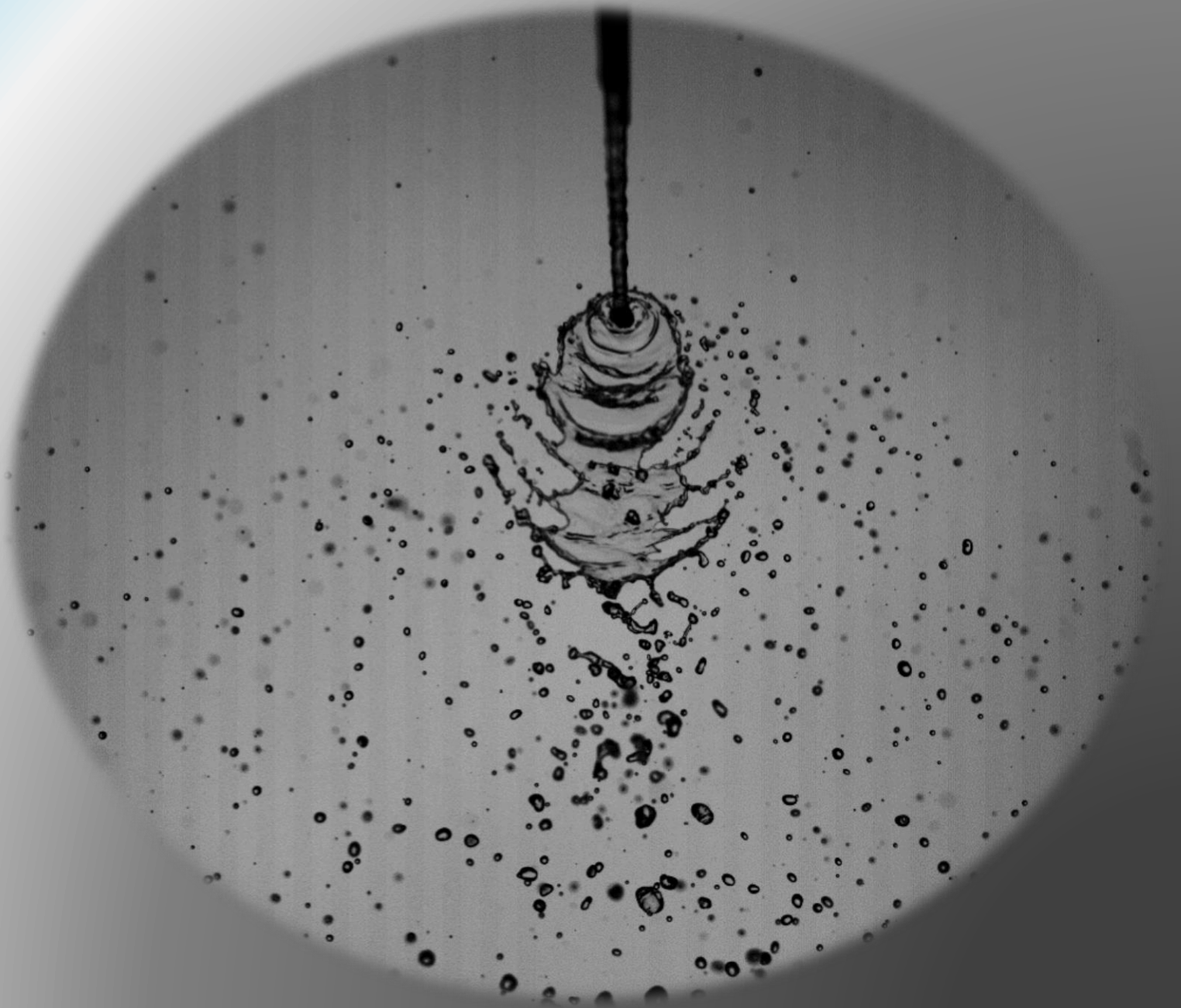


ILASS Asia 2022

22nd Annual Conference of Institute for
Liquid Atomization and Spray Systems~Asia

October 28-30, 2022, Indore, India



ILASS-Asia 2022

book of the abstracts of the

22nd Annual Conference of

**Institute for Liquid Atomization and Spray
Systems-Asia**

organized by



Indian Institute of Technology Indore

supported by



Combustion Institute (Indian section)

October 28-30, 2022, Indore, India

Content

Sponsors and Exhibitors		2
Conference Organization		3
Welcome from the Chief Patron of ILASS-Asia		3
Letter from the Chair of ILASS-Asia 2022		5
Organizers Committee		6
Advisory Committee		7
Conference Information		8
Conference Schedule		9-13
Plenary Talks		14-20
Abstracts of ILASS-Asia 2022		21-112

Sponsors and Exhibitors



DST- SERB, Govt. of India



Council of Scientific &
Industrial Research



Welcome from the Chief Patron



Dr. V. K. Saraswat

Chairman, Combustion Institute (Indian Section).

Full Time Member, National Institute for Transforming India (NITI) Aayog, Govt. of India, New Delhi

Former Director General, DRDO & Scientific Advisor to Defence Minister.

Former Secretary, Department of Defence R & D

डॉ. वी.के. सारस्वत

Dr. V.K. Saraswat

सदस्य

Member

Tele : 23096566, 23096567

Fax : 23096603

E-mail : vk.saraswat@gov.in



भारत सरकार
नीति आयोग, संसद मार्ग
नई दिल्ली - 110 001

Government of India
National Institution for Transforming India
NITI Aayog, Parliament Street,
New Delhi - 110 001

MESSAGE

I am happy to note that Indian Institute of Technology Indore is organising the 22nd Annual Conference of Institute for Liquid Atomization and Spray Systems - Asia, on October 28-30, 2022. The conference being held under the auspices of the Combustion Institute (Indian Section) promotes the interactions among researchers, academicians, scholars and scientists interested in Combustion Science and Technology through various technical meetings, workshops and biennial conferences. I appreciate the efforts of the organizing committee for conducting the conference in hybrid mode which will enable wide reach.

Liquid atomization and sprays play a crucial role in various industrial applications and is one of the most researched topics. The applications range from combustion engines to painting and inhaler sprays. This conference will provide a platform for scientists and engineers to discuss the challenges in analysis, modelling and design challenges in sprays and systems to improved efficiency and reliability.

I congratulate the IIT Indore for hosting the conference and wish them all success in this endeavour.

New Delhi
26.10.2022


(Dr. V. K. Saraswa



Preface



Dr. Devendra Deshmukh
Conference Chair
Indian Institute of Technology Indore

It is my great pleasure to invite you to attend ILASS-Asia 2022, the 22nd Annual Conference of Institute for Liquid Atomization and Spray Systems - Asia, to be held on October 28-30, 2022, at IIT Indore. The ILASS-Asia 2022 is the annual conference, between the members of ILASS- Asia such as ILASS-India, ILASS-Korea, ILASS-Japan, ILASS-Taiwan, and ILASS-China. In this conference, all aspects of atomisation and spray applications including the design of atomiser, liquid breakup, instrumentation, spray systems and various forms of spray applications will be presented.

I believe that the conference will be an effective forum to share new information and technologies on liquid atomisation and spray applications. I am confident that the ILASS-Asia 2022 will contribute to the development of the spray technology, the science in liquid atomisation and its applications.

We have around 80 papers being presented at this conference with delegates participating from China, Japan, Korea, Taiwan and from USA and Europe, in addition to India. I wish all delegates a very productive and enjoyable conference.

Dr. Devendra Deshmukh
Conference Chair,
ILASS-ASIA -2022

Organizing Committee

Chairs



Dr. Devendra Deshmukh
Conference Chair

Indian Institute of Technology Indore



Dr. Anand T.N.C.
Program Chair

Indian Institute of Technology Palakkad

Program Committee



Dr. Madan Mohan Avulapati
Scientific Committee Chair
Indian Institute of Technology Tirupati



Prof. Ashok De
Scientific Committee Associate Chair
Indian Institute of Technology Kanpur



Dr. Hrishikesh Gadgil
Scientific Committee Associate Chair
Indian Institute of Technology Bombay

Local Host Committee



Dr. Eswar Prasad Korimalli
Local Arrangement Chair
Indian Institute of Technology Indore



Dr. I. A. Palani
Local Arrangement Chair
Indian Institute of Technology Indore



Dr. Harekrishna yadav
Local Arrangement Chair
Indian Institute of Technology Indore

