic validation cases. This study has shown that it might be worth the effort.

*Corresponding author: pischke@wsa.rwth-aachen.de

Effect of Diffusion Interactions between Droplets on Gas Absorption of Highly Soluble Gases in Sprays and Clusters

T. Elperin, A. Fominykh and B. Krasovitov*
Department of Mechanical Engineering
The Pearlstone Center for Aeronautical Engineering Studies
Ben-Gurion University of the Negev, P. O. B. 653, 84105, Israel
elperin@bgu.ac.il, fominykh@bgu.ac.il and borisk@bgu.ac.il

Abstract

Industrial and naturally occurring absorption air pollutants removal processes include wet removal of moderately and highly soluble gases by dispersed liquid phase in various types of packed and spray towers and in-cloud scavenging of highly soluble gases. In this connection the study of gas absorption by clusters of droplets is of great interest for environmental and chemical engineering.

In this study we describe a method that takes into account diffusion interactions between droplets caused by the overlap of the depleted of soluble gas regions around the neighboring droplets. The suggested method employs a cellular model of a gas-droplet suspension whereby a suspension is viewed as a periodic structure consisting of the identical spherical cells with periodic boundary conditions at the cell boundary. It is assumed that gas absorption by cluster of droplets is accompanied by the subsequent aqueous-phase equilibrium dissociation reaction. We showed that scavenging of highly soluble trace gas in a cluster of droplets is described by a system of transient diffusion equations with the corresponding initial and boundary conditions at the droplet center, droplet surface and at the boundary of the cell. The initial boundary value problem was solved using the method of lines and Monte Carlo simulations. Using the proposed model we determined temporal and spatial dependencies of the concentration of the soluble trace gas in a gaseous phase and in a droplet and calculated the rate of gas absorption. It is showed that absorption of highly soluble gases by clusters of droplets leads to a significant decrease of soluble trace gas concentration in the interstitial gas. We found that scavenging coefficient for gas absorption by cloud droplets remains constant and sharply decreases only at the final stage of absorption. This assertion implies the exponential time decay of the average concentration of the soluble trace gas in the gaseous phase and can be used for the parameterization of gas scavenging by cloud droplets in the atmospheric transport modeling.

In the calculations we employed gamma size distributions of droplets. We performed our calculations for several soluble gases such as hydrogen peroxide (H_2O_2) and nitric acid (HNO_3). It was shown that despite of the comparable values of Henry's law constants for the hydrogen peroxide and the nitric acid, the nitric acid is absorbed more effectively by cluster of droplets than the hydrogen peroxide due to a major affect of the dissociation reaction on nitric acid scavenging. We obtained also the analytical expressions for the "equilibrium values" of concentration of the active gas in a gaseous phase and for the total concentration in the liquid phase for the case of the hydrogen peroxide and nitric acid absorption by cloud droplets.

The results of the present study can be useful in an analysis of different meteorology-chemistry models and in particular in various parameterizations of the in-cloud scavenging of the atmospheric soluble gases. Analysis of the diffusion interaction between droplets in stagnant clusters provides a fundamental knowledge of the processes that occur in atomization and spray systems.

The developed cell model of highly soluble gases absorption or parameterizations based on the obtained results can be easily integrated into online coupled meteorology-chemistry or climate-chemistry models, where the cloud processes and chemical transformation of atmospheric pollutants are considered together with two-way interactions as well as it can be used for the modeling of gas absorption by mist formed in gas streams inside various types of packed and spray towers.

---

* Corresponding author: borisk@bgu.ac.il

Combustion of aerosols: droplet sizing study in microgravity

Mouhannad NASSOURI[1*], Christian CHAUVEAU[1], Fabien HALTER[2], Iskender GÖKALP[1]

[1] Institut de Combustion, Aérothermique, Réactivité et Environnement (ICARE), CNRS-INSIS, Orléans, France
[2] Laboratoire PRISME, University of Orléans, Orléans, France

Abstract

Spray and aerosol cloud combustion accounts for 25% of the world's energy use, and yet it remains poorly understood from both a fundamental and a practical perspective. Realistic sprays include a liquid breakup region, dispersed multiphase flow, turbulent mixing processes, and various levels of flame interactions through the spray. Idealization of spray configurations in a quiescent environment (the starting point for models) has been impossible in 1 g due to the settling of large droplets and buoyant pluming of post combustion gases. The objective of the present research is the experimental determination of the flame propagation velocity in aerosols. The first step of this work was to characterize experimentally the size of droplets composing the aerosol. This characterization was performed using a laser diffraction particle size analyzer "Sympatec-HELOS", using ethanol and isooctane as fuel. High pressure studies have been performed using of a high pressure combustion chamber (max pressure 12 MPa). This equipment was designed to be used aboard the Airbus A300-0g of the CNES. The interest of microgravity in this study is related to the possibility of creating aerosols without sedimentation effects. After ground tests, three parabolic flight campaigns were conducted. The results obtained allowed to determine the effects of initial temperature and pressure on the droplet diameter distribution of the aerosol. A systematic comparative analysis for identical initial conditions in normal and reduced gravity was performed, also between ethanol and isooctane droplets. During the experiments, the temporal evolution of the growth of droplet sizes was followed starting from the beginning of the condensation of expansion cooling. The durations of growth and evaporation were also measured.

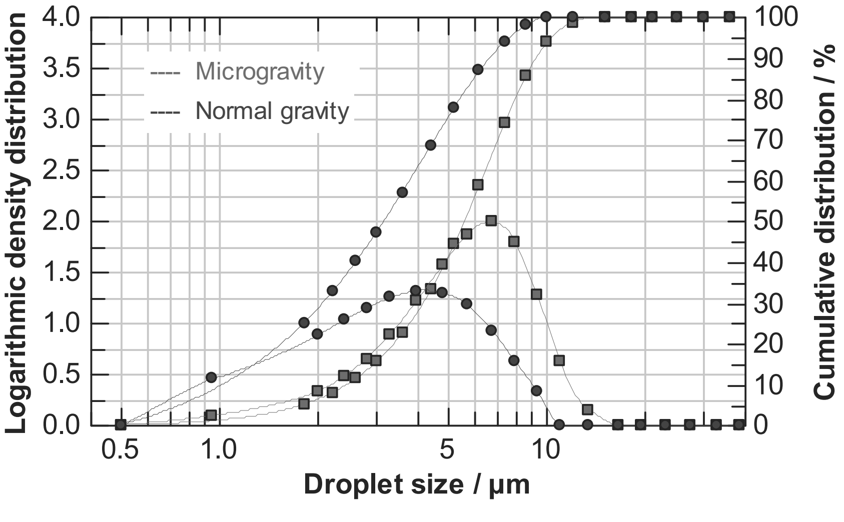


Figure: Ethanol droplets size distribution at 45 ° C, 8 bars and Ø=1 in microgravity --■-- and normal gravity --●--

* Corresponding author: mouhannad.nassouri@cnrs-orleans.fr

Influence of air pressure on bubble entrapment in drop impact onto solid surfaces

J. Palacios, P. Gómez, J. Hernández*
Dept. de Mecánica, E.T.S. de Ingenieros Industriales, UNED, Madrid, Spain

Abstract

An experimental study of the impact of glycerol/water drops onto a dry glass surface at different ambient pressures is presented. During the impact of a high-viscosity liquid drop onto a solid surface, air can be entrapped in the form of bubbles of different sizes. Depending on the impact conditions, the bubbles are distributed forming several patterns. In a previous work on drop impact, we observed, for sufficiently large capillary numbers, the existence of a ring of micro-bubbles, delimiting an outer cloud of bubbles. This outer ring, along with the smaller ring surrounding the entrapped central bubble, were characterized for wide ranges of impact conditions. In the present work, we investigate the influence of the ambient air pressure, which in previous works has been found to be a relevant parameter for splashing inception, on the formation of the bubble patterns. Investigating the dependence of the rings size on liquid viscosity and ambient pressure may help assess different possible scenarios occurring during the impact process. We carried out new experiments of impacts of drops of a glycerol/water mixture in a vacuum chamber that allowed the ambient air pressure to vary between 20 and 100 kPa. A high-speed digital camera was used to capture the images of the impact at a rate of about 63 000 fps, with a shutter speed varying from 1 to 13.75 μm, an image size of 256×128 pixels and a typical resolution of 15 to 25 pixel/mm. The images were captured from below, with back lighting provided by a 575 W metal halide lamp, without a diffuser placed between the lamp and the drop impact area. This method, although not appropriate to visualize the spreading of the lamella and entrapment of bubbles in detail, indirectly reveals some flow features that may shed some light on the phenomena involved in bubble entrapment and help determine the instant at which the lamella separates from the solid surface. Attention is mainly focused on the existence and evolution of the observed rings of micro-bubbles. We propose correlations for the size of the rings of micro-bubbles measured in impacts of drops of three different glycerol/water mixtures at atmospheric pressure and impacts of 3:2 (v/v) glycerol/water drops at different ambient pressures, as a function of the relevant parameters. The dependence on the impact Reynolds number and ambient pressure of the critical Stokes number at which an abrupt jump in the size of the outer ring of micro-bubbles occurs is discussed. Some authors have proposed the hypothesis that the liquid sheet might originate as a result of the interaction of the drop liquid with the intervening gas layer, before the drop contacts the solid surface. However, some findings of the present work do not point to the persistence of a thin air layer beneath the spreading drop at the time a splash is produced. We have observed effects of the liquid viscosity on the location of the micro-bubble rings, which suggests an extensive liquid-solid contact before a liquid sheet is ejected. On the other hand, it has been observed that the size of the inner ring of micro-bubbles increases with the Stokes number, following a dependence law that compares reasonably well with the prediction made by other authors for the radius at which sheet ejection (accompanied by a breakdown in the continuum theory) occurs, which suggests that the model proposed by these authors is adequate at least for the early stages of impact. It has also been found that the size of the outer ring of micro-bubbles increases with decreasing ambient pressure, and that the influence is higher for Stokes numbers smaller than the critical value where the jump in the radius of this outer ring occurs. Another finding is the slight dependence on the ambient pressure of the critical Stokes number, which increases with decreasing ambient pressure (i.e., the jump occurs at smaller velocities when the ambient pressure is reduced). These trends are consistent with the known fact that a reduction in the ambient pressure tends to suppress splashing.