Advisory Committee

Name	Affiliation
Dr. V. K. Saraswat	Chairman, Combustion Institute (Indian Section) and Member NITI Aayog
Prof. R. V. Ravikrishna	Indian Institute of Science, Bangalore
Prof. Masataka Arai	Gunma University and Tokyo Denki University
Prof. Min Xu	Shanghai Jiao Tong University
Prof. Youngbin Yoon	Seoul National University
Prof. Soo-Young No	Chungbuk National University
Prof. David Hung	University of Michigan-Shanghai Jiao Tong University Joint Institute, SJTU
Prof. Satya Chakravarthy	Indian Institute of Technology Madras, Chennai
Prof. B. N. Raghunandan	Indian Institute of Science, Bangalore
Prof. P. S. Mehta	Indian Institute of Technology Madras, Chennai
Prof. Dr. V. Raghavan	Indian Institute of Technology Madras, Chennai
Prof. Shamit Bakshi	Indian Institute of Technology Madras, Chennai
Prof. D. Sivakumar	Indian Institute of Science, Bangalore
Prof. Achintya Mukhopadhyay	Jadavpur University
Prof. Mun Soo Chon	Korea National University of Transportation
Prof. Abhijit Kushari	Indian Institute of Technology Kanpur, Kanpur
Prof. Nobuyuki Kawahara	Okayama University
Prof. Fumiteru Akamatsu	Osaka University, Japan
Prof. Chang Sik Lee	Hanyang University
Prof. Jiro Senda	Doshida University
Prof. Shijin Shuai	Tsinghua University
Prof. Wei-Hsin Chen	National Cheng Kung University
Prof. Masatoshi Daikoku	Hachinohe Institute of Technology
Prof. Tsuneaki Ishima	Gunma University, Japan
Prof. Akira Sou	Kobe University, Japan
Prof. Jung Goo Hong	Kyungpook National University
Prof. Sung Wook Park	Hanyang University
Prof. Liguang Li	Tongji University
Prof. Seong Hyuk Lee	Chung-Ang University, Korea
Prof. Seoksu Moon	Inha University, Korea
Prof. Chih-Yung Wu	National Cheng Kung University, Taiwan
Prof. Yueh-Heng Li	National Cheng Kung University, Taiwan



22nd Annual Conference of
Institute for Liquid Atomization and
Spray Systems - Asia



Overview of ILAS-Asia 2022

Title: 22nd Annual Conference on Liquid Atomization and Spray Systems-Asia

Homepage: <https://ilassasia2022.com/>

Dates: 28-30 October 2022

Venue: Indian Institute of Technology Indore, Madhya Pradesh, India.

Official Language: English

Program Venues (IIT Indore,

Program	Venue
Registration	Kalidas Seminar Hall (Vikramshila Lecture Hall complex)
Inauguration	Kalidas Seminar Hall (Vikramshila Lecture Hall complex)
Plenary talks	Kalidas Seminar Hall (Vikramshila Lecture Hall complex)
Cultural Program and Banquet Dinner	Summit Hall 1, Radisson Blu Hotel, Indore
Technical Sessions	Room no. L11, L12 & L13 (Takshshila Lecture Hall Complex)
Valedictory Session	Kalidas Seminar Hall (Vikramshila Lecture Hall complex)

Conference Schedule

Note: All times quoted here and in the detailed program are for India.

Venue for Registration, Inauguration, Plenary talks, Valedictory Session: **Kalidas Seminar Hall**

ILASS-Asia 2022, IIT Indore, India				
Friday, 28th October				
18:00	19:30	Registration at Kalidas Seminar Hall		
19:30	21:30	Dinner at Carbon Building		
Saturday, 29th October				
9:00	10:00	Inauguration		
10:00	10:35	High Tea		
10:35	11:20	Plenary Talk 1: Prof. Cameron Tropea, Optical Measurement of Drops in Flows		
		Chair: Prof. R V Ravikrishna		
Parallel Sessions:		Experimental Atomization-I (Room No. L11)	Modelling and Computations-I (Room No. L12)	Applications-I (Room No. L13)
		Chair: Prof. Kirti Chandra Sahu	Chair: Prof. Ashok De	Chair: Prof. Anubhav Sinha
11:20	11:40	Dominic Sam Sebastian, Sonu K. Thomas, Yazhini V.I. and T.M. Muruganandam: Primary breakup of liquid jet in supersonic crossflow	Nobuyuki Kawahara: Evaluation of ignition characteristics of diesel spray using large eddy simulation	Naga Balaji Putireddy, Ashish Kumar Vishwakarma and Srikrishna Sahu: Characterization of a Turbulent Spray Puff in a Cough Simulator
11:40	12:00	Ashutosh Jena and Avinash Kumar Agarwal: Spray characterisation of an evaporating liquid spray in a cross flow using a multi-hole injector	Utkarsha Sonawane and Avinash Kumar Agarwal: A Numerical Study on Spray Characteristics, Mixing and Evaporation of diesel and dimethyl ether under Diesel Engine like Conditions	Vignesh Kumar D, Ondrej Cejpek, Jan Jedelsky and Madan Mohan Avulapati: Evaluation of Air Assisted Impinging Jet Atomization for CO2 capture
12:00	12:20	Dhananjay Kumar and Avinash Kumar Agarwal: Comparative Assessment of Macroscopic Spray Characteristics for the Direct-Injected Spray of Gasoline, Methanol, and Ethanol in SI Engine-Like Conditions	Ankur Kalwar, Rahul Kumar Singh, Quangkhai Pham, Suhan Park, Sungwook Park and Avinash Kumar Agarwal: Numerical Study of Direct Injection Spray Behavior of Gasoline and Methanol-Gasoline Blends Under Split-Injection Strategy in Engine-Like Conditions	Niraj Kumar, Abhay Mane, Supriya Sarkar, Anand T N C and Shamit Bakshi: Estimation of Mass Flow Rate of Molten Metal through the Melt Delivery Tube in a Gas-atomization System
12:20	12:40	Medipati Chandrasekhar, Deivandren Sivakumar and Govardhan Raghuraman N: Spray characterization study in liquid jet into supersonic cross flow using pulsed laser shadowgraphy	Nilojendu Banerjee and Satyanarayanan Seshadri: A Mathematical Approach for prediction of Spray Heat Transfer for Various Textured Cylindrical Surfaces	Mohamed Nour, Zhe Sun, Hongyu Wang, Mingli Cui, Xuesong Li, David Hung and Min Xu: Influence of Butanol Isomers on Flame Kernel, Late Burn, and PM Emissions of optical SIDI Engine
12:40	13:00	Surendra Soni, Abhishek Gupta, Noorul Huda and Santanu De: Understanding the two-phase mixing pattern inside the twin-fluid atomizer using high-speed visualization	Abhay Mane, Niraj Kumar, Supriya Sarkar, Anand T N C and Shamit Bakshi: Methodology for the Design of Supersonic Annular Gas Nozzle for a Closed-Coupled Gas-Atomization Application	Yi-Kai Chih and Wei-Hsin Chen: Effect of feeding reactants with ultrasonic sprays on hydrogen production in an methanol steam reforming process

13:00	14:00	Lunch	Carbon Building	
14:00	14:45	Plenary Talk 2: Prof. Mahesh Panchagnula: Discovering spatiotemporal structure of sprays through applying tools of data science		
		Chair: Prof. Pramod S Mehta		
14:45	15:30	Plenary Talk 3: Prof. Sungwook Park: Visualization of in-cylinder flow and mixture formation using an optically accessible DISI engine		
		Chair: Prof. Nobuyuki Kawahara		
15:30	15:50	Tea		
Parallel Sessions:		Experimental Atomization-II (Room no. L11)	Modelling and Computations-II (Room no. L12)	Applications-II (Room no. L13)
		Chair: Prof. Sungwook Park	Chair: Prof. Ravikrishna R V	Chair: Prof. Mahesh V Panchagnula
15:50	16:10	Venkatesha, C M Ajay, G Hariseetharam and R Surya Prakash: Experimental Investigation of the Influence of Propeller Downwash on the Spray Characteristics in UASS	Eshan Sharma and Santanu De: Assessment of mixing time-scale models for dilute spray flames using the multiple mapping conditioning approach	Riddhideep Biswas, Anish Pal, Sourav Sarkar and Achintya Mukhopadhyay: Investigation on Assessment and Mitigation of Risk due to Transmission of Respiratory Droplets inside a Classroom using OpenFOAM
16:10	16:30	Abhishek Gupta, Surendra Kumar Soni, Noorul Huda and Santanu De: Effect of preheated swirling air on the spray structure of flow blurring atomizer	Balaji S, Ankur Kumar, Aishwary Pratap and Anubhav Sinha: Surface Breakup of Liquid Jet – Insights from DNS study Surface Breakup of Liquid Jet – Insights from DNS study	Aranyak Chakravarty, Sourav Sarkar, Achintya Mukhopadhyay and Swarnendu Sen: Numerical investigation of water-assisted atomization of molten metal jets for additive manufacturing applications
16:30	16:50	Amitesh Kumar Chaudhary and R. V. Ravikrishna: Spray Characterization of Non-Newtonian liquid jets using a Rotary Atomizer	Mingyun Xie, Shengqi Wu, Bin Zhang, Xiabin Huang and Hong Liu: Numerical investigation of the bifurcating behavior of liquid jet in crossflow	Bhuvanesh Kumar, Ravi Kumar and Akhilesh Gupta: Rewetting On Horizontal Tube Surface During Air-Atomized Spray Cooling: An Experimental Investigation
16:50	17:10	Vivek Sahu, Aman Bakshi and K. P. Shanmugas: Characterization of the swirl velocity field and atomization processes of a gas turbine swirl injectors	Chaoqun Hu, Pengfei Leng, Zhijun Wu, Wenbo Zhao, Chunyu Yang and Liguang Li: Numerical study of the influence of the ambient density on the GDI nozzle internal flow and spray characteristics	Yueh-Heng Li Yueh-Heng Li, Vadlakonda Sirisha and Chia-Wei Chang: The study of coal particle combustion in oblique-impinging methane/air premixed flames
17:10	17:30	Zhao Lyu, Zhimu Ye, Hucheng Zhang, Xinqi Qiao and Zhiwei Jin: Experimental Study on Injection and Spray Characteristics of Coal-Derived Liquid Fuel	Yuanyuan Shen, Chuqiao Wang, Zhixia He and Yanzhi Zhang: Prediction of thermophysical properties of n-dodecane over a wide range of operating conditions	Saewoong Park, Giyoung Park, Taewon Hwang and Sungwock Lee: Investigation of damage type and PM emission characteristics of DPF for diesel vehicle
18:30	19:30	Cultural programme	Summit Hall 1, Radisson Blu Hotel, Indore, Vijaynagar, Indore	
19:30	21:00	Dinner	Summit Hall 1, Radisson Blu Hotel, Indore, Vijaynagar, Indore	
Sunday, 30th October				

9:00	9:45	Plenary Talk 4: Prof. Norihiko Iki, Development of Ammonia Combustion and Hydrogen Energy Technologies for a Decarbonized Society	
		Chair: Prof. Jan Jedelsky	
9:45	10:30	Plenary Talk 5: Prof. Kuo-Long Pan: Droplet combustion of multi-component biofuels doped with nanoparticles	
		Chair: Prof. Cameron Tropea	
10:30	10:50	Tea	
Parallel Sessions:		Experimental Atomization-III (Room No. L11)	Modelling and Computations-III (Room No. L12)
		Chair: Prof. Pramod S Mehta	Chair: Prof. Anubhav Sinha
10:50	11:10	Vivek K, Kundan Kumar Singh, Aditya Saurabh and Lipika Kabiraj: Liquid-sheet break up characteristics of gel propellant simulant	Sudipta Saha, Sourav Sarkar and Aranyak Chakravarty: A computational analysis of hydrodynamic instabilities encountered during co-current flow of air over a thin liquid film
11:10	11:30	Young Soo Yu and Sungwook Park: Experimental study on the spray structure and nozzle tip wetting with LPDI injector under flash boiling	Xinyao Lei, Xiaodong Li, Shuai Zhou and Hongjun Liu: Numerical study on atomization and combustion characteristics of multi-jet gas-liquid pintle injector
11:30	11:50	Kaixiang Li, Di Xiao, Shangning Wang, Shuyi Qiu, Xuesong Li and Min Xu: Experimental investigations of nozzle pitch circle impact on tip wetting	Yaquan Sun, Kaushal Nishad, Yongxiang Li, Louis Dreßler and Amsini Sadiki: Investigation of Spray in Crossflow using Automatic Coupled Volume of Fluid and Lagrangian Particle Tracking within Large Eddy Simulation Framework
11:50	12:10	Yan Lei, Wenbo Jin, Tao Qiu and Chenxi Liu: Experimental study on spray Impingement characteristics of hydrogen / diesel dual fuel	Chih-Yung Wu and Tien-Chiu Hsu: Numerical modelling of fuel spray autoignition using open-source code
12:10	12:30	Vikram Kumar, Shanti Mehra, Hardikk Valera and Avinash Kumar Agarwal: Comparative Study of Microscopic Spray Characteristics of Dimethyl Ether (DME) with Diesel in Atmospheric Ambient Conditions	Sandip Dighe, Gautam Khemraj Kshetri, Hrishikesh Gadgil and Kowsik Bodi: Experimental verification of OpenFoam-based simulations of secondary atomization
12:30	12:50	Surendra Soni and Pankaj Sharadchandra Kolhe: Large-scale flow features in swirl airblast atomization	Kaushal Nishad, Yaquan Sun, Yongxiang Li, Christian Hasse and Amsini Sadiki: Numerical Investigation of Atomization Characteristics of AdBlue Injection in SCR-DeNOx system using VOF-LES
12:50	14:00	Lunch	Carbon Building

14:00	14:45	Plenary Talk 6: Prof. Fu Qingfei: Unsteady Spray of Liquid Propellants		
		Chair: Prof. B N Raghunandan		
Parallel Sessions:		Cavitation and Flash Boiling-I (Room No. L11)	Atomizer Design-I (Room No. L12)	Bubbles and Micro-explosions (Room No. L13)
		Chair: Prof. Madan Mohan A	Chair: Prof. Harekrishna Yadav	Chair: Prof. Shamugadas K P
14:45	15:05	Dongping Shen, Takashi Miwa, Akira Sou, Yoshitaka Wada, Yoshiharu Ueki and Hideaki Yokohata: Single String Cavitation and Swirling Flow in a Nozzle and a Hollow-Cone Spray	Anjith Kumar and Prof. Thirumalachari Sundararajan: Influence of swirl number and air flowrate on the spray characteristics of a micro-channel based rotary atomizer (MCR)	Jisoo Shin and Sungwook Park: Ammonia flash break-up modeling based on bubble dynamics with correlating analysis of surface tension and pressure force
15:05	15:25	Samsu Dlukha Nurcholik, Takashi Miwa, Akira Sou, Mikimasa Kawaguchi, Yuhei Matsumoto, Keiya Nishida and Yoshitaka Wada: Vortex Flow Patterns with Single or Twin String Cavitation in Multi-Hole Mini-Sac Diesel Fuel Injectors and Sprays	Ondřej Cejpek, Milan Malý, Miloslav Bělka, Louis Decanay, Madan Avulapati and Jan Jedelsky: Single Orifice Effervescent Atomizer in simulated Counter-flow Conditions	Ramajayam D and Vadivukkaran M: Study of air bubble entrapped droplet evaporation on oil-coated surfaces
15:25	15:45	Shangning Wang, Shangze Yang, Yijia Zhang, Xuesong Li, David Hung and Min Xu: Breakup Mechanism of Flash Boiling Sprays Based on Rim Structures	Bikash Mahato, Vikram Ramanan, Ganesh Paramasivan and K. P. Shanmugas: Characterization of primary atomization of like on unlike impinging injectors	Abiram Gummala, Himanshu Singh and Madan Mohan Avulapati: A model for prediction of inflight micro explosion in bicomponent fuel droplets
15:45	16:05	Shuyi Qiu, Xiao Di, Xuan Zhang, Shangning Wang, Tongyang Wang, David Hung, Xuesong Li and Min Xu: Experimental investigations of the phase change impacts on flash boiling spray propagations and impingements	Meng Ji, Zhijun Wu and Alessandro Ferrari: Study on real time gasoline-water mixture fuel preparation and spray characteristics based on impingement method	Vivek K, Tikaram Rajput, Aditya Saurabh and Lipika Kabiraj: Droplet evaporation characteristics of nanobubble-liquid suspensions
16:05	16:25	Tea		
Parallel Sessions:		Cavitation and Flash Boiling-II, Other topics (Room No. L11)	Atomizer Design-II (Room No. L12)	Droplets and Impact (Room No. L13)
		Chair: Prof. Eswara Prasad Korimill	Chair: Prof. Anand T N C	Chair: Prof. Devendra Deshmukh
16:25	16:45	Mingli Cui, Zhe Sun, Xuesong Li, Min Xu and David Hung: Study on Laminar Combustion Characteristics of Flash Boiling Spray under Cold Start Environment	Donghui Wang and Yong Huang: Semi-empirical Prediction of SMD Spatial Distribution Downstream of Swirl cups at Different Air Pressures	Arshdeep Singh, Srikrishna Sahu and Dalton Maurya: Influence of Adjacent Droplet-plume Interaction on Spray Characteristics in Slinger Atomizers
16:45	17:05	Chen Li, Zhixia He, Genmiao Guo and Wei Guan: Investigation of string cavitation and air ingestion during injection duration in the scaled-up diesel injector nozzle	Tianhao Pu, Shengqi Wu, Mingyun Xie and Yanshuai Pang: Temperature effect on the breakup characteristics of ultra-high-pressure spray of a single-hole injector	Vishal Jagadale, Devashish Chorey, Yogeshwarnath Mishra, Mats Andersson, Dag Hanstorp and Devendra Deshmukh: Laser induced breakdown in Acoustically Levitated Fuel Droplets
17:05	17:25	Bihe Hu, Jianquan Wang, Zhixia He, Chen Li and Genmiao Guo: Cavitation flow and spray characteristics of gasoline-diesel hybrid fuels in the nozzle of a high-pressure common-rail injector	Mohammed Asad Khan Khan, Hrishikesh Gadgil and Sudarshan Kumar: Development and characterization of self-aspirating liquid fuel spray ejector for microcombustion applications	Gourav Parmar, Vignesh Kumar D, Jan Jedelsky and Madan Mohan Avulapati: Experimental studies on effect of neighbouring droplets on evaporation of aqueous ammonia and