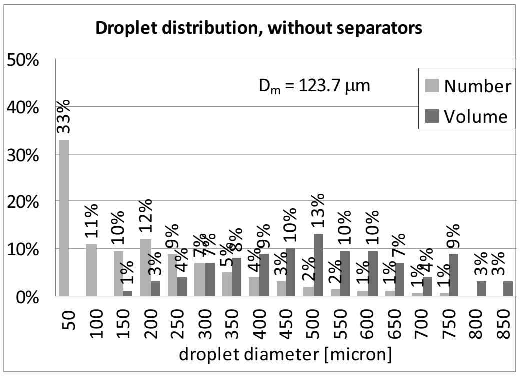
*Corresponding author: jhernandez@ind.uned.es

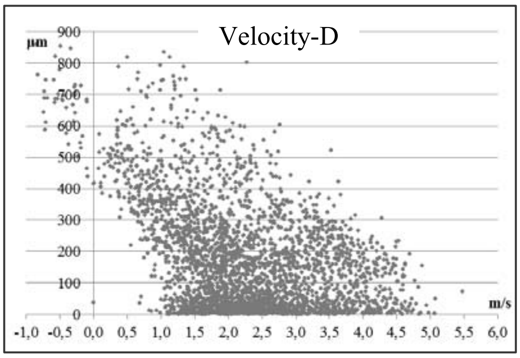
Droplet separators for evaporative towers: efficiency estimation by PDA

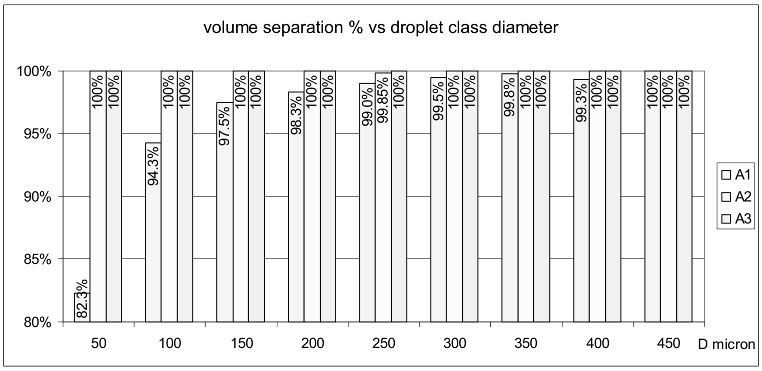
L. Araneo
Department of Energy, Politecnico di Milano, Italy
lucio.araneo@polimi.it

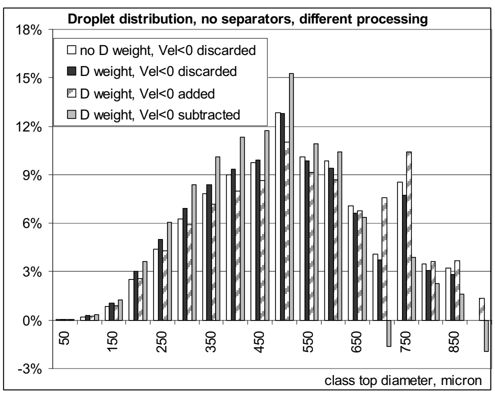
Abstract

A PDA system has been set up to characterize the behavior of different arrangements of static panels that are used as drift eliminators to intercept and remove residual water droplets entrained in the hot air flow released by an evaporative tower. The tested tower has a square section of 60x60 cm, that is the standard size for modular separation filters; it was tested while working with or without the droplet separators to allow comparisons. The residual water droplets entrained by the air flow and expulsed by the tower are measured few centimeters above the tower exhaust and characterized by the PDA system; the droplet population main parameters (number, velocity, mean diameters) can be calculated. The top chart on the right shows the droplet distribution measured over the whole exit section, binned per diameter classes. The percentage of droplets contained in each bin is calculated both respect to the total number and to the total volume of the detected droplet. The velocity-diameter plot of the same droplets shows other aspect of the population. The velocity spread reflect the high turbulence of the exhaust air flow. Larger droplet have smaller velocity, since the gravitational downward force is not negligible compared to the aerodynamic upward drag from the air flow. Few very large droplets have negative velocity: they are interpreted as droplet that after expulsion, are falling back downward. The use of an LDV-PDA allows to detect such droplets and to discard them from the efficiency calculations. The separation efficiency can be calculated by direct comparison of the number of water droplets that are detected in the exhaust air flow, both globally and for specific classes of droplet diameters. Global results can be calculated on the basis of the number or of the volume of the droplet. The test can be repeated with different arrangements; in the chart on the right it is reported the effect of using up to three layer of droplet separators to increase the separation efficiency. The accuracy of the efficiency estimation is also studied. Two main aspects are considered. The first is that the PDA measurement volume dimension changes with the detected droplet size: it should have negligible effect on the result per classes of diameters, but the effect on the global efficiency is present and will be discussed. The second aspect is the presence of droplet that are falling downward in the upward air flow: their presence should be considered and corrected for. The results of this paper are useful when comparing them to other measurements obtained with techniques that are not able to detect the droplet velocity.









A Spray Interaction Model with Application to Surface Film Wetting

K. V. Meredith*, X. Zhou
Research Division, FM Global, USA
karl.meredith@fmglobal.com and xiangyang.zhou@fmglobal.com

A. Heather
OpenCFD Ltd., UK
a.heather@opencfd.co.uk

Abstract

In a fire suppression environment, sprinkler spray wets solid surfaces via droplet impingement. A framework for modeling the interaction of sprays with solid surfaces, with the intention of simulating fire suppression, has been developed. The spray interaction model includes impingement, splashing, and subsequent liquid-film transport over solid surfaces. The thin film-transport equations for mass continuity, momentum, and energy form the basis for the spray-film interaction model. The model has been implemented in OpenFOAM®† and subsequently coupled with a fire growth model (FireFOAM). Experimental measurements of splashing behavior are shown and used for model validation. Model validation is demonstrated for droplets impinging on wet corrugated cardboard surfaces over a range of Weber numbers. Comparisons include the splashed mass measured as a function of radial distance from the initial impingement location.

*Corresponding author: karl.meredith@fmglobal.com

†OpenFOAM®is a registered trademark of the OpenFOAM Foundation.

Controlled spray quenching in heat treatment process

B. Hinrichs[1], N. Hornig, S. Schüttenberg, U. Fritsching
Process & Chemical Engineering, Foundation Institute for Materials Science (IWT),
Badgasteiner Str. 3, Bremen, Germany
b.hinrichs@iwt.uni-bremen.de and hornig@iwt.uni-bremen.de

Abstract

Spray quenching is a method for precise control of material and workpiece properties of metallic specimen in heat treatment processes. Specific material microstructure distributions in combination with the avoidance of workpiece distortion in heat treatment processes may be realized by impressing precisely controlled cooling with flexible flow fields based on multiple spray nozzle arrangements. This controlled liquid quenching by means of two-phase spray cooling enables the possibility to generate specific local heat transfer conditions on workpiece surfaces, which builds the basis for asymmetric quenching strategies for control of workpiece properties and reducing distortion.

The quenching process within heat treatment or hardening processes for specimen is a production step within the manufacturing procedure of metallic workpieces. The quenching process often is associated with geometric distortion of the specimen in the manufacturing process, which is activated by high temperature gradients and stress in the workpiece during the fast cooling process. In industrial practice, this specimen distortion is compensated by material allowance in the manufacturing and finishing rework after heat treatment process, respectively.

The asymmetric quenching process in heat treatment processes has originally been developed for gaseous flow processes. Here by local control of gas jet flows precise distribution of heat transfer rates can be established. However, the pure convective heat transfer with gaseous flows causes too low heat transfer rates for certain material classes and specimen sizes. When using liquid jets instead of quenching gas at high specimen surface temperatures a vapour layer formation on the heated workpiece surface occurs. For that reason spray of fine liquid and an overlaid gaseous flow is impinged on the heated surface. The impinging liquid fraction of the spray may be restricted so that on the specimen surface no liquid film with a vapour layer underneath is built. By means of spray cooling, the efficiency of a complete evaporative cooling process can ideally be reached. The use of two phase atomizers allows the adjustment of impressed heat transfer condition with the choice of spray parameters (liquid mass flow and gaseous pressure). For this reason, it is possible to set up local heat transfer conditions in ranges between pure gas and full liquid quenching processes.

This contribution describes the use of spray cooling techniques for specific quenching of simple shaped aluminum and steel specimen in heat treatment processes. For efficient estimation of the cooling process a spray characterization, consisting of drop velocity, drop diameter and impingement density measurements in combination with calculations of local heat transfer coefficients (HTC) is performed. The HTC-profile calculations were obtained from the solution of the inverse heat conduction problem based on thermographic temperature measurements during spray cooling of thin sheets. The used parameter range was a fluid volumetric flow rate of 0.25 l/h – 2.9 l/h and a gas pressure of 0.1 MPa – 0.56 MPa. The initial specimen temperature was about 1073 K and the detected surface temperature range reach from 1050 K - 400 K. Two experimental setups were used, the thermographic setup and a flexible spray field setup, as industrial used quenching setup. The estimated HTC have peak values of more than 6000 W/ (m²K) and depend on specimen surface temperature and position. The main parameter for the HTC distribution is the specimen surface temperature. The gas pressure influence on the cooling process is minimal. With an increasing flow the fluid flow exhibits a faster cooling reaction and a faster cooling of the specimen and therefor a higher HTC. Analysis of the cooling curves assumes a wide range HTC function and low range HTC function. Low range HTC function exhibits the influence of the Leidenfrost effect. Wide range HTC function demonstrated the function of temperature and position to the HTC. The temperature depending HTC distribution over the specimen surface also includes the non-directly impinged faces, which show the influence of the three dimensional heat conduction. The results are implemented and validated with established calculation, model approaches and numerical cooling simulations. Here the knowledge of HTC profiles for chosen spray operation parameters and the estimation of hardness measurements, process dependent resulting distortion and metallographic analysis build the basis for continuous analysis and validation of the cooling effect by asymmetric spray quenching of complex shaped workpieces.

---

1 Corresponding author: b.hinrichs@iwt.uni-bremen.de

Application of dimensional analysis in estimating the wall film thickness created by a liquid spray impact

D. Kalantari*, C. Tropea**
*Dep. of Mech. Eng., University of Sari (SANRU), P.O.Box 578, Sari, Iran
**Institute of Fluid Mechanics and Aerodynamics, Center of Smart Interfaces, Technische Universität Darmstadt, Germany
dkalantari2000@gmail.com and ctropea@sla.tu-darmstadt.de

Abstract

This study presents a theoretical investigation on the formation and spreading of a liquid film on a flat and rigid wall due to spray impact. In the impingement region of an inertial spray, the average film thickness created on the wall depends on the several parameters of the impacting spray ; normal and tangential component of impact velocity $\bar{u}_b$ and v_b , volume flux density of impacting spray ($\dot{q} = q/A$; "q" and "A" to be volume flux of the impacting spray(m^3/s) and the reference area over which flux is measured), volume-averaged diameter of impacting droplets (d_{30b}) defined by $d_{30b} = \left(\sum_{i=1}^{N} d_i^{\,3}/N\right)^{1/3}$, density ($\rho_L$) and dynamic viscosity of the liquid (μ), as well as the boundary condition of the target (e.g., flat or curved target surface or target surface completely covering with spray); average target surface roughness ($\bar{\varepsilon}$) and target size (D). For characterizing the average film thickness accumulated on the wall due to a normal spray impact (negligible tangential velocity component of the impacting droplets) and for a spray covering the target surface (i.e., $D_{spray}/D>1$), variables influencing the film thickness can be reduced to

$$\bar{h} = \Psi(\bar{u}, d_{30b}, \dot{q}_b, \rho_L, \mu, \bar{\varepsilon}) \qquad (1)$$

The parameters $\bar{u}_b$, $\dot{q}_b$, and d_{30b} vary with nozzle pressure and nozzle height above the target. Based on the Π-theorem, the functional relationship for the (1) can be written in the form of

$$\left(\frac{\bar{h}}{d_{30b}}\right) = f\left(\frac{1}{Re_b}, \frac{\dot{q}_b}{\bar{u}_b}, \frac{\bar{\varepsilon}}{d_{30b}}\right) \qquad (2)$$

Practically it is more convenient to use d_{10b} instead of d_{30b} for non-dimensionallizing the target surface roughness in the form of $\bar{\varepsilon}^* = \bar{\varepsilon}/d_{10b}$. For further developing the obtained relationship (2) and trying to remove the Re number from within the bracket, an asymptotic solution of a single droplet impact will be used in this study. The dependency of h^* on $Re_b^{\,-1/2}$ in the case of a single drop impact for inertial impact condition indicates that we can extract the Reynolds number outside of Eq. (2) in the form

$$\left(\frac{\bar{h}}{d_{30b}}\right) = Re_b^{\,-1/2} f\left(\frac{\dot{q}_b}{\bar{u}_b}, \frac{\bar{\varepsilon}}{d_{10b}}\right) \qquad (3)$$

One simple form of the (3) for a negligible surface roughness can be written as

$$\bar{h} = \alpha d_{30b} Re_b^{\,-1/2} \left(\frac{\dot{q}_b}{\bar{u}_b}\right)^{\gamma} \qquad (4)$$

where α and γ are constant values found to be 4 and -0.5, respectively for negligible surface roughness and normal impact condition.