				Monoethanolamine (MEA) droplets
17:25	17:45	Natsuha Okami, Akira Sou and Toshiyuki Saito: Prediction of Cavitation Erosion in Diesel Fuel Injector	Sonu Kumar and Saptarshi Basu: Geometrical Sensitiveness of a High Shear Injector Over Spray Flow Field: Mean and Dynamical characteristics	Xiaoyuan Yang, Bingyao Huang, Yi Zhang and Yuyang Li: Characterizing sputtering behaviors in water/ethanol binary droplet impact on a heated plate
17:45	18:05	Bingyao Huang, Xiaoyuan Yang, Yi Zhang, Haodong Zhang, Wei Li and Yuyang Li: Exploring combustion and atomization characteristics of DMMn and DMMn/ethanol binary droplets: Influence of boiling point difference and light component	Swapnil Soni and Avishek Ranjan: Electromagnetically driven Jet in Liquid Metal Electrode	Quangkhai Pham, Mengzhao Chang, Byungchul Choi, Sungwook Park, Avinash Kumar Agarwal and Suhan Park: Visualization and analysis of CNG wall-impingement spray characteristics under various injection pressures and impingement geometry conditions
18:05	18:25	Kirti Chandra Sahu: Fragmentation and size distribution of raindrops in airstreams	Jacob Koshy Mulamootil, M Manish, Boggavarapu Prasad and R V Ravikrishna: Insights to the Near-nozzle Flow Characteristics of Pressure Swirl Atomizers used in Aero Engines	Tinglan Tang, Tai Jin and Gaofeng Wang: Numerical study on splash of oblique hollow droplet impact on a liquid film
18:25	19:00	Valedictory session	Kalidas Seminar Hall	
19:00	21:00	Dinner	Carbon Building	

PLENARY TALKS

PLENARY TALK – 1



Prof. Cameron Tropea

TU Darmstadt, Germany

Title: “Optical Measurement of Drops in Flows”

PLENARY TALK – 2



Prof. Mahesh Panchagnula

Department of Applied Mechanics, Indian Institute of Technology Madras

Title: “Discovering spatiotemporal structure of sprays through applying tools of data science”

Abstract

Data Science techniques have helped develop deep insights into several physical phenomena. Sprays cannot be an exception since the experimental tools that are in common use on sprays are 'big data' generators. We will first (briefly) discuss the set of data science tools available to fluid mechanicians with examples of their applications to sprays. We will then discuss the application of time series analysis tools towards developing insights into PDPA data. From the PDPA data analysis of air-blast and pressure swirl sprays, we will show that every single point data contains information of the class of atomizer from where the sprays originated. We will also show that it is possible to produce a data-science driven synthetic drop injection model that will preserve the microstructure of the spray, for use in CFD simulations. Finally, we will discuss the exciting directions that are being revealed in applying data science and machine learning models to spray data.

PLENARY TALK – 3



Prof. Sungwook Park

School of Mechanical Engineering, Hanyang University, Seoul,
Republic of Korea

Title: “Visualization of in-cylinder flow and mixture formation using an optically accessible DISI engine”

Abstract

This paper presents an experimental and numerical study on the in-cylinder flow and mixture formation process of an optically accessible direct injection spark ignition engine equipped with a center-mounted injector. A novel optical engine was designed for full access to the combustion chamber by using a quartz liner and oilless system. The cross-correlation PIV method was used to visualize the in-cylinder flow quantitatively and the spray development processes were visualized using the high-speed camera. In addition, CFD simulations using the CONVERGE were performed to analyze the experimental results. We swept the engine operating parameters to understand their effects on the in-cylinder flow development.

PLENARY TALK – 4



Prof. Norihiko Iki

The National Institute of Advanced Industrial Science and Technology
(AIST)

Title: “Development of Ammonia Combustion and Hydrogen Energy
Technologies for a Decarbonized Society”

Abstract

As with global decarbonization efforts Japan research and development is underway to achieve carbon neutrality. In the Sixth Strategic Energy Plan, the key theme is to show the path of the energy policy to realize carbon neutrality by 2050 and reduce greenhouse gas emissions by 46% in FY 2030 from its FY 2013 levels. 1% Hydrogen/ammonia will be positioned in power generation mix in FY2030. The Government of Japan has begun demonstrating the supply chain of various hydrogen carriers, and liquid hydrogen, MCH, and ammonia projects are underway. Fukushima Renewable Energy Institute, AIST (FREA) works on hydrogen energy technologies, especially R&D of hydrogen carriers including ammonia fuel. Ammonia is liquified gas similar to LPG and no carbon fuel, so it is expected as an alternative fuel to fossil fuels. FREA tries ammonia gas combustion and ammonia spray combustion for gas turbine engine, etc.

PLENARY TALK – 5



Prof. K. L. Pan

Mechanical Engineering, National Taiwan University, Taiwan

Title: “Droplet combustion of multi-component biofuels doped with nanoparticles”

Abstract

Single droplet burning could be treated as a unit that is key to understanding the fundamental features of spray combustion in liquid-fuelled engines. Moreover, in the environment of reduced gravity, the configuration of spherical symmetry has been adopted as a paradigm to reveal the essential behaviours of droplet combustion. By means of a small drop tower built in our lab, together with a technique allowing for free-falling droplets, we have investigated the combustion of droplets made of various biofuels and mixtures, even doped with nanoparticles. Different modes of disruptions including micro explosions have been studied, demonstrating specific ways toward or against the occurrence. Comprehension of the roots would pave a route to various applications underlying the combustion in complex geometries of engines.

PLENARY TALK – 6



Prof. Qing-Fei Fu

School of Astronautics, Beijing University of Aeronautics and Astronautics,
Beijing, China

Title: “Unsteady Spray of Liquid Propellants”

Abstract

Unsteady atomization is a common phenomenon in the liquid rocket engines, especially when the combustion instability occurs. Understanding the process, characteristics and mechanism of unsteady atomization is critical to understanding the role of atomization in unstable combustion. This presentation describes the research progress of unsteady atomization in recent years. We divide the unsteady atomization into two kinds of problems: injector dynamics caused by pulsation of incoming supply pressure, and liquid injection in the standing wave sound field downstream of the injector plate. It focuses on the transfer relationship between the pulsating pressure and flow rate of the nozzle, as well as the effects of the sound field strength, the position of injector in the sound field on the atomization.

Abstracts

Experimental Atomization - I

Primary Breakup of Liquid Jet in Supersonic Crossflow

Dominic.S. Sebastian^{1*}, Yazhini V.I.¹, T.M. Muruganandam¹, Prokhorov, A. N ^{2,3}., and Son, E. E ^{2,4}

¹Indian Institute of Technology, Madras, Tamil Nadu, 600036 India

²Moscow Institute of Physics and Technology, Dolgoprudny, 141700 Russia

³Central Institute of Aviation Motors, Moscow, 111116 Russia

⁴Joint Institute for High temperature, Russian Academy of Sciences, Moscow, 125412 Russia

Abstract

Liquid jet in supersonic crossflow is used mainly in Scramjet combustors for fuel injection. Primary breakup of the jet is influenced by the combined effect of Rayleigh-Taylor and Kelvin-Helmholtz instability. Because of the high velocity gradient between liquid-gas interfaces, droplets get stripped off from the surface waves leading to surface breakup. Primary breakup region of a liquid jet in supersonic crossflow is very difficult to visualize experimentally because of the dense cloud of droplets formed from surface breakup, obstructing the breakup region. A new experimental approach using inclined front light imaging was adopted to overcome the above said challenges to visualize the primary breakup region and other spray structures, for three different momentum flux ratios. Proper Orthogonal Decomposition analysis showed that there was flapping mode of the jet, which was dominant along with the convective mode.

*Corresponding author: dominicsam66@gmail.com

Spray Characterisation of an Evaporating Spray in a Cross-flow Using a Multi-Hole Injector

Ashutosh Jena, Avinash Kumar Agarwal*

Engine Research Laboratory, Department of Mechanical Engineering

Indian Institute of Technology Kanpur, Kanpur-208016, India

Abstract

Liquid spray finds widespread applications in engineering systems. Fundamental aspects of atomisation have been crucial in designing spray atomisers for various applications. Liquid jet in a cross flow is a common physical condition for several engineering applications, including gas turbine and internal combustion (IC) engines. The momentum ratio of the cross flow and the liquid jet is a governing parameter in this study. However, in the case of port fuel injection of evaporating fuels like gasoline, the downstream mixing is expected to be affected by wall impingement and evaporation of spray droplets. Most studies in this area have focused on the spray atomisation behaviour of a single jet emerging from a single-hole injector, unlike the multi-hole injectors used in IC engines. The location and orientation of the injector play a crucial role in determining engine performance and emissions from Spark Ignition (SI) and dual-fuel Compression Ignition (CI) engines. This study focuses on the spray atomisation and downstream mixing of gasoline injected from a multi-hole port fuel injector (PFI) in a cross flow. In this study, the PDI technique was employed to get the droplet size and velocity distributions for the flow confined in a circular duct with a diameter similar to the intake port of the dual-fuel CI engine. The flow velocity was adjusted to simulate the flow conditions of a light-duty diesel engine operating at a speed of 3000 rpm. The spray evolution was analysed using a high-speed imaging technique.

Keywords: PFI, Spray atomisation, cross-flow, SMD

Comparative Assessment of Macroscopic Spray Characteristics for the Direct-Injected Spray of Gasoline, Methanol, and Ethanol in SI Engine-Like Conditions

Dhananjay Kumar, Avinash Kumar Agarwal*

Engine Research Laboratory, Department of Mechanical Engineering

Indian Institute of Technology Kanpur, Kanpur-208016, India

Abstract

In the last few decades, stringent emission norms have forced the engine research community and OEMs to develop new spark ignition engine technologies, such as variable valve lift, turbocharging, and direct injection spark ignition (DISI) engines. For further development, greater control of parameters such as in-cylinder air motion, spray characteristics, injection, and ignition events are required. Spray characterisations are crucial for understanding the mixing phenomena in heated and pressurised ambient conditions. Spray pattern, fuel injection pressure (FIP), rate shape, and thermodynamic conditions of the combustion chamber play a vital role in the mixture preparation. The present study uses Mie-Scattering techniques to examine spray structures of fuels like methanol and ethanol, which are of great interest to DISI engines. Three different temperatures of 50, 100, and 200°C and two ambient pressures, 4 bar and 8 bar, were considered to simulate typical engine-like conditions. It was observed that the initial chamber conditions greatly influenced the spray structure. Spray collapse was lesser for alcohol than gasoline. Spray penetration length increased with an increasing FIP.

Keywords: Macroscopic Spray Characteristics, Methanol, Direct Injection Spark Ignition, Mie-Scattering Technique.

*Corresponding author: akag@iitk.ac.in

Spray characterization study in liquid jet into supersonic cross flow using pulsed laser shadowgraphy

Chandrasekhar Medipati *, S. Deivandren, R.N. Govardhan

Affiliation

Indian Institute of Science

Bangalore, Karnataka, 560012, India

Abstract

The present work focuses on the characterization of spray droplets formed from the transverse injection of a liquid jet into the supersonic crossflow of freestream Mach number 2.5. High resolution temporally uncorrelated microscopic images of the spray obtained using pulsed laser shadowgraphy were used to deduce the salient details of spray morphology and the quantitative measurements of size and velocity distribution of spray droplets. The experiments were conducted at different values of momentum flux ratio (J , defined as the ratio of momentum flux of the transverse jet to the crossflow freestream) and the images were recorded at various spatial locations in the spray plume. The image analysis revealed that, at locations away from the injection point, the spray is free from ligaments and droplet clusters and the droplets are spherical in shape. The spray statistics viz. Sauter mean diameter (SMD), mean streamwise and cross streamwise velocities, and their variations were calculated using Particle/Droplet Image Analysis (PDIA) technique. Using probability density functions, the range of droplet sizes and velocities at different spatial locations were estimated. The noteworthy observation from the study is that the variations of SMD and mean streamwise velocity in the transverse direction follow 'S' and 'C' shaped trends, respectively, which agree with the existing literature data obtained using phase Doppler (PDA) technique. These trends are invariant with streamwise location ($x/D > 60$) and J . Further, the details of secondary atomization are shown in terms of droplet Weber number.

*Corresponding author: mchandrashek@iisc.ac.in

Understanding the two-phase mixing pattern inside the twin-fluid atomizers using high-speed visualization

Surendra Kumar Soni, Abhishek Gupta, Noorul Huda, Santanu De*

Dept. of Mechanical Engineering, Indian Institute of Technology Kanpur, India

Abstract

The internal flow structure in the internally mixed twin fluid atomizer is essential towards understanding the spray formation, as the two-phase mixing regime inside the atomizer controls the external spray structure and, thereby, the final drop size and velocities. This category of atomizers uses atomizing air to enhance the two-phase mixing. The present study uses a transparent fuel injector to investigate internal two-phase flows resulting from interactions between liquid and atomizing air. The fuel injector replicates the counter nozzle concept, which yields a much greater surface-to-volume ratio than conventional air-blast injectors. In order to improve the two-phase mixing and enhance secondary atomization, this configuration is augmented based on a coaxial swirler in the atomizing air stream. This hybrid design innovatively promotes vigorous interaction between atomizing air and ligaments/droplets originating from primary atomization. This study compares the internal mixing process in a counterflow atomizer and that of a flow-blurring atomizer as the baseline case for the same flow conditions. Air-liquid interactions inside the injector are captured using high-speed shadowgraphy images. The captured images are analyzed to examine the importance of air entrainment inside the liquid tube for a detailed understanding of the different atomization regimes and to establish the relationship between the two-phase internal mixing and external spray structure.

*Corresponding author: sde@iitk.ac.in

Modelling and Computations - I

Evaluation of ignition characteristics of diesel spray using large eddy simulation

Nobuyuki Kawahara*

Okayama University

Okayama, 700-8530 JAPAN

Abstract

Compression ignition (CI) engine will continue to play an important role in heavy duty engine applications. The fuel injection pressure in CI engines has been increased to improve atomization, and to achieve higher thermal efficiency under high supercharging and high EGR. Understanding of the atomization and ignition characteristics of fuel injection spray under higher pressure and temperature conditions can contribute to improving the performance of CI engines. The purpose of this study is to evaluate the ignition characteristics of diesel spray in a rapid compression machine (RCM) more precisely. Spray behavior and spray combustion were simulated by Generalized Tank and Tube (GTT) code considering chemical kinetics with LES (Large Eddy Simulation). Multidimensional CFD simulations of diesel spray combustion using the developed elementary reaction scheme with LES were carried out under RCM experimental conditions with 900 K of ambient temperature and 4.1 MPa of ambient pressure with different injection pressure conditions. In this chemical reaction mechanism, n-decane, iso-octane, methylcyclohexane (MCH), and toluene are used to simulate as n-paraffin, iso-paraffin, cycloalkane, and aromatic hydrocarbons in light oil fuels. LES calculations with multi-component fuels showed good agreement with the experimental results. As the injection pressure is increased, the ignition timing becomes earlier and the ignition position moves downstream of the spray. LES calculation simulated the vortex formed around the spray axis and confirmed that the spray was ignited inside the vortex. It is considered that evaporation is promoted by the atomization of droplets at the outer edge of the spray, and that the turbulent eddies promote mixing, which results in the ignition.

*Corresponding author: kawahara@okayama-u.ac.jp

Simulation Study of Spray Atomization and Evaporation Characteristics of Di-Methyl Ether to Improve Cold-Start Issues of Diesel Engines

Utkarsha Sonawane, Avinash Kumar Agarwal*

Engine Research Laboratory, Department of Mechanical Engineering

Indian Institute of Technology Kanpur, Kanpur-208016, India

Abstract

Diesel engines face cold-start issues at lower ambient temperatures, resulting in higher emissions and combustion instability. This study provides insights into tackling cold-start issues faced by diesel engines. Dimethyl ether (DME) is a new generation fuel that can overcome the cold-start issues of diesel engines. It has a higher cetane number (CN) and can efficiently lead to superior atomization. This computational study compares diesel, and dimethyl ether (DME) sprays in a constant volume spray chamber (CVSC) maintained at 626 K. Comparison of enhancement in spray characteristics due to adaptation of DME is undertaken in this study. This study is based on the Eulerian-Lagrangian approach adopted in the LES framework. Cold-start conditions were replicated by lowering the fuel temperatures to 250 K to simulate freezing outside conditions. The liquid spray penetration obtained by simulations matched well with the experimental results for diesel under cold ambient conditions. It was observed that fuel properties greatly influenced the spray atomization and evaporation characteristics. Lower penetration of DME than diesel was because of faster atomization of DME. This was also due to lower viscosity and higher evaporation rate. SMD of spray droplets was much lower for DME due to its lower viscosity, surface tension, and density. Since DME has a lower density, deceleration of droplets is expected to be more downstream for DME sprays. The vapour penetration length of both fuels was almost the same; however, the vapour mass fraction was higher for DME than diesel under cold ambient conditions. DME spray showed better atomization and improved evaporation of DME spray droplets under cold ambient conditions. Hence, using DME as fuel is useful for tackling cold-start issues.