The expression (4) has been derived for a general spray impact condition and the only important condition in using this expression for film thickness is that the entire target surface should be exposed to the impacting spray. Results obtained in this study indicate a significant influence of the Reynolds number on the average film thickness accumulated on the wall due to liquid spray impact. The results obtained in this study indicate good prediction of the average film thickens in the case of thin film condition, i.e., $\bar{h}/d_{10b} < 1$.

* Corresponding author: dkalantari2000@gmail.com

Geometry Effects on Steady and Acoustically Forced Shear-Coaxial Jet Sprays

S. Teshome[*+], I.A. Leyva[+&], J. Rodriguez[+], D. Talley[+]
[*]UCLA, Los Angeles, CA, USA, [+]Air Force Research Lab, Edwards, CA, USA
steshome@ucla.edu and ivett.leyva@edwards.af.mil

Abstract

This experimental study investigated the mixing behavior and characteristics of dynamic flow structures of cryogenic, non-reactive shear coaxial jet sprays under varying flow conditions, with and without the presence of pressure perturbations due to acoustic forcing transverse to the flow direction. The role of injector geometry was examined using four shear coaxial injectors with different outer-to-inner jet area ratios and different inner jet post thickness to inner jet diameter ratios. Flow conditions at a high pressure spray (reduced pressure of 0.44) with varying outer-to-inner jet momentum flux ratios ($J \sim 2 - 20$) and maximum amplitude in the pressure perturbation (pressure antinode) at the jet axis location were considered. Nitrogen was used as the test fluid for the injector streams and to pressurize the chamber. The inner and outer jet temperatures were independently controlled so that the inner condensed flow was cooled down to or below the saturation temperature of the liquid. Back-lighting the coaxial spray resulted in a silhouette of the dense inner core, which appeared as a dark column. This distinguished it from the outer gaseous flow, and thus, enabled high speed images to capture the jet flow dynamics. Dark-core length measurements, pertaining to the unmixed portion of the dense inner flow, were used to indicate the extent of mixing under the different flow conditions and injector geometries. A basic application of proper orthogonal decomposition on the intensity fluctuation of the high speed images enabled the extraction of the spatial and temporal characteristics of the dominant flow structures that existed in the flow field at baseline conditions and during exposure to acoustic forcing.

The extent of the influence of outer-to-inner momentum flux ratio on mixing was dependent on the injector geometry. The dark-core lengths of the inner jet for injectors with large outer-to-inner jet area ratio (LAR), with both thin and thick inner jet posts, were more influenced by increasing momentum flux ratios. In contrast, the dark-core length of the inner jet for a geometry consisting of a small outer-to-inner jet area ratio (SAR) with a thin inner jet post was very insensitive to *J* for the range of values tested. The response of the injector flows to a pressure antinode was also dependent on the particular geometry. The flow of the SAR injector with a thin inner jet post showed a strong response to acoustics regardless of *J*, while the response of the other injectors to a pressure antinode was dependent on *J*. The observations from this study point to the significance of how simple design alterations in injector geometries may bring about drastic changes in the mixing and response of shear-coaxial jet flows to external pressure disturbances.

[&]Corresponding author: ivett.leyva@edwards.af.mil

Posters

September 3 - 6, 2012

Contents

Atomization Modeling of Liquid Jets using Two Surface Density Equations

B. Mandumpala Devassy*[1], C. Habchi[2], E. Daniel[3]
1 & 2: IFP Énergies nouvelles, 1-4 av. Bois-Préau - 92852 Rueil Malmaison, France,
md.bejoy@ifpenergiesnouvelles.fr & chawki.habchi@ifpenergiesnouvelles.fr
3: Aix-Marseille Université, IUSTI, 5 rue E. Fermi 13453 Marseille cedex 13, France,
eric.daniel@polytech.univ-mrs.fr

Abstract

Atomization is a phenomenon of high importance for many practical devices involving turbulent mixing of compressible flows having significant density and velocity gradients. This study focuses on the modeling of liquid jets in the aim of improving the Diesel spray simulations. Here a compressible two-phase flow model is used which relies on an Eulerian-Eulerian (E-E) diffused interface approach [1] implemented in the three dimensional CFD code IFP-C3D [2]. The current trend to track the liquid-gas interface is done by defining surface density ($\overline{\Sigma}$) equations. This quantity represents the area of gas-liquid interface and will allow the calculation of the interfacial exchange terms like heat transfer and phase changes. The first model of this kind in two-phase flows has been done by Vallet $et\ al.$ [3]. Compared to previous $\overline{\Sigma}$ models [3, 4], the significance of the present model lies in the fact that, near to the nozzle, during the time of primary breakup there happens a transfer of liquid from the liquid core (separate phase) to the dispersed phase. The initial developments of this model is summarized in the work of Mandumpala Devassy $et\ al.$ [5]. While modeling the atomization of liquid jet using $\overline{\Sigma}$ equation the need of separating the liquid phase into the separate and dispersed phase thus focuses attention. This idea is then formulated and thereby a new atomization model is developed by defining separate equations of $\overline{\Sigma}$ for liquid core and droplets (dispersed phase) coupled with their corresponding volume fraction equations. This new atomization model based on two surface density equations thus proposed made it possible to distinguish between the liquid core and droplets (Figure (1)) more precisely.

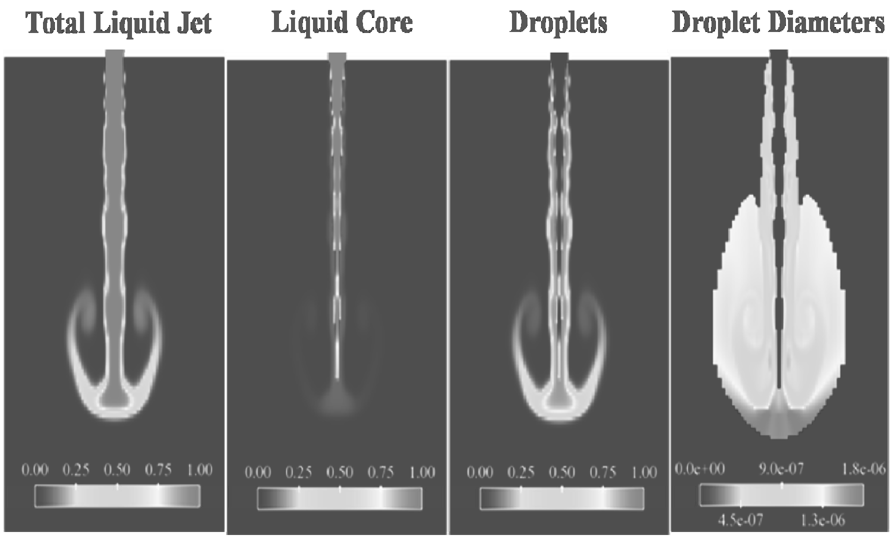


Figure 1. Liquid jet atomization by the proposed atomization and breakup model at $20\mu s$ after start of injection

References

[1] Saurel, R. and Lemetayer, O., *Journal of Fluid Mechanics* 431:239-271 (2001).
[2] Bohbot, J., Gillet, N., Benkenida, A., *Oil and Gas Science and Technology- IFPEN* 64:309-335 (2009).
[3] Vallet, A., Burluka, A., Borghi, R., *Atomization and Sprays* 11:619-642 (2001).
[4] Lebas, R., Ménard, G., Beau, P.A., Demoulin F.X., *Int. J. Multiphase Flow* 35:247-260 (2009).
[5] Mandumpala Devassy, B., Habchi, C., Daniel, E., *Proc. 24th Ann. Conf. ILASS Europe* (2011).

*Corresponding author: md.bejoy@ifpenergiesnouvelles.fr

Effect of split-injection strategies on spray characteristics studied for an outward opening pintle-type DISI piezo-injector

L. Zigan[1,2*], D. Loos[1], I. Schmitz[2,3], M. Wensing[1,2], A. Leipertz[1,2]
[1]Lehrstuhl für Technische Thermodynamik (LTT), Univ. Erlangen-Nuremberg, Germany
[2]Erlangen Graduate School in Advanced Optical Technologies (SAOT), Germany
[3]now at Lehrstuhl für Technische Thermodynamik (TTS) Univ. Siegen, Germany
Lars.Zigan@cbi.uni-erlangen.de

Abstract

Direct Injection Spark Ignition (DISI) combustion engines provide a promising technology to minimize fuel consumption and pollutant emissions. Especially, piezo-actuated outward opening pintle nozzles show advantages in generating a very fine and reproducible hollow-cone spray [1] for a fast mixture formation. This is currently of great interest for reduction of soot formation during mixing-controlled combustion of spray-guided DISI engines. Furthermore, the precise control of the injection quantity and timing is favourable to realize high-efficiency combustion. Split injection strategies can be used to extend the stratified combustion regime and to control the mixing timing. In this study, the liquid spray structure was analyzed for single injection and different split injection schemes in an injection chamber for a piezo-actuated outward opening pintle nozzle (Continental). Laser-based techniques such as planar Mie-Scattering and phase Doppler anemometry were used and a numerical spray model was set up applying the 3D-CFD-Code OpenFOAM. The injection quantity was 12 mg/injection (dwell time: 400 µs, injection pressure: 20 MPa, fuel: iso-octane) representing a part-load condition. The ambient conditions were 1.5 MPa at 283 K and the fuel temperature was set to 263 K for a non-evaporating spray investigation. First, the numerical model was calibrated for the single injection regarding spray shape, droplet size distribution as well as size and position of the recirculation zones (see Figure 1). The simulation shows very good agreement and was validated for the different split injection schemes at constant injected mass. The effects of variable pulse duration of the first and second injection as well as the delay time between the pulses on the spray structure were studied in comparison to the single injection. For all tested double-injection schemes, the spray length decreases compared to the single injection spray whereas the radial penetration increases. The average droplet sizes are very similar for the single and split injection scheme, the Sauter Mean Diameters (SMD) were around 26 µm due to collision and coalescence effects. Longer pulse duration of the first injection strongly increases the early axial and radial spray penetration due to the larger initial spray momentum, whereas the delay and duration of the second pulse determines the number of vortices and their shapes as well as the radial spray propagation at later times after start of injection.

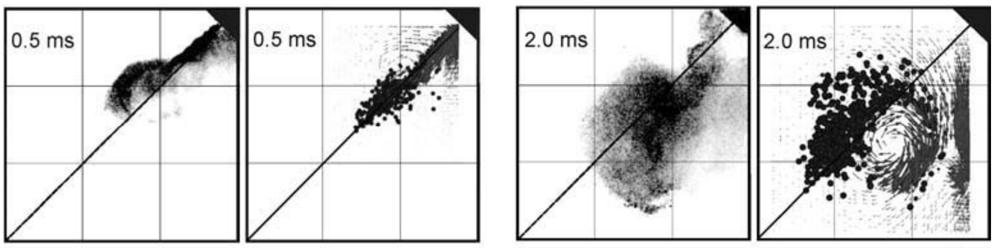


Figure 1: Comparison of the spray shape for single pulse injection, left: 2D Mie-scattering image (single-shot), right: simulation of the flow field (arrows) and the liquid phase (black dots)

References

[1] Zigan, L., Schmitz, I., Flügel, A., Wensing, M., and Leipertz, A., Structure of evaporating single- and multi-component fuel sprays for 2nd generation gasoline direct injection. Fuel 90 (2011), p. 348–363

Acknowledgements

The authors gratefully acknowledge the financial support for parts of this work by the Erlangen Graduate School in Advanced Optical Technologies (SAOT) which is funded within the framework of the German Excellence Initiative by the German Research Foundation (DFG). Furthermore, we thank Dr. Jun-Mei Shi, Dr. Milos Tichy, Dr. Eberhard Kull and Manfred Klepatsch (Continental Automotive GmbH, Regensburg) and Dr. Jerome Helié (Continental Automotive SAS, Toulouse) for the stimulating discussions and for the technical support.