Keywords: Cold-Start, Simulation, Diesel engines, Large Eddy Simulation, Dimethyl ether

*Corresponding author: akag@iitk.ac.in

Numerical Study of Direct Injection Spray Behavior of Gasoline and Methanol -Gasoline Blends Under Split-Injection Strategy in Engine-Like Conditions

Ankur Kalwar^a, Rahul Kumar Singh^a, Quangkhai Pham^b, Suhan Park^c, Sungwook Park^d,
Avinash Kumar Agarwal^{a,*}

^aEngine Research Laboratory, Department of Mechanical Engineering,
Indian Institute of Technology Kanpur, Kanpur-208016, India

^bDepartment of Mechanical Engineering, Graduate School of Chonnam National University,
77 Yongbong-ro, Buk-gu, Gwangju, 61186, Republic of Korea

^cSchool of Mechanical and Aerospace Engineering, Konkuk University,
120 Neungdong-ro, Gwangjin-gu, Seoul 05029, Republic of Korea

^dSchool of Mechanical Engineering, Hanyang University, Seoul 04763, Republic of Korea

Abstract

Several studies have shown that the split-injection strategy has improved the shortcomings of the high particulate matter (PM) emissions and cycle-to-cycle variations in homogeneous and stratified combustion modes of Gasoline Direct Injection (GDI) engines. However, the spray behaviour under the split injection strategy is poorly understood. This study uses CFD simulations in Converge software to investigate the spray characteristics of gasoline and methanol-gasoline blends under a split-injection strategy. The simulation studies were performed for a multi-hole GDI injector in a constant volume spray chamber (CVSC), replicating the ambient conditions similar to a GDI engine's homogeneous and stratified combustion modes. Based on the literature, appropriate models for simulating different spray phenomena, such as spray break-up, collision, and coalescence, were used. The CFD model was validated using the experimental spray penetration length available in the engine combustion network (ECN) database. The results showed that the split injection reduced the Sauter mean diameter (SMD) of fuel spray droplets and liquid fuel mass content compared to a single injection case. The dwell time (DT) of 2 ms was found suitable for homogeneous mode conditions, while its effect was insignificant in stratified mode conditions. Further, a split ratio (SR) of 80:20 resulted in smaller SMD during the injection period and a higher fuel evaporation rate. The effect of methanol addition was also explored under the split injection mode. The impact of fuel properties of methanol on spray characteristics was considerable in split-injection mode also. M85 showed higher deviations in the liquid spray penetration, SMD, and liquid spray mass content than G100.

*Corresponding author: akag@iitk.ac.in

A Mathematical Approach for prediction of Spray Heat Transfer for Various Textured Cylindrical Surfaces in nucleate boiling regime

Nilojendu Banerjee *, Satyanarayanan Seshadri*

Department of Applied Mechanics

Indian Institute of Technology Madras Chennai-600036, India

Abstract

Spray cooling is one of the most promising technologies for high heat transfer phase change (boiling) applications. Many industrial systems such as process heat exchanger tubes, cooling towers, fuel injection in gas turbine combustion, internal combustion engines, thermal spray coating etc., need significant amount of heat removal from surfaces with high heat flux. Several literature studies provide empirical correlations to aid the development of heat removal techniques from these surfaces. These models partially capture the effect of various parameters of spray cooling within a narrow range. In this study, a new analytical model is proposed to predict the interaction mechanism between spray and the heated surface in the nucleate boiling regime. Firstly, spreading of a single droplet on a heated surface is studied, which is then used to model the mutual interactions of spraying droplets with the surface. In order to capture a realistic spray behaviour, log-normal distribution of droplets inside the spray is considered. Further, surface roughness is incorporated to analyse the effect of active cavities on nucleate boiling heat transfer. Predictions from the model shows good agreement with well-established literature. We find that the wetting efficiency increases with spray average mass flux, surface temperature, overall surface area and the surface texture while a decreasing trend is observed with respect to droplet median diameter. Similar behaviour is observed for spray averaged Nusselt number based on diameter of the cylinder with previously mentioned parameters. This analysis is crucial for selecting suitable materials to develop a compact heat exchanger by optimizing the heat transfer rate.

*Corresponding author: satya@iitm.ac.in

Methodology for the Design of Supersonic Annular Gas Nozzle for a Closed-Coupled Gas-Atomization Application

Abhay Mane^{1,*}, Niraj Kumar¹, Supriya Sarkar², T.N.C. Anand^{1,3}, Shamit Bakshi^{1,†}

¹Department of Mechanical Engineering, IIT Madras, Chennai 600036, TamilNadu, India

²Alleima India Pvt. Ltd., Pune 411012, Maharashtra, India

³Department of Mechanical Engineering, IIT Palakkad, Palakkad 678557, Kerala, India

Abstract

The gas-atomization of molten metal is a widely used technique for producing metal powders which are used in hot isostatic pressing, 3-D printing, metal injection molding, etc. It is a complex process involving the interaction between the molten metal stream and the high-speed inert gas flow. The transformation of the molten metal stream into metal powder is governed by high and rapid heat and momentum transfer between the high-speed inert gas phase and the falling molten metal stream. The gas nozzle is the heart of the gas-atomization system. The gas nozzle's geometry, profile, orientation, and location relative to the melt delivery tube (MDT) play a significant role in the performance of the gas nozzle for efficient powder production. Well-known techniques used in rocket engine nozzle like a Method of Characteristics (MOC) have been used to design the gas nozzle for the gas-atomization application. No specific method has been developed to design the gas nozzle geometry considering gas-atomization parameters. In the present work, a methodology is developed to design the geometry of an annular slit (AS), inclined, convergent-divergent (CD) gas nozzle specifically for the closed-coupled (CC) gas-atomization application. For the same, a code is written in MATLAB[®] using gas dynamics concepts to generate a data-field of geometric and performance parameters of all possible gas nozzles for given user inputs to the code. The code has considered thermo-physical properties of the gas and the molten metal along with flow and geometric parameters related to the gas-atomization application. The code predicts the mass median size (D_{50}) of particles for a given molten metal flow conditions using empirical correlations for each gas nozzle geometry obtained from the code. This methodology is applicable to any gas and molten metal.

Keywords: gas-atomization, annular slit (AS), convergent-divergent (CD), closed-coupled (CC), gas dynamics, mass median size (D_{50}).

*Corresponding author: me19s011@smail.iitm.ac.in

†Corresponding author: shamit@iitm.ac.in

Applications - I

Characterization of a Turbulent Spray Puff in a Cough Simulator

Ashish Kumar Vishwakarma^{a,*}, Naga Balaji Putireddy^a, Srikrishna Sahu^a

^a Thermodynamics and Combustion Engineering Laboratory, Department of Mechanical Engineering, I.I.T Madras.

Abstract

The unforeseen outbreak of COVID-19 has infected over 450 million people across the globe. It is well known that the droplets exhaled during the respiratory action are potential carriers of pathogens, which are primary source of disease transmission. This demands detail investigation into the dynamics of the ejected turbulent gas cloud carrying droplets. In this context, the artificial cough simulators allow repeatability, disinfection, and safety. In this work, we aim to investigate the characteristics of droplets ejected from a cough simulator which issues a turbulent puff of air, laden with aerosols as well as droplets that mimics a human coughing process. The Laser Sheet Imaging (LSI) technique is employed to visualize the Mie-scattering images of the droplets using a high-speed camera. The time development of the ejected spray is also demonstrated. Accordingly, the spray velocity is measured by considering pair of successive images. The droplet size is measured using the Interferometric Laser Imaging for Droplet Sizing (ILIDS) technique. A theoretical model for droplet transport and evaporation is described, and the predictions are compared with the experimental data which demonstrates the role of collective evaporation of the droplet cloud on spatial evolution of droplet size.

*Corresponding author: me21s062@smail.iitm.ac.in

Evaluation of Air Assisted Impinging Jet Atomizer for CO₂ capture

D. Vignesh Kumar^{1*}, O. Cejpek², J. Jedelsky², M. M. Avulapati¹

¹Department of Mechanical Engineering, Indian Institute of Technology Tirupati, Andhra Pradesh, India

²Faculty of Mechanical Engineering, Brno University of Technology, Brno, Czech Republic

Abstract

In this present work, the suitability of air assisted impinging jet atomization for Carbon dioxide (CO₂) capture using Monoethanolamine (MEA) is evaluated. The desired spray characteristics such as droplet size and distribution for CO₂ capture by MEA solution is noted from the literature. The suitable operational conditions in terms of gas and liquid injection pressure and Gas-to-liquid ratio (GLR) to achieve the same efficiently is evaluated. The atomizer configuration consisted of two liquid jets inclined at an angle of 90° to each other with a gas jet placed above the impinging point at an angle 45° and in between both the liquid jets. A mixture of glycerol, water and tertiary butanol at the desired composition is used to achieve the similar physical properties of MEA. The spray experiments are performed with this surrogate of MEA at different liquid flow rates and at different GLRs. The droplet size characteristics at 50 mm downstream the impingement point are estimated with the help of Malvern Panalytical's Spraytec laser diffraction system. The results showed that a fine droplet size of the order of 50 µm with a sparse droplet density required for CO₂ capture can be achieved even at low liquid injection pressure conjunction with GLR of around 7 %.