* Corresponding author: Lars.Zigan@cbi.uni-erlangen.de

The primary breakup of a generic prefilming airblast atomizer using the embedded DNS (eDNS) concept

B. Sauer*, N. Spyrou, A. Sadiki, J. Janicka
Institute for Energy and Powerplant Technology, TU Darmstadt, Petersenstr. 30, 64287 Darmstadt, Germany
sauer@ekt.tu-darmstadt.de, spyrou@ekt.tu-darmstadt.de, sadiki@ekt.tu-darmstadt.de, janicka@ekt.tu-darmstadt.de

Abstract

Lean premixed and prevapourized (LPP) combustion is the concept desired for future aircraft engines. Thus, a high demand to fuel preparation and fuel atomization is required. An improved understanding of the breakup processes of two-phase flows is essential to effectively control the fuel atomization. A detailed insight into the phenomena of primary breakup is a major limitation in gaining this knowledge. Aircraft engines apply airblast atomizers to provide the fuel atomization. The geometries of airblast atomizers are complex, the operating conditions are characterized by high Reynolds- and Weber numbers. Consequently, Direct Numerical Simulations (DNS) of liquid breakup under realistic conditions and geometries are hardly possible. The embedded DNS (eDNS) concept aims to fill this gap.

The concept consists of three steps: (1) a geometry simplification, (2) the generation of realistic boundary conditions for the DNS and (3) the DNS of the breakup region itself. Airblast atomizers are described by two air flowing channels at high velocity and a fuel wall film at low velocity. At the trailing edge of the prefilmer the liquid sheet is subjected to high shear due to the high air velocities of the two channels resulting in a breakup process from large liquid structures to ligaments and droplets. The realistic annular airblast atomizer geometry is simplified to a Y-shaped channel representing a planar geometry. Inside this domain the embedded DNS is located downstream of the trailing edge. The embedded DNS domain requires the generation of boundary conditions. A single-phase LES of the Y-shaped air channel and a two-phase RANS of the liquid wall film are differentiated. The inflow of the single-phase LES is initiated by a synthetic turbulence inflow generator. Within the two-phase RANS, the liquid fuel enters the wall through a small liquid slit. The Volume-of-Fluid method (VOF) is used. Downstream of the slit, the data are stored as input to the lower channel of the DNS domain. The inlet to the upper channel, the bottom and the top edge of the embedded domain are generated by the single-phase LES. The fields are stored transiently for the DNS time steps. Linear interpolation in space and time is performed from the coarse LES to the fine DNS. Periodic boundary conditions are set for the lateral edges of the embedded domain. The DNS of the breakup region is computed using VOF. Inside the embedded domain, an equidistant grid is applied.

In a first parametric study the influence of the surface tension force on the primary breakup is investigated. For this study, the air flow conditions are kept constant at a bulk Reynolds number of around 11,000. The surface tension force is varied between 0.026 N/m and 0.0085 N/m corresponding to kerosene fuel temperatures between 300 K and 500 K. An equidistant grid spacing of 8 μm is set. The results provide an insight into the dynamics of the phase interface. A decrease of the surface tension force (corresponding to an increase in temperature) highly affects the stability of the liquid sheet. A series of instantaneous snapshots in time indicate the evolution of the liquid sheet deformation, the contraction and the generation of ligaments and droplets. For the low surface tension case, the entire interface is subjected to high shear resulting in strong topology changes and simultaneous collapsing of the liquid sheet at different sheet locations. The high surface tension case instead only generates a few single ligaments. The entire liquid sheet is stable.

This study proves the applicability of the eDNS concept for investigating breakup processes as the transient nature of the phase interface behaviour can be captured. The approach offers the potential of simulating realistic airblast atomizer geometries under realistic conditions.

* Corresponding author: sauer@ekt.tu-darmstadt.de

CFD Study of the Air Flow Field and Particle Deposition in the Human Nasal Cavity

X. G. Cui*, E. Gutheil
Interdisziplinäres Zentrum für Wissenschaftliches Rechnen
Ruprecht-Karls-Universität Heidelberg, Germany
xinguang.cui@iwr.uni-heidelberg.de and gutheil@iwr.uni-heidelberg.de

Abstract

The nasal cavity is a main part of the human upper airway, and it has the important functions of respiration, heating and humidifying the inhaled air. The properties of the air flow in the nasal cavity may contribute to the occurrence of respiratory disease. In addition, particles are filtered by the nasal cavity. This function prevents toxic particles to enter into the human body, but it also blocks the aerosol drug to reach the targeted position such as the middle region of the nasal cavity. Thus, it is very important to study the characteristics of both the air flow field and particle deposition in the nasal cavity.

In the present work, the computational surface grid of the nasal cavity is constructed based on computed tomography (CT) scans using NeuRA2 [1]. Ansys ICEM-CFD 11.0 is used to generate the numerical volume grid. Large eddy simulation (LES) with the Smagorinsky sub-grid model is used to simulate the gas flow, and the method is combined with Lagrangian particle motion. The open source software of OpenFOAM 1.5 is used to solve the governing equations, where a new solver has been constructed to account for the particle motion using a Lagrangian tracking method within the LES formulation for the flow field. The inspiration flow rate is 7.5 L/min, and 10,000 uniformly distributed particles are randomly released at the inlet plane with the same velocity as the air flow shown in Fig. 1. The particle density is 912 $\mathrm{kg/m^3}$, and the particle radius is 5 μm.

The numerical results reveal that the maximum velocity exists in the region of nasal valve, and the air impinges on the wall of the nasal valve. The air flow field in the left and right nasal passages is not uniform, which results from the asymmetric configuration of the nasal cavity. Only little air can reach the tips of meatuses and the olfactory region. The air flow passes the nasal cavity mainly through the main air passage. Vortices are observed in the nasopharynx region. The particle deposition in the nasal cavity is been compared with the numerical results of Matida et al. [2]. The present particle deposition rate in the nasal cavity is 24.54%, which is close to their result of 26.5% [2]. It is observed that the particle deposition in the nasal cavity mainly occurs in the anterior region, and particles mainly deposit on the impact sides such as the nasal valve, the anterior region of the middle meatus and the nasopharynx as shown in Fig. 2. The particle release positions at the inlet plane influence the particle deposition positions on the wall, and their ability to pass through the nasal cavity.

In the future, the grid will be refined, and different inspiration flow rates and particle diameters will be investigated. The interaction between the air flow field and particle transport will be studied in more detail.

Figure 1. Initial positions of the released particles at the inlet plane.

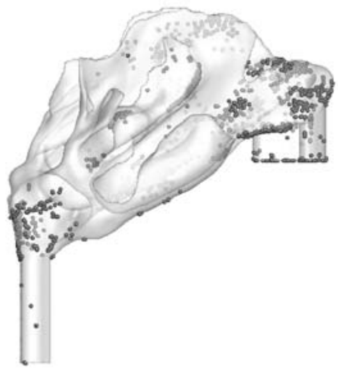


Figure 2. Particle deposition characteristics on the wall of the nasal cavity.

[1] Jungblut, D., Queisser, G., and Wittum, G., *Computing and Visualization in Science* 14: 181-186 (2012).
[2] Matida, E. A., Liu, Y., Gu, J., and Johnson, M. R., *Journal of Aerosol science* 38: 683-700 (2007).

*Corresponding author: xinguang.cui@iwr.uni-heidelberg.de

Real multiple orifice Diesel nozzle geometry by X-ray micro-CT for internal flow simulation

M. Santini*[1], S. Fest-Santini[1], P. Conti[1], G.E. Cossali[1], R. Dondé[2]
[1]Department of Industrial Engineering, University of Bergamo, Italy
[2]CNR - IENI, Italy
santini@unibg.it, fest@unibg.it, conti@unibg.it, cossali@unibg.it, donde@ieni.cnr.it

Abstract

Traditionally, CFD simulations of the flow inside fuel nozzles are performed on ideal meshes, obtained from the nominal nozzle geometry. The growth of disturbances which ultimately leads to disintegration of the liquid jet or sheet into ligaments and then into drops is affected by the real internal injector geometry, which differs usually from the nominal one due to fabrication tolerances, roughness, cavitation erosion, etc.

Micro-CT instrumentation allows investigating at micrometric resolution the internal injector geometry. The real three-dimensional geometry is reconstructed from a set of planar X-ray shadow images and the internal surface is extracted based on isogray-values out of the volumetric data set. The achieved accuracy depends mainly on the spatial resolution of the volumetric data set.

Production accuracy of injector can be controlled by superposition of nominal CAD geometry and reconstructed 3D volume (see Figure 1). The extracted surface itself can be used as boundary domain to perform CFD simulations in order to evaluate the effect of real geometry on the internal fluid-dynamics.

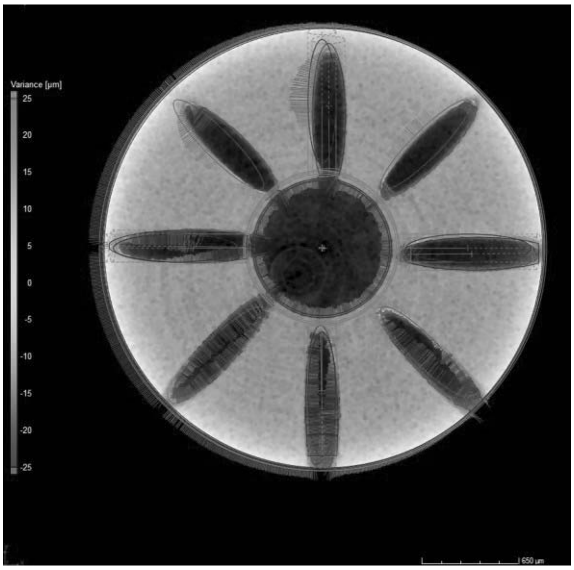


Figure 1: Difference of real and nominal injector geometry

The reported work shows the capability of the micro-CT technique to reconstruct the internal geometry of a real Diesel injector and to create a proper mesh to perform CFD simulations of the fuel flow inside the nozzle holes and cavity. Internal flows computed for real and nominal geometries are compared.

Modelling Mixing and Particle Formation in Supercritical Antisolvent (SAS) Processes

J. J. Choi and A. Kronenburg*
Institut für Technische Verbrennung, Universität Stuttgart, Germany
jung-jin.choi@itv.uni-stuttgart.de and kronenburg@itv.uni-stuttgart.de

Abstract

The fluid properties at supercritical conditions such as small thermal diffusivity and close-to-zero surface tension favour complete mixing between solution and antisolvent and thus facilitate the production of ultra-fine nano-sized particles with a narrow size distribution. The supercritical antisolvent (SAS) process exploits these properties by injecting a liquid jet with a solute into a fluid at supercritical conditions, usually CO_2, that acts as antisolvent for the solute. The SAS process is an inherently complex system, that involves the interplay of equilibrium/non-equilibrium thermodynamics/transport properties of supercritical fluids with real gas effects as well as the hydrodynamics for turbulent mixing. In addition, the particle dynamics including nucleation and growth coupled with supercritical thermodynamic properties and mixing at molecular scale need to be carefully examined and modelled. Of all these processes, the mixing is pivotal in the SAS process. It is of primary importance for the formation of supersaturation conditions that lead to particle nucleation and thus controls the dynamics of particle production and evolution. A Large-Eddy Simulation (LES) method is used for the simulation of the flow and mixing fields. Compressibility effects cannot be neglected, and a compressible Navier-stokes formulation employing an AUSM$^+$-up scheme needed to be implemented for the description of the compressible but low Mach number flows predominant in SAS processes. The Peng-Robinson equation of state is used to describe real gas effects, and the particle evolution and size distribution are modelled by the population balance equation using the method of moment approach. In LES only the large scale processes flow structures are spatially and temporally resolved. The small, sub-grid processes, in particular the micro-mixing that locally affects supersaturation and particle nucleation, and the interactions between turbulence and particle dynamics need to be modelled. We introduce a conditional moment closure (CMC) method as a sub-grid model to account for the unresolved effects of turbulence on particle nucleation and growth. CMC is a sub-grid model for non-premixed combustion [1] where chemical reaction is strongly dependent on micro-mixing, and the same modelling ideas can be used for the prediction of the nucleation term. Preliminary investigations of particle formation of paracetamol in the mixture of ethanol and CO_2 at T=314K and P=16MPa have been carried out. Computations with varying mole fractions of CO_2 using the equilibrium thermo-hydrodynamics assumption show good agreement with experimental results from [2]. The poster will present preliminary simulation results from a ternary system of paracetamol-ethanol-CO_2, where ethanol and the solute are injected into the mixing chamber by a jet. The LES of the hydrodynamics is fully coupled with the solution of the transport equations of the conditionally averaged moments of the particle dynamics and size distribution. The effects of sub-grid micro-mixing (and essentially turbulence) on particle nucleation and growth can be quantified, and the results can be compared with earlier experiment by Bristow *et. al.* [2].

References

[1] Navarro-Martinez, S., Kronenburg, A. and diMare, F., *Flow, Turbulence and Combustion* 75: 245-274 (2005).
[2] Bristow, S., Shekunov, T., Shekunov, B. and York, P., *Journal of Supercritical Fluids* 21:257-271 (2001).

*Corresponding author: kronenburg@itv.uni-stuttgart.de

Modeling of Spray Formation and Combustion

A. Kösters, A.Karlsson
Chalmers University of Technology, Sweden
Anne.Kosters@chalmers.se and Anders.anka.Karlsson@volvo.com

Abstract

The break-up of the droplets in a fuel spray represents a very important process. Due to different nozzle shapes and injection pressures the break-up of Diesel and gasoline sprays differs. The high injection pressures in Diesel engines and hence high relative velocities between the fuel and the air cause a fast break-up of the liquid. The main break-up process can be described by secondary break-up. The relative velocity in gasoline sprays is much lower and in e.g. hollow cone sprays a liquid sheet is breaking up into droplets (atomization) and the droplets then break-up into smaller droplets (secondary break-up). In order to satisfactorily model the mixing of fuel with air, a good understanding of all the different processes occurring, such as spray break-up and evaporation, is required and these must be captured by the model. The VSB2 spray model, implemented in OpenFOAM®, is based on the idea of constructing a model that treats the spray and its break-up as one process, instead of summing individual, fragmenting droplets to a spray. The design criteria of the model were that it should be unconditionally robust, the number of tuning parameters should be minimal, and it should be possible to implement in any CFD software package. Spray simulations of both full cone Diesel sprays and hollow cone gasoline sprays were performed. Predicted Diesel spray liquid and vapor penetration are compared to experimental data from the Engine Combustion Network (ECN), Sandia National Laboratories. The gasoline spray simulations are compared with data acquired at Chalmers.

The modeling of turbulent combustion offers difficulties, since both the turbulent mixing of vapor fuel with air and the turbulence-chemistry interaction have to be described. There exist different approaches to modeling turbulent combustion. The Volume Reactor Fraction Model (VRFM) is based on the Eddy Dissipation Concept, introduced by Magnussen in 1981. The idea of the Volume Reactor Fraction Model is to define a reactor with a volume smaller than or equal to the computational cell volume. The reactor volume is depending on the state of unmixedness in mixture space and in chemical progress space. With this definition the problem of finding representative timescales of mixing and chemical reactions is removed. The simulated lift-off lengths of n-heptane sprays are compared to experimental data from the ECN.

Study on Spray Propagation under Pressurized Flow Field in Circuit Wind Tunnel

H. Tajima[1*], H. Kato[1], D. Tsuru[1] and R. Ishibashi[2]
[1] Interdisciplinary Graduate School of Engineering Sciences, Kyushu University, JAPAN
[2] MITSUI ENGINEERING & SHIPBUILDING CO., LTD., JAPAN
tasima@ence.kyushu-u.ac.jp

Abstract

Interaction between a spray and its surrounding air is a key factor for engine CFD codes since spray penetration and droplet breakup have their source from the relative velocity between the droplets and the ambient flow. It applies especially to the sprays in large marine diesels which often propagate nearly parallel to a swirl flow to promote their penetration and distribution inside the large combustion chamber. Various sub-models in spray simulations, however, have been developed for smaller engines where the sprays are injected orthogonally into swirling in-cylinder gas. This may lead to obscurity in the spray prediction in large diesels. Practical and well-founded spray models are definitely needed in the field, but a lack of experimental information keeps it difficult to derive and validate such models. In this study, a diesel nozzle of a single injection hole was inserted into a channel of a closed-circuit wind tunnel (Fig.1), which has a cross-section of 50 mm×50 mm and realizes maximum free stream velocity of 22 m/s under air pressure of 1.1 MPa. The spray was injected parallel to the tunnel stream windward or leeward, and its propagation was recorded by a high-speed camera. The measurement results (Fig.2A) showed the free stream velocity has significant effects both on the spray penetration and on the spray cone angle (Fig.3). Although the breakup process near the nozzle exit showed less dependency on the ambient flow, child droplets detached from an outer edge of the everted spray tip were observed to impinge on the liquid column of the spray body (Fig.2B). The observed spray propagation inside the ambient flow was successfully reproduced via a KIVA-3V code based on the unique modification in the collision length treatment (Fig.4).

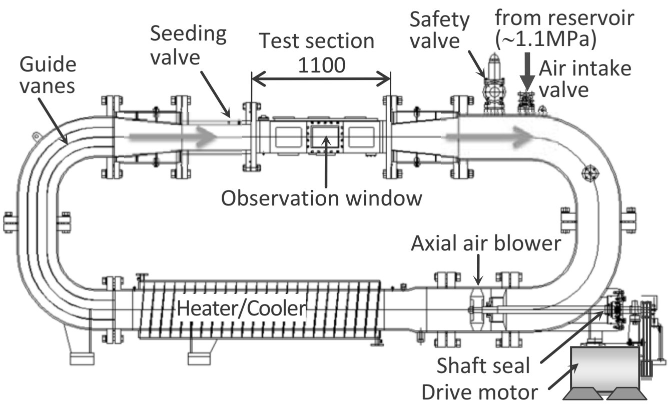


Fig.1. Overview of Closed-circuit Wind Tunnel (CWT)

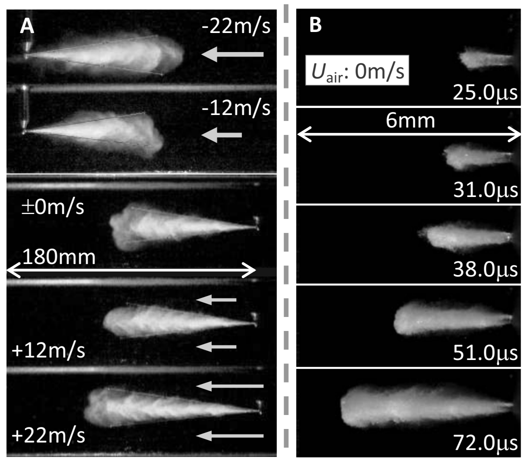


Fig.2. Spray penetrations under different wind velocities (A, left) and macro-graphic spray images near nozzle exit (B, right)

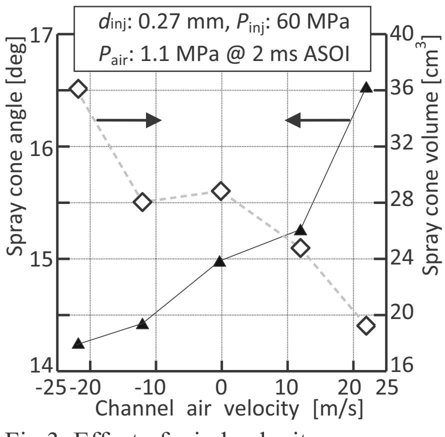


Fig.3. Effect of wind velocity on spray cone angle and estimated spray volume

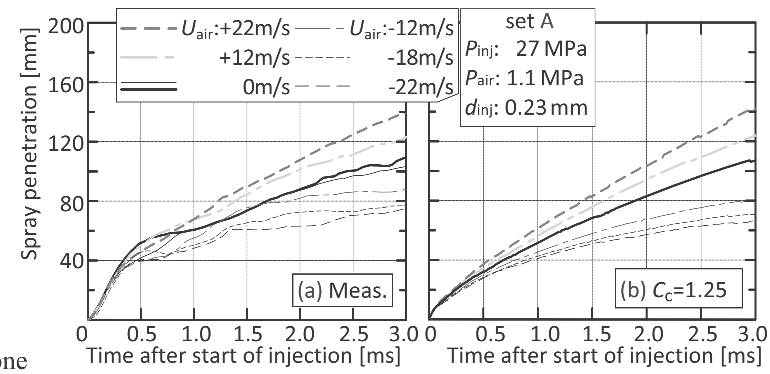


Fig.4. Comparison of measured spray penetration with simulated ones of different collision lengths

---

* Corresponding author: tasima@ence.kyushu-u.ac.jp

Transcritical droplets: An experimental and numerical analysis

F. Weckenmann*[1], I. Shevchuk[2], B. Bork[3], G. Lamanna[1], B. Weigand[1], J. Janicka[2],
A. Dreizler[3]
[1] Uni Stuttgart, ITLR, Pfaffenwaldring 31, 70569 Stuttgart, Germany
[2] TU Darmstadt, EKT, Petersenstraße 30, 64287 Darmstadt, Germany
[3] TU Darmstadt, RSM, Petersenstraße 32, 64287 Darmstadt, Germany
Florian.Weckenmann@itlr.uni-stuttgart.de, bork@csi.tu-darmstadt.de and
shevchuk@ekt.tu-darmstadt.de

Abstract

Numerous research efforts in the field of combustion technologies, e.g., gas-turbines, liquid propellant rockets or diesel engines, have the goal to increase the overall efficiency of the deployed facilities. A standard approach is the rise of system temperature and pressure, in modern devices even beyond the critical point of the injected fuel. The injection and disintegration of the liquid fuel is a crucial aspect in this context as it defines the mixing behaviour inside the combustion chamber. A liquid droplet, injected at a subcritical temperature and embedded in a supercritical environment, undergoes two simultaneous processes: 1) warming up due to the heat transfer from the gas phase towards the droplet; 2) cooling down due to the rapid evaporation in the unsaturated surroundings. For injection temperatures approaching the critical value (e.g. $T_d \approx 0.9\,T_c$), the drop might experience a transition to a supercritical state if the heat transfer from the surroundings prevails over the cooling effect of the vaporisation process. These characteristics strongly influence the evaporation and mixing process of the fuel and change it distinctively, compared to that of a subcritical combustion. For a better understanding of the fundamental phenomena and disintegration processes in a supercritical environment a detailed experimental and numerical study is undertaken in the framework of the SFB TRR75 collaborative research project.

A test facility for the experimental investigation of droplet evaporation in the vicinity of the critical point has been set up. It consists of a drop on demand droplet generator that is enclosed in a heatable pressure chamber. The drop detachment process is triggered by means of electric field forces. The temperature of the droplet liquid can be controlled independently of the surrounding gas temperature. Optical access to the process is provided by four fused silica windows allowing the coupling in and out of laser light and the observation by optical detectors. The functionality and reproducibility of the drop detachment process has been tested and quantified by high-speed shadowgraphy imaging. These images showed typical ligaments in the wake of the droplet, as already observed by different authors. Preliminary investigations with glare points and diffuse light indicate that these ligaments consist of an optically dense, gaseous mixture of the droplet fluid and the surrounding gas. The vapour layer is rapidly convected away due to the relative motion between the drop and the gas. In order to confirm these findings, a detailed experimental study will be undertaken incorporating advanced measurement techniques, e.g. Raman spectroscopy and Planar Laser Induced Fluorescence and Phosphorescence (PLIFP).