*Corresponding author: me19d509@iittp.ac.in

Estimation of Mass Flow Rate of Molten Metal through the Melt Delivery Tube in a Gas-atomization System

Niraj Kumar¹, Abhay Mane¹, S. Sarkar³, T.N.C. Anand^{1, 2}, S. Bakshi^{1*}

¹Department of Mechanical Engineering, IIT Madras, Chennai 600036, India

²Department of Mechanical Engineering, IIT Palakkad, Palakkad 678557, India

³Allema India Pvt. Ltd., Pune 411012, India

Abstract

Gas-atomization is a process of manufacturing metal powders. In this process, molten metal is atomized using highspeed inert gas jet from a gas nozzle, eventually solidifying to form metal powder. The molten metal flows from the crucible into the atomization chamber through a tube called the melt delivery tube (MDT). The relative position of MDT and the gas nozzle, and the protrusion of the MDT into the chamber determine whether aspiration (suction) or pressurization (back pressure) conditions occur at the tip of the MDT. A pressurization condition (pressure above atmospheric) imposed in the crucible on the melt increases the mass flow rate, whereas pressurization condition at the tip of the MDT is expected to reduce the molten metal flow rate and thus increase the gas-to-metal ratio (GMR), resulting in finer particles. In this study, determination of the mass flow rate of molten metal through the MDT from the crucible, and temperature measurement on the outer surface along the length of the MDT were done for different superheat of the molten metal. Experimental investigations reveal that the mass flow rate of the molten metal gradually decreases after a sudden rise at the start, and abruptly stops at the end. A significant change in the surface temperature along the length of the MDT also occurs during the flow of the molten metal through the MDT.

Keywords: gas-atomization, metal powder, melt flow rate, superheat

*Corresponding author: shamit@iitm.ac.in

Influence of Butanol Isomers on Flame Kernel, Late Burn, and PM Emissions of optical SIDI Engine

Mohamed Nour^{1,2*}, Zhe Sun¹, Hongyu Wang¹, Mingli Cui¹, Xuesong Li^{1*}, David L.S.

Hung^{1,3}, Min Xu¹

School of Mechanical Engineering, Shanghai Jiao Tong University, Dongchuan Road 800, Shanghai, 200240, China.

Mechanical Engineering Department, Benha Faculty of Engineering, Banha University, Benha 13512, Egypt.

University of Michigan-Shanghai Jiao Tong University Joint Institute, Shanghai Jiao Tong University, Shanghai, 200240, China.

Abstract

Butanol has a high potential as an alternative renewable fuel for spark-ignition direct-injection (SIDI) engines. Butanol isomers have different flame-spreading behaviors, strongly related to their flame kernel initiation (FKI). Stable FKI is crucial for combustion stability and engine efficiency. The relationship between the butanol molecular structure and FKI stability needs investigation. This study investigates the FKI from ignition to CA01 (crank angle where 1% of the total heat is released) using an optical SIDI engine and a high-speed camera. Four fuel mixtures of 30% butanol isomers (1-butanol (n), 2-butanol (s), isobutanol (i), tertbutanol (t)) and 70% toluene reference fuel (TRF) were tested. The results elucidate that TRF-1-butanol (TRF-n) has a more stable FKI than TRF. The FKI stability ranked as TRFn > TRF > TRF-isobutanol (TRF-i) > TRF-2-butanol (TRF-s) > TRF-tertbutanol (TRF-t). More than 50% of TRF-t cycles showed delayed or no FKI. The more stable the FKI, the better the flame circularity and the lower the cyclic variations. TRF-n has the highest apparent flame speed, then decreased for TRF, TRF-i, TRF-s, and TRF-t, respectively. The flame laminar-to-turbulent transition was detected when the flame area ratio ranged from 0.05 to 0.1, which was at least 5 crank angle degrees (CAD) earlier than CA01. TRF, TRF-n, and TRF-t showed higher diffusion flame intensity and particulate number (PN) emissions than TRF-s and TRF-i. Butanol with a linear chain molecular structure and more internal C-H bonds offers better stable FKI than branched-chain structure butanol.

*Corresponding author: mohamed.nour@sjtu.edu.cn, xuesonl@sjtu.edu.cn

Effect of feeding reactants with ultrasonic sprays on hydrogen production in an methanol steam reforming process

Yi-Kai Chih¹, Wei-Hsin Chen^{1,2, 3*}

¹Department of Aeronautics and Astronautics

National Cheng Kung University

Tainan, 700, Taiwan

²Research Center for Smart Sustainable Circular Economy

Tunghai University

Taichung, 407, Taiwan

³Department of Mechanical Engineering

National Chin-Yi University of Technology

Taichung, 411, Taiwan

Abstract

Methanol solution, the main reactant of a hydrogen-producing methanol steam reforming (MSR) process, was fed into the system with ultrasonic sprays to speed up its reaction with a copper-based catalyst. Generally, hydrogen productivity is affected by such factors as reaction temperature, steam/carbon (S/C) ratio, and carrier gas flow rate. It is thus reasonable to expect an adjustment in the method of reactant feeding to serve to improve methanol conversion and hydrogen yield, our main indicators for hydrogen productivity. Actual experimental findings show a considerable increase in both of the indicators (by 41% and 28%, respectively) with the use of ultrasonic sprays, which helped to feed methanol solution in the form of evenly and highly dispersed droplets onto 16 g and 32 g of the catalyst at 250 °C. This suggests the potential of reducing the use of heavy metals as catalysts in the MSR process while achieving high hydrogen productivity with the use of ultrasonic sprays.

Keywords: Hydrogen production and yield; Methanol steam reforming (MSR); Cu-Zn catalyst; Clean energy; Sustainable energy.

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

Experimental Atomization - II

Experimental Investigation of the Influence of Propeller Downwash on the Spray Characteristics in UASS

Venkatesha, C M Ajay, D Harshad Prakash, G Hariseetharam, R Surya Prakash*

Department of Mechanical, Materials and Aerospace Engineering

Indian Institute of Technology Dharwad

Dharwad, Karnataka, 580011, India

Abstract

Recently, Unmanned Aerial Spray Systems (UASS) are being widely used for precision agriculture. UASS reduces operator exposure to chemical sprays, covers large areas in a short duration, and is flexible, economical, and portable. The spray generated by UASS is greatly influenced by downwash flow characteristics formed by the propeller blades. As the distance from the nozzle exit and the magnitude of the thrust are changed, the airflow under the rotor influences the motion of spray particles. Knowledge of how the downwash and spray pattern produced by UASS is influenced by operating variables is essential to increase the efficacy of the spraying process. In this study, spray mass distribution, spray cone angle and velocity distribution are quantitatively evaluated both in the absence and presence of the downwash - at different rotor speeds. Under the influence of downwash, spray patterns exhibit tilted distributions - in comparison to the base patterns - along the direction of propeller rotation. The effect of downwash on the velocities of spray droplets at different heights from the nozzle exit are measured and it is observed that the axial velocities drastically decrease in the absence of downwash. However, in the presence of downwash, the spray droplets are expectedly accelerated along the flow direction and exhibit higher axial velocities. These results show the importance of characterizing the resultant spray to achieve greater spraying efficacies. The present study throws more light on the influence of downwash flow created by propellers on the spray characteristics, using a commercial flat-fan spray nozzle which is in actual use.

Keywords: *Downwash, Flat-fan Nozzle and UASS, Laser Diagnostics, Patternation*

*Corresponding author: surya@iitdh.ac.in

Effect of heated swirling air on the spray structure of flow blurring atomizer

Abhishek Gupta, Surendra Kumar Soni, Noorul Huda, Santanu De*

Department of Mechanical Engineering, Indian Institute of Technology Kanpur, India

Abstract

Flow blurring atomization is one of the most efficient atomization techniques that converts the liquid into very fine droplets at a low-pressure drop. Swirling primary air in flow blurring spray is expected to improve the secondary atomization and the spray dispersion. Furthermore, it would be very effective in anchoring the flame and achieving a short and intense reaction zone from the perspective of efficient combustion. The present work qualitatively investigates the effect of the external swirl strength on the spray structure through Mie-scatter imaging. In these experiments, the swirl strength is varied by increasing the swirling air flow rate at a fixed swirl vane angle and the air-to-liquid mass ratio of the flow blurring atomizer. Atomization experiments are performed on a lab-scale, heated, swirling airflow. The burner configuration consists of a centrally mounted injector body and coaxial recess for fixing the swirler. The preliminary investigation demonstrated a significant effect of the swirling air and preheat temperature of the swirling air on the axial and radial dispersion of the spray structure.

*Corresponding author: sde@iitk.ac.in

Spray Characterization of Non-Newtonian liquid jets using a Rotary Atomizer

Amitesh K. Chaudhary^{1*}, R.V. Ravikrishna²

¹Mechanical Engineering, Indian Institute of Science, Bengaluru, Email: amiteshc@iisc.ac.in

²Mechanical Engineering, Indian Institute of Science, Bengaluru, Email: ravikris@iisc.ac.in

Abstract

In this study, the breakup process of a polymer jet was investigated experimentally, using a rotary atomizer. Different polymer concentrations (0.1 – 1.0 wt.%) of xanthan gum (XG) were prepared in distilled water using a magnetic stirrer, to examine the concentration dependence of the spray characteristics with rotational speeds varying from 10,000 RPM to 30,000 RPM. Rheological measurements of the polymer solutions were obtained before the atomization, using a stress-controlled rheometer. Polymer solutions exhibited shear-thinning non-Newtonian behavior on a log-log apparent viscosity-shear rate plot. Shadowgraphy technique was used for jet visualization and spray characterization. For rotational speeds less than 15,000 RPM, very less jet breakup was observed due to dominating viscoelastic forces. The viscoelastic behavior of polymer solutions showed negligible effect on Sauter Mean Diameter (SMD) for rotational speeds greater than 25,000 RPM.