For the numerical investigation, a multiphase solver was created using the open source CFD toolbox OpenFOAM. The solver utilizes a Volume of Fluid (VOF) based approach for capturing the liquid-gas interface, allowing to account for significant interface deformations. It is capable to obtain solutions on general unstructured meshes, to adaptively increase the resolution of the interfacial regions and boundary layers using mesh refinement and to carry out parallel calculations on multiple CPUs. Species and energy transport in both phases as well as phase change is considered. Due to the strong dependence of material properties on temperature, pressure and mixture composition near the critical point, variable properties, including density fluctuations, are considered. Since focus is first put on free falling droplets in resting isobaric environment, low Mach number assumptions are made to reduce the complexity. An important objective of this project is the validation of the numerical method for simulation of droplet dynamics and evaporation based on the experimental database.

---

*Corresponding author: Florian.Weckenmann@itlr.uni-stuttgart.de

Theoretical Investigation of Breakup Length of Primary Atomization: Energetic Approach

I, Sher* and E.O. Diemuodeke**
Department of Process and Energy Systems Engineering, Cranfield University, Cranfield, Bedfordshire MK43 0AL, United Kingdom.
i.sher@cranfield.ac.uk* and **o.e.diemuodeke@cranfield.ac.uk

Abstract

Atomization process plays a significant role in the combustion of liquid fuels injected into gas turbine, internal combustion (IC) engines and incinerators combustors in order to achieve proper mixing of fuel and air and for rapid evaporation and combustion. The atomization process is divided into primary atomization and the secondary atomization. The primary atomization, which is the bridge between the nozzle internal flow and the secondary flow, is further classified, by the predominant breakup mechanism, into four regimes. One of the regimes is the atomization regime, which is the relevant operational zone for most combustion systems. There are divided opinions regarding the predominant breakup mechanisms for the atomization regime, but plausibly is by: aerodynamics interaction, turbulence, cavitaion and bursting effect. The properties of jets of most interest are the breakup length and the drop size. The breakup length provides a measure of the growth rate of the disturbance created by the interaction of cohesive and disruptive forces on the liquid jet and, also, it defines the point or region where secondary atomization starts, which is the fully dispersed multiphase flow region. Experimental measurements have contributed significantly to the understanding and development of empirical and semi-empirical models for the breakup. Liquid breakup length correlations, based on extensive experimental data, for turbulent jets are mainly grouped into three categories. However, of interest is the breakup of entire liquid column as a result of the turbulent primary breakup mechanism. It is obvious that empirically developed models are bounded and are heavily depended on the experimental approaches and conditions. Thus, attempt has been made by some researchers to manipulate experimental data and correlations, mechanistically, to arrive at models for mean breakup length for the breakup of entire liquid column due to the turbulent primary breakup mechanism. With the general acceptance of these models, however, these models do not account for the aerodynamic interactions of the issuing jet with the continuous phase and viscous effect. Therefore, this work presents model that uses a more robust development approach, the energetic approach, for the determination of the breakup length for the turbulent breakup mechanism of the atomization regime of primary atomization process, which accounts for aerodynamic interactions and viscous effect. Nevertheless, simplification of the original model to a more simplified one, which does not account for the aerodynamic and viscous effects, predicted experimental data accurately more than its counterpart, which has been widely accepted and used. Therefore, the developed model will find application in combustion systems, especially diesel direct injection system, since most of their operational zone are with the atomization regime of the primary atomization process.

* Corresponding author: i.sher@cranfield.ac.uk
Tel: +44 (0) 1234 754702

An attempt to visualize spray inside the premixing duct of a coaxial-staging lean burner at simulated full power conditions of modern/future high pressure ratio aero engines

K. Matsuura*[1], S. Ohori[2], Y. Yoshiura[2], Y. Kurosawa[1], H. Yamada[1], K. Shimodaira[1], S.Hayashi[2]

1: Japan Aerospace Exploration Agency, Japan
2: Department of Mechanical Engineering, Hosei University, Japan

matsuura.kazuaki@jaxa.jp

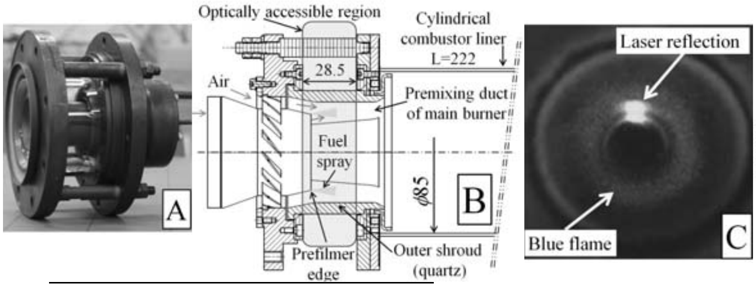
Abstract

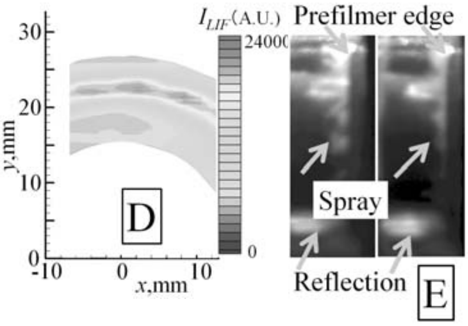
In order to reduce the fuel burn and the environmental impact by CO_2, the pressure ratio of modern aero engines tends to increase for better thermal efficiency. It is getting close to fifty even now and possibly higher in the future. This results in an increase of NOx emissions due to high combustor inlet pressure (p_{in}) and temperature (T_{in}). A solution for this problem is a coaxial-staging lean burner: It typically consists of a non-premixed-mode pilot burner at the center for flame stability and a lean-premixed-mode main burner surrounding the pilot for drastic NOx reductions. As the degree of fuel/air mixing in the main burner plays a key role in the NOx reductions, it is important to evaluate the spray and fuel vapor behavior inside the premixing duct of the main burner. Further, the evaluation should be carried out in realistic conditions, as the spray physics is strongly influenced by those of the ambient airflow. However, because of difficulty of experiments at such high pressures and temperatures and limited availability of huge facilities, the published data based on optical techniques on the phenomena inside the premixed duct in a realistic injector configuration are hardly available.

In this poster paper, our recent research activity on visualization of spray inside the premixing duct of a research-purpose coaxial-staging lean burner in such conditions is reported. The final targets of the test conditions will be up to p_{in} =4.7MPa and T_{in} =1000K. This can be realized by the JAXA high temperature and high pressure combustion test facility equipped with a 2MW electric heater with reliable temperature control for long-period continuous operations. The photograph and schematic drawing of the burner is presented in Figs. A and B. Its outer shroud is made of quartz and has an optical access of 28.5mm in length, including the position of the prefilmer edge of the main burner. A ceramic-coated cylindrical combustor with effusion cooling, 85mm inner diameter and 222mm length, was utilized. The flame was monitored by a periscope-like imaging system from the rear side, as shown in Fig. C. So far, for the visualization inside the premixed duct, the shadowgraphy and the laser sheet imaging (Mie scattering from the fuel spray) have been attempted. In addition, the kerosene LIF visualization is ongoing though it is qualitative at this stage.

Some typical results are presented in Figs. D and E. They were obtained at p_{in}=4MPa and T_{in} =900K, simulating typical takeoff conditions (almost full power) of modern civil aero engines. No pilot fuel was supplied for these cases. The main burner local air to fuel ratio (AFR_m) was kept at modest values since currently this research-purpose burner had not been optimized and combustion oscillations took place in the lower AFR_m range. For the stable combustion case at AFR_m=40, an example of time-averaged kerosene LIF results is presented in Fig. D, showing circumferential variation of fuel distribution in the premixing duct (6mm downstream from the prefilmer edge). For the oscillating combustion case at AFR_m=32.5, the two spray pictures close to the filmer edge taken at different times (Fig. E) show its temporal variation caused by the strong pressure oscillations.

Some technical issues on conducting experiments at such conditions are also discussed in our paper. This includes 'schlieren' effects or in other words 'beam steering' due to large density gradients along the optical paths, and contamination of the optical outer shroud by a green substance presumed as chromium oxide, which is probably originated from heating wires of the electric heater.





* Corresponding author: matsuura.kazuaki@jaxa.jp

Influence of feed viscosity on liquid drop breakup during effervescent atomization of aqueous polyvinylpyrrolidone solutions

A. Kleinhans*, J. Schröder, H. Schuchmann, V. Gaukel
Karlsruhe Institute of Technology, Institute of Process Engineering in Life Sciences, Section 1: Food Process Engineering
Agnes.Kleinhans@kit.edu, Jewe.Schroeder@kit.edu, Heike.Schuchmann@kit.edu, Volker.Gaukel@kit.edu

Abstract

The atomization of highly viscous liquids into a fine dispersed spray e.g. for spray drying processes is very demanding. For this process, pneumatic atomizers are widely used atomizers in industrial applications, despite the disadvantage of their high atomization gas consumption. A special type of internal mixing pneumatic atomizer with measurably lower atomization gas consumption is the effervescent atomizer [1], for which the formation of a two phase flow in the nozzle is characteristic. In previous investigations, the effervescent atomization of solutions with different viscosities led to significant differences of the liquid breakup near the nozzle which is exemplary shown in fig. 1 for 3 different viscosities. Simple measurement of the drop size distribution in the spray cone center gives only integral information about the fineness of the spray. For better understanding of the liquid breakup the local drop size distributions are measured in dependence of the axial [2] and radial distance of the nozzle outlet.

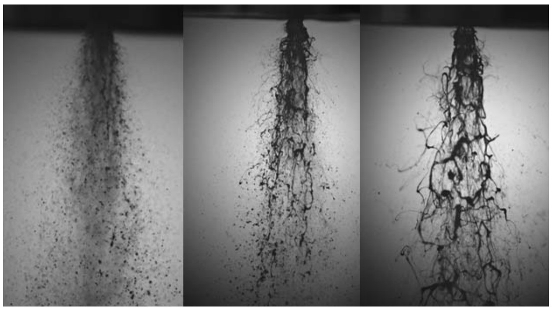


Fig.1: Shadowgraphy images of effervescent atomization of different concentrated PVP solutions near the nozzle: 20 % PVP (left, about 25 mPa·s), 30% PVP (middle, about 100 mPa·s) and 40 % PVP (right, about 500 mPa·s)

As feed aqueous solutions of the polymer polyvinylpyrrolidone (PVP) with different concentrations and therefore different viscosities up to 1200 mPa·s were used. Laser diffraction spectroscope (Malvern Spraytec) for measurement of drop size distribution and shadowgraphy for analyzation of the spray morphology were employed. Abel Inversion was applied to calculate the local, radially dependent drop size distributions from the integral measurements.

The sauter mean diameter of the spray droplets increases with rising viscosity and increases with the radial coordinate with the smallest droplets occurring in the cone center. Results depict that there may be drop coalescence at a distance of 25 cm underneath the nozzle. This will be subject of further investigations.

The work has been carried out with financial support of the German Research Foundation (DFG) within the research program DFG-SPP 1423 “Prozess-Spray”.

[1] Petersen, F. J. et al.: *Effervescent atomization of aqueous polymer solutions and dispersions*, Pharmaceutical development and technology, 6 (2) 201-210, 2001.
[2] Schmidt, F., Mewes, D.: *Zerstäuben von Suspensionen - Tropfengröße und -geschwindigkeit im Spray eines innenmischenden Zweistoffzerstäubers*, Chemie Ingenieur Technik, 79 (12) 2075-2082, 2007.

* Corresponding author: agnes.kleinhans@kit.edu

Spray analysis of an internal steam-assisted atomizer for multiple geometries of the fuel injector, mixing chamber and nozzle

C. Mirat*[1,2], T. Schuller[1,2], D. Durox[1,2]
(1) CNRS, UPR 288 « Laboratoire d'Energétique moléculaire et macroscopique, combustion », Grande Voie des Vignes, 92295 Chatenay-Malabry France
(2) Ecole Centrale Paris, Grande Voie des Vignes, 92295 Chatenay-Malabry France

Abstract

Steam-assisted atomizers are widely used in thermal power plants fed by heavy fuel. In the present study, a generic internal steam-assisted atomizer formed by an assembly of three different pieces, a central nozzle for liquid fuel injection, a mixing chamber for steam addition and a nozzle for spray injection has been designed. The fuel and steam supplies feed the mixing chamber through a central fuel injector intersected by four radial steam jets distributed around the mixing chamber. This two-phase fluid mixture leaves the mixing chamber through five small injection holes regularly distributed around the atomizer nozzle with an angle of 52.5° with respect to the atomizer axis. The system was developed to conduct parametric analyses of the resulting spray structure for different geometrical configurations of the atomizer, steam and fuel injection pressures. The analysis is conducted here for a central fuel injector with a hole diameter of 0.10 mm, a mixing chamber with steam hole diameters equal to 0.10 mm and a spray hole diameter of 0.25 mm. The atomizer is embedded in a regulated experimental setup with controlled mass flow rate, temperature and inlet gauge pressure for both steam and fuel flows. Steam is supplied with an evaporator that provides a maximum water mass flow rate of 2 g/min, evaporated by a 500 W thermal resistance. The steam flow is overheated at 453 K and can be delivered at a pressure up to 5 bar. A pressurized tank and a Coriolis mass flow rate controller are used to deliver a maximum fuel flow rate of 14 g/min at 13 bar. The whole test rig is kept at constant temperature. The gas to liquid mass flow rates ratio GLR is varied in these experiments between 0.02 to 0.15 as in typical industrial applications. The analysis is conducted here for a surrogate fuel with dodecane. Two different types of experiments are conducted to characterize the spray. The spray is first examined using back light emission with a CCD camera. A Particle Doppler Anemometer PDA is used to characterize the influence of GLR on the droplet size distribution in a selected number of geometrical cases at different axial positions downstream the nozzle exit. Measurements are compared to classical correlations found in the literature.

---

*Corresponding author: clement.mirat@ecp.fr

Damped Spring-Mass Analogy of Droplet Oscillations

D. Banks, R. Sanchez, and G. Aguilar
Department of Mechanical Engineering, University of California-Riverside, United States

Abstract

After a droplet strikes a solid surface, there is a period of time where the drop oscillates between a nearly hemispherical shape and a toroid until it comes to rest. In the study of drop solidification and drop heat transfer, understanding the motion of the fluid after impact is essential. Connecting the impact characteristics, droplet properties, and surface features to the oscillating motion of the drop after impact will benefit applications of spray cooling and spray materials deposition.

Using a high speed video camera (Phantom V7.1), drop fluids spanning two orders of magnitude range of viscosity (1.0E-6 to 1.1E-4 m^2/s), and three surfaces with differing wetting characteristics, we have measured the characteristics of drop oscillation for low Weber number ($We \approx$ 20-30) impacts. Measuring the thickness at the center of the drop splat, the magnitude and frequency of oscillation are found over time.

We examine this oscillation through the lens of a harmonic vibration analogy. The ultimate goal of this study is to develop this analogy such that, based on fluid properties and wetting and impact characteristics, the magnitude and duration of oscillations can be predicted from the equations of harmonic motion. This vibration analogy has been made in the past- notably, "Water Spring: A model for bouncing drops" by Okumura, et al [1], who used the idea of the analogy to predict deformation and bouncing of drops impacting highly hydrophobic surface. We have carried the idea one step further in attempting to directly apply the equation of harmonic motion to the drop behavior. Based on the hypothesis that the drop behaves as a damped harmonic oscillator, the analogues of the damping coefficient and the spring constant are computed and compared to the drop fluid properties and surface wetting tendencies.

Drop viscosity and surface wetting are shown to interact non-trivially in their effects on droplet oscillations. Separately, each follows intuitive trends- more viscous liquids tend to have more strongly damped oscillations on a wetting-similar surface, and hydrophobic surfaces tend to reduce damping for most drops. Examining the spread across the range of both characteristics, however, leads to behavior akin to frequency forced damping, resonance, and fluctuating amplitude of oscillation counter to intuitive trends.

References

1. Okumura, K., Chevy, F., Richard, D., Quéré, D., and Clanet, C., *Water spring: A model for bouncing drops*, Europhysics Letters, 2003. **62**:2, 237-243.

Corresponding author: gaguilar@engr.ucr.edu

Investigation of Spray Combustion Phenomena with Swirl Flow in a Lab-Scale Aircraft Gas Turbine Combustor

Kazuki Tainaka[1], Ryuhei Kawano[1], Jun Hayashi[*1], Noriaki Nakatsuka[1], Hideki Moriai[2] and Fumiteru Akamatsu[1]
Affiliation, Osaka University, Japan[1], Mitsubishi Heavy Industry, Ltd., Japan[2]
tainaka@combu.mech.eng.osaka-u.ac.jp, kawano@combu.mech.eng.osaka-u.ac.jp, j.hayashi@mech.eng.osaka-u.ac.jp, nakatsuka@combu.mech.eng.osaka-u.ac.jp, hideki_moriai@mhi.co.jp, akamatsu@mech.eng.osaka-u.ac.jp

Abstract

Low NOx emission gas turbine engines for small scale aircrafts have been required because of the growing concern about environmental issues. In order to develop low NOx emission gas turbine engines, it has been required to understand spray combustion phenomena in combustors of gas turbine engines. It is, however, difficult to clarify the underlying physics of spray combustion phenomena in actual combustors because of difficulties of measurements: Laser diagnostics require the light paths. The objective of this study is, therefore, to understand spray combustion phenomena in combustors of gas turbine engines. To understand the spray combustion phenomena, the experimental combustor to which the laser diagnostics is applicable is needed. In this study, the lab-scale experimental combustor with optical path, which simulates the primary combustion region of actual Rich-burn, Quick-quench, Lean-burn (RQL) type aircraft engines, has been developed and fabricated. The combustion chamber, which has three quartz windows, has been designed to simulate a unit section of the annular combustor. Kerosene/air spray combustion proceeds with double (inner and outer) co-swirl flow in the combustion chamber. The fuel nozzle supplies kerosene as liquid sheet and forms solid V-shaped spray cone with swirling air flow. This pre-filming nozzle is utilized in actual aircraft gas turbine combustors. In this paper, flame structure, combustion reaction region and spray region are visualized by direct photographs, chemiluminescence of OH*(Detecting wavelength: 310 nm) and Mie scattering from fuel droplets, respectively. The direct photographs of the flames are taken by digital camera. Images of OH* chemiluminescence and Mie scattering are captured by high-speed CCD camera. Two dimensional images of OH* chemiluminescence is obtained by using the inverse Abel transform. In addition, the axis distribution of gas temperature is measured by B-type thermocouple.

It is found from direct photographs and two dimensional images of OH* chemiluminescence that OH* chemiluminescence doesn't exist in the luminous flame. This result indicates that gaseous-phase combustion reaction doesn't occur in the luminous flame region, and this means that brightness of the luminous flame consists of incandescence of soot heated in the upstream combustion reaction region. The combustion reaction region expands to the downstream with decreasing the fuel flow rate and increasing the air flow rate. Decrease in the fuel flow rate and increase in the air flow rate make the fuel atomization effective because of increase in sheer stress between the gas phase and the liquid phase. Therefore, it can be assumed that small droplets formed by fine atomization moved along with the swirling flow. Both evaporation of those droplets and combustion reaction take place in downstream of the combustor. In fuel rich condition, it can be also assumed that oxygen is used up in the upstream and combustion reaction doesn't occur in the downstream. The gas temperature measured by B-type thermocouple indicates that the peak value in the combustion reaction region exists on the center axis. Two dimensional images of Mie scattering shows the spray region moves downstream with increasing the air flow rate. On the other hand, the spray region doesn't move with decreasing the fuel flow rate.

* Corresponding author: j.hayashi@mech.eng.osaka-u.ac.jp

Experimental Study on Dynamic Characteristics of Open-type Swirl Injector

Yunjae Chung[1], Jiwook Kim[1] Vladimir Bazarov[2] and Youngbin Yoon[1*]
yunjae85@snu.ac.kr, jiuek@snu.ac.kr, v_g_b_2@yahoo.com and ybyoon@snu.ac.kr

Abstract

Liquid rocket engines are operated with combustors where the high pressure combustion processes are done. This fact led the liquid rocket engine to have a great demand for combustion stability. Due to the great demand, extensive research has been done on combustion instability since 1940s. Despite the large amounts of human and financial resources invested on the research, the combustion instability is still an unsolved problem since it is a very complicated phenomenon which involves many physical phenomena. With these efforts for revealing the cause, efforts for suppressing the symptom of combustion instability has been made also. As the result, the baffle, which is the additional structures installed in the combustion chamber to suppress the combustion instability was developed in the United States and has been used widely in their engines. But these additional structures mean additional mass which is a critical disadvantage for a liquid rocket engine. Also it can spoil the combustion efficiency. But in Russia where the swirl type injectors are used, they tried to suppress the combustion instability by modifying the injectors. By this method, they were able to take out the instability without adding any structures. The study for this method is called injector dynamics. Combustion instabilities in a liquid rocket engine are generated by coupling between the heat release oscillations and the acoustic pressure oscillations in the combustor. This instability from the combustor can affect the injector and the feed line which are in front of the combustor. These transfers worsen the instability of liquid rocket engine. A dynamically properly designed injector can be used to prevent the transfer and suppress the combustion instability. For this reason, studying the dynamic characteristics of the injectors is essential to liquid rocket engine design. An open-type swirl injector was designed in order to investigate the dynamic characteristics of an open-type swirl injector through experimental study. A hydro-mechanical pulsator was installed in front of the manifold of the open-type swirl injector which produces pressure oscillations in the feed line. Pressure in the manifold, liquid film thickness in the orifice and the pressure in the orifice were measured simultaneously in order to understand the dynamic characteristics of the open-type swirl injector. The liquid film thickness was measured by electrical conductivity between two electrodes installed in the orifice as the method proposed by Lefebvre. A direct pressure measuring method was used to calculate the axial velocity of the propellant in the orifice and the mass flow rate through the orifice. These measured and calculated values were analyzed to observe the amplitude and phase differences between the input value in the manifold and the output values in the orifice. As a result, a phase-amplitude diagram was obtained which exhibits the injector's responses to certain pressure fluctuation inputs. The mass flow rate was calculated by the direct pressure measuring method and measured directly through the actual injection to ensure the method's accuracy. Furthermore, the difference between the open-type injector's dynamic characteristics and that of the closed-type swirl injector can be suggested for the selection of injector type during the designing process of liquid rocket engine.

*Corresponding author: ybyoon@snu.ac.kr

Synchrotron X-Ray Diagnostics for Cavitating Nozzle Flow

D. Duke*, A. L. Kastengren, C. F. Powell
Center for Transportation Research
Argonne National Laboratory
Argonne, IL 60439 USA

Poster Abstract

Cavitation plays an important role in the structure of fuel sprays. Visible light imaging and multiphase numerical simulation are the prevalent diagnostic tools. However, there remains an unmet need for higher resolution experimental data to validate models and simulations. To this end, quantitative data on the vapor distribution through the whole volume of the nozzle is required. We present a technique for measuring cavitation inside model nozzles through the use of flat-field line of sight radiography using synchrotron X-rays. Unlike phase-contrast imaging, flat field X-ray radiography operates over short propagation distances to prevent phase interference. Such an approach permits a mean quantitative measurement of the vapor fraction & distribution of cavitation zones at high spatial resolutions integrated through the full extent of the nozzle depth via a simple application of the Beer-Lambert law. Unlike raster-scan techniques, the beam is unfocused and gives a full-field depth-integrated absorption measurement in a time-mean sense rather than a time-resolved measurement at a single point. We present the first preliminary results from these new synchrotron X-ray measurements conducted at Sector 7-BM of the Advanced Photon Source at Argonne National Laboratory, Illinois. Gasoline calibration fluid is delivered at steady pressure from a piston-accumulator system and the flow is imaged with monochromatic X-rays at 8 keV. A high resolution incompressible Large Eddy Simulation of the flow has also been undertaken using the Argonne LCRC Fusion cluster. Regions of intense cavitation measured with radiographic X-ray techniques correspond well to the predicted low-pressure separation zone in the nozzle throat, as expected. Further work will include extending the comparison to compressible flow models which can then be directly validated with the experimental measurement of void fraction.

*Corresponding author: dduke@anl.gov

Atomization Enhancement of Four-Fluid Spray by Electric Field

S. Miyashiro*
Tokushima College of Technology, Itano 779-0108, TOKUSHIMA, Japan
T. Tanaka
Anan National College of Technology, Anan 774-0017, TOKUSHIMA, Japan

Abstract

The technology which atomizes the liquid in the particles of a few microns can be effectively applied not only to the high-efficient, low-pollutive spray combustor but also to the high-efficient manufacturing methods for such as drugs absorbed better in human body, cosmetic powders, electrode materials of new-model batteries, new catalysers and raw materials for new semiconductors by using it as "spray drier".

A new spray drying nozzle[1] (special quadruplet fluid spray nozzle) has been developed by us for drug manufacturing and we have succeeded in producing fine particles of 2 μm diameter of 1/15 ratios to those currently in use.

There are three types of nozzle, namely, circle one(radial flow), flat one(straight flow) and pencil one(coaxial flow). The former two have the same wedge-shaped edges on their nozzle tips. The latter has the additional center air jet.

A developmental study of these supersonic air-assist four-fluid spray nozzles was reported on the 17th Annual Conference on Liquid Atomization & Spray Systems in Zurich[2]. We have succeeded in manufacturing a large quantity of several microns particles (SMD) with these spray drying nozzles. Using the optical visualization techniques, the atomization mechanisms of these nozzles were made clear and their basic characteristics were investigated by measuring the droplet size distributions applying the laser light scattering method.

In case of these nozzles, the two air jets become under-expanded on both edge sides of the nozzle, generate shock and expansion waves alternately on each side and reach the edge tip, where they collide, unite, and spout out while shock and expansion waves are again formed in the mixed jet. When the edge surfaces are supplied with water, the water is extended into thin film by the air jet and intensely disturbed. At the nozzle tip, it is torn into ligaments, which are further atomized afterwards in shock waves. At the spray tip, the friction with ambient air shears the droplets furthermore, and they seem to decrease further in size.

The further experimental works were carried out to enhance the liquid atomization by applying the electric field to the same nozzle sprays. A circular plate electrode with a circular hole in the center for jet to penetrate and also a ring electrode were set in front of the nozzle exit. In this study, the measurements of the droplet diameter and the atomization angle from the spray photographs were made with applied voltage and air pressure as parameter. Either positive or negative potential was applied to above electrodes, and the nozzle was grounded. We show mainly the results of pencil type nozzle for laboratory use.

Experimental results have shown that the atomization angle decreases as the air pressure of the nozzle increases up to about 0.4 MPa and that the average diameter of the droplets decreases as the electrical potential increases. As the air pressure grows larger, the atomization effect of the electric field on the droplet size diminishes accordingly. We assume that the effect of electric field on atomization becomes smaller as the speed of air flow increases

. The characteristics of atomization by this superimposed electric field and the further prospects are described in this contribution.

References

[1] Japanese patent No. 2797080, 1998, etc.

[2] S. Miyashiro, H. Mori and H. Takechi, DEVELOPMENTAL STUDY OF SUPERSONIC AIR ASSIST FOUR-FLUID SPRAY NOZZLES, Proceedings of the 17th Annual Conference on Liquid Atomization & Spray Systems, 2001, Zurich.

* Corresponding author: s.miyashiro@tokuco.ac.jp

The Effect of Index of Refraction on Low Angle Laser Light Scattering

P. A. Corber*, M. Player
National Research Council of Canada, Ottawa, Canada
Andrew.Corber@nrc.ca and Michael.Player@nrc.ca

Abstract

In many practical circumstances the refractive index (RI) of the particles and/or dispersant that are under investigation are unknown. This poses a problem to the experimenter as these values are often necessary software inputs for the diagnostic tool in use. The purpose of this study is to determine the additional uncertainties that occur from entering an incorrect RI into the operating parameters of a commercial laser diffraction instrument. The experiment consists of attempting to measure particles of known size and optical properties with a Malvern Spraytec using incorrect values for RI in the operating software. The particles used were glass microspheres ranging in size from 10 µm to 200 µm. This calibration media has real and imaginary RI's of 1.33 and 0.00 respectively. Erroneous RI values for both the particle and dispersant were varied from 1.00 to 2.00. The results indicate that an incorrect RI has little effect on the measured particle size, provided that the RI of the particle and dispersant are sufficiently different. The acceptable difference was observed to decrease with particle size. Significant uncertainties were observed for a 10 µm particle if the RI of the particle and dispersant were within +/- 0.08, while for particles greater than 140 µm a difference of only +/- 0.01 is sufficient for a measurement within the uncertainty limits of the calibration materials.

* Corresponding author: Andrew.Corber@nrc.ca

Precision Spray Hole Drilling at High Speed with Femtosecond Lasers

M. Mielke*, M. Lerner, M. Greenberg, and T. Booth
Raydiance, Inc., 1450 North McDowell Blvd., Petaluma, CA, USA
mmielke@raydiance.com and mlerner@raydiance.com

Abstract

In a growing number of micro-manufacturing applications, commercial grade femtosecond lasers are providing the required precision and cost efficiency. With femtosecond laser microfabrication tools now accessible to industry, the relevance of femtosecond lasers to modern manufacturing has become clear based on compelling economics, unparalleled precision, and new flexibility with respect to materials and feature geometry.

Efficacy in producing the required precision and cost-efficiency is proven by recent experimental results of high precision, high speed fabrication of gas direct injection (GDI) fuel injector spray holes. Precise control of the GDI fuel spray pattern is a dominant factor in engine performance, and the spray pattern depends directly upon the geometry and surface quality of the machined spray nozzle holes. To maximize fuel economy and minimize emissions, the SAE gas fuel injector steering committee recommends very tight control over:

- Flow
- Spray angle
- Drop size distribution
- Fuel mass distribution
- Spray tip penetration

Control of these parameters in high volume production of GDI fuel injectors is made deterministic and cost-effective by the hole-drilling performance to be revealed in this presentation. Figure 1 shows a scanning electron microscope (SEM) image (300x magnification) of a 200 μm diameter hole drilled through 250 μm thick stainless steel (316L) using a Raydiance R-100 laser and a multi-axis galvo scanner for laser trepanning. The SEM image reveals pristine substrate material (no heat affected zone), negligible taper of the hole (nominally 0°), and excellent surface roughness figure ($R_a < 0.1$ μm). It should be noted that no additional process steps beyond the laser drilling were executed to achieve this result. Laser process characteristics were: 45 μJ, 9 W (200 kHz) on target; helium purge gas (60 psi); and 1.3 s total drilling time.

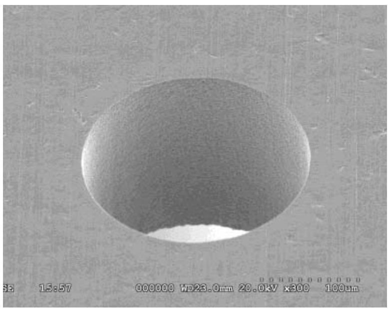


Figure 1 200 μm diameter fuel injector spray hole drilled in 250 μm thick stainless steel in 1.3 seconds using femtosecond laser trepanning.

We analyzed the repeatability of this hole drilling process by measuring the diameters of the hole entrance and exit faces for 99 consecutive iterations of the process. The standard deviations for the entrance and exit face diameters are, respectively, 0.488 μm and 0.233 μm. The maximum variations for the entrance and exit face diameters are, respectively, 1.37 μm and 0.88 μm. Furthermore, using the femtosecond laser trepanning process, a multitude of through-hole geometries can be created. This new flexibility is widening the design space for fuel injector spray hole geometry, and the new geometrical flexibility should enable greater control of spray patterns and improved engine combustion. In this presentation, we shall provide the salient experimental data regarding the hole drilling capability along with the connection between these figures of merit and spray characteristics.

* Corresponding author: mmielke@raydiance.com

Rotary Atomizer Spray in an Air Cross Flow

S. M. Choi*, H. J. Jeong, S. Yun, P. A. Corber
National Research Council of Canada, Ottawa, Canada
csman@jbnu.ac.kr, Sean.Yun@nrc.ca, thrust@jbnu.ac.kr, Andrew.Corber@nrc.ca

Abstract

This poster presents an experimental investigation of fuel sprays generated by a rotary atomizer exposed to a cross flow of air. The effects on the droplet size due to injector rotational speed and air flow velocity were studied. The rotary atomizer was driven at speeds up to 15,000 rpm with a fuel flow rate of 4.2 g/s and cross flows ranging from 10 m/s to 150 m/s. For any given rotational speed, fuel droplet sizes were found to decrease as air cross flows increased, reaching a Sauter mean diameter (SMD) of 8 μm at 150 m/s. At cross flow velocities less than 50 m/s, fuel droplet sizes decreased as rotational speed increased. At intermediate air velocities, fuel droplet sizes were unaffected by the nozzle's rotational speed. Of the two variables studied, each of them is shown to influence the droplet size differently in certain cross flow regimes. This effect can be generalized into three distinct zones based on Weber number defined by orifice diameter and the non-dimensional droplet size.

* Corresponding author: csman@jbnu.ac.kr

Free Thin Liquid Films: Efficient Nonlinear Lubrication Model and Panoramic Diagnostic Set-Up

C. Mehring*
Parker Aerospace, Irvine, CA
carsten.mehring@parker.com

Abstract

Reviewed is a previously developed nonlinear thin-film lubrication model aimed at predicting nonlinear stability, distortion and disintegration of thin liquid films such as those found in inertial fusion reactors, industrial coating systems, spray drying and in atomization systems for jet propulsion engines and heavy fuel-oil burners. Usefulness and versatility of the employed approach with respect to the consideration of different physical effects and their significance for the film distortion and rupture process is outlined. Qualitative and quantitative comparison with other analysis approaches is made. Extension into a general nonlinear thin film dynamics/atomization model or FSIC (Filament Source In Cell) model for use in general purpose CFD solvers is described. The concept of conical fluid film interrogation using panoramic/360° photography for the purpose of model validation is outlined.

---

* Corresponding author: carsten.mehring@parker.com

Preliminary Assessment of Methods for the Characterization of a Coal Slurry Spray

P. A. Corber[*1], M. Player[1], R. Hughes[2], D. Lu[2], B. Anthony[2]
[1]National Reseach Council of Canada, Ottawa, Canada
[2]Natural Resources Canada, Ottawa, Canada
Andrew.Corber@nrc.ca, Michael.Player@nrc.ca, Robin.Hughes@NRCan-RNCan.gc.ca, Dennis.Lu@NRCan-RNCan.gc.ca, Ben.Anthony@NRCan-RNCan.gc.ca

Abstract

This poster presents some preliminary attempts to characterize coal-water slurries (CWS) sprays. The measurements and photographs were taken to provide the data for determining the performance of the analytical equipment prior to constructing a high pressure test devices and to prepare for future testing performed at industrial operating conditions of the CanmetENERGY gasifier. This first round of experiments offer encouraging results, and indicate that detailed information can be collected about the structure of CWS sprays.

* Corresponding author: Andrew.Corber@nrc.ca