*Corresponding author: amiteshc@iisc.ac.in

Characterization of the swirl velocity field and atomization processes of a gas turbine swirl injectors

Vivek Sahu*, Aman Bakshi, K.P. Shanmugadas

Department of Mechanical Engineering – Indian Institute of Technology, Jammu, J&K,

181221, India

Abstract

The external spray structure and gas velocity field at the exit of a gas turbine swirl injector is characterized to understand the flow topology and effect of shear layers in accelerating the atomization process. The injector geometry consists of a primary swirler and a counter-rotating secondary swirler separated by a pre-filming surface. Experiments are conducted at scaled-down atmospheric ambient conditions, and the air-to-liquid ratio is varied from 12-25. Axial and cross-plane particle image velocimetry experiments are conducted to obtain the three-component gas velocity field at the injector exit. Direct laser sheet imaging experiments are adopted to capture the spray topology and spray shear layer characteristics. Spray PIV experiments are conducted to obtain the axial droplet velocity field. The shear strength of the swirler plays an important role in accelerating the ligament breakup and atomization at the injector lip.

Experimental Study on Injection and Spray Characteristics of Coal-Derived Liquid Fuel

Zhao Lyu, Zhimu Ye, Hucheng Zhang, Xinqi Qiao*, Zhiwei Jin

Key Laboratory of Power Machinery and Engineering of Ministry of Education,

School of Mechanical Engineering, Shanghai Jiao Tong University

Shanghai, China

Abstract

Coal-derived liquid fuel is one of the alternatives to traditional petroleum diesel. The injection characteristics of diesel from direct coal liquefaction (DDCL), diesel from indirect coal liquefaction (DICL) and their blends (DBCL) are studied by experiments on the high-pressure common rail test bench, and the spray characteristics of test fuels are also studied in the constant volume chamber using high-speed camera. The injection delay, injection rate, injection quantity, spray penetration distance, spray cone angle, spray area and Sauter mean diameter (SMD) are analyzed and compared with petroleum diesel. Another research is made on the injection process of the injectors of vehicle engines fueled with DBCL and petroleum diesel for a long time. The results show that the maximum volume injection rate and injection quantity of coal-derived liquid fuel are larger than petroleum diesel, but the injection delay is smaller. The spray penetration distance and SMD of coal-derived liquid fuel are smaller than petroleum diesel and spray cone is larger, due to their differences in density, viscosity and surface tension. Besides, the degree of injector deposits of vehicle engine fueled with DBCL is better than that of petroleum diesel.

*Corresponding author: qiaoxinqi@sjtu.edu.cn

Modelling and Computations - II

Assessment of mixing time-scale models for dilute spray flames using the multiple mapping conditioning approach

Eshan Sharma*, and Santanu De

Combustion and Energy Conversion Systems Laboratory,

Department of Mechanical Engineering, Indian Institute of Technology Kanpur, India

Abstract

This article presents simulations of turbulent, pilot-stabilised, dilute spray flame using the sparse-Lagrangian multiple-mapping conditioning (MMC) approach in the large-eddy simulation (LES) framework. MMCLES is a three-way coupled Eulerian-Lagrangian-Lagrangian approach for the evolution of gas-phase flow, composition, and liquid fuel droplets, respectively, where MMC acts as a molecular mixing model for the sub-grid scale (*sgs*) composition. MMC localises the molecular mixing in an independent reference space, allowing a sparse resolution of stochastic particles, which permits computationally efficient simulations of this flame series. Two acetone flames, viz. AcF1 and AcF3, with a decreasing fuel mass loading but the same carrier air flow rate, are investigated to understand the effect of fuel mass flow rate on the flame structure. The chemical kinetics of acetone is represented by the GRI mechanism augmented by a sub-mechanism and consists of 39 species and 226 reactions. Validation of MMC-LES predictions of liquid and gas-phase statistics are compared with experimental measurements. A good overall agreement is observed for both flames, with minor discrepancies for AcF3 in the outer flame region.

*Corresponding author, email : eshans@iitk.ac.in

Surface Breakup of Liquid Jet – Insights from a DNS study

S. Balaji, Ankur Kumar, Aishwary Pratap, Anubhav Sinha*

Department of Mechanical Engineering

Indian Institute of Technology (Banaras Hindu University)

Varanasi, Uttar Pradesh, 221005, India

Abstract

This study explores the surface breakup of a liquid jet under high-pressure conditions using DNS simulations. The effect of non-dimensional numbers, viz. Reynolds number (Re), aerodynamic Weber number (We) and liquid-to-gas-density ratio are explored. Initial surface instability, its growth, and contribution to breakup is studied. High fidelity simulation images are examined to understand the surface breakup process and development of jet tip. Results show the 2D structure of the initial wave growth. This structure quickly gets transformed to 3D disturbances with ligaments and droplet formation. The surface wave growth is observed to amplify with increase in We , relative velocity and decrease in density ratio. Reynolds number does not appear to be the controlling parameter.

*Corresponding author: er.anubhav@gmail.com

Numerical investigation of the bifurcating behavior of liquid jets in crossflow

Mingyun Xie*, Shengqi Wu*, Bin Yu, Miaosheng He, Bin Zhang, Xiaobin Huang, Hong Liu

Shanghai Jiao Tong University

Shanghai, China

Abstract

Liquid jets in crossflow are widely applied in a variety of combustors for the excellent effect on fuel and oxygen mixing. In this paper, numerical simulation of liquid jets in crossflow was conducted using OpenFOAM by applying the volume of fluid (VOF) method to capture the liquid interface, and the Lagrangian method with the ReitzDiwaker breakup model to simulate the atomization process. Adaptive mesh refinement was applied for cost-reducing. The trajectory and breakup position of the jet were validated against experimental results. A shear breakup vortex (SBV) region and a counter-rotating vortex pair (CVP) region were identified in the leeward side of the liquid jet due to interaction between the liquid jet and crossflow. These two regions were separated by a transition point (TP). The integrated liquid volume distribution of the spray region was demonstrated to show the intuitive bifurcating characteristic. Bifurcations of the jet were identified through a separated line, which was described based on a mathematical formula. A bifurcating location was defined as the intersection point of the separated line and the trajectory, which is slightly above the TP, and the jet was departed into two distinct streamers from this point. Relationship of the bifurcating behavior and the flow structures were analyzed and investigated, showing the important role of vortex morphology in spray atomization.

Key Words: Liquid jet in crossflow, OpenFOAM, shear breakup vortex, counter-rotating vortex, bifurcation.

*Corresponding author: 2016sege@sjtu.edu.cn (M. Xie) , shengqiwu@sjtu.edu.cn (S. Wu)

Numerical study of the influence of the ambient density on the GDI nozzle internal flow and spray characteristics

Chaoqun HU, Pengfei LENG, Zhijun WU*, Wenbo ZHAO, Chunyu YANG, Liguang LI

School of Automotive Studies, Tongji University, China

Abstract

The nozzle internal flow characteristics under different ambient gas densities were compared and results indicated that the internal flow condition at 1kg/m^3 ambient gas density was significantly different from those at other conditions. The liquid adhered to the wall at the outlet of the nozzle, and the liquid core was broken before it shot out of the sink hole. To further explore the influence of nozzle internal flow condition on the spray characteristics, a high-fidelity spray model based on the one-way coupling method was established and the flow field information at the nozzle exit from the Eulerian in-nozzle flow simulation were used as the initial conditions of the Lagrangian spray modelling. The results highlight the size constant of KH model and RT model have greater influence on Sauter mean diameter, while turbulence model constant and size constant of RT have greater influence on penetration distance. The simulation results of penetration distance are in good agreement with the experimental results. The deviation between the numerical and experimental results of Sauter mean diameter is 2.6% at 1kg/m^3 ambient gas density.

*Corresponding author: zjwu@tongji.edu.cn

Prediction of thermophysical properties of n-dodecane over a wide temperature and pressure range based on molecular dynamics

Yuanyuan Shen^a, Chuqiao Wang^b, Zhixia He^{a,*}, Yanzhi Zhang^a

^a Institute for energy research

^b School of Energy and Power Engineering

University of Jiangsu

Zhenjiang, Jiangsu Province, 212000, China

Abstract

In order to meet the new requirements of the “dual carbon” target for the range of thermophysical properties of hydrocarbon fuels, a new computational correction method is proposed based on the traditional molecular dynamics method to predict the thermophysical properties of n-dodecane over a wide temperature and pressure range. By integrating stability correction, kernel density estimation (KDE) correction, and zero potential energy surface correction, the calculation accuracy of the thermophysical properties is enhanced, and the average absolute relative deviation (AARD) together with standard error (SE) are used to perform error analysis on the existing and nonexistent properties data in the National Institute of Standards and Technology (NIST) database, respectively. The results indicate that the calculation correction method proposed in this paper can effectively improve the calculation accuracy of the thermal properties of n-dodecane, and the obtained results are in good agreement with the experimental values. Among the environmental factors, pressure has a greater impact on density and thermal conductivity, while internal energy, enthalpy, and entropy are more sensitive to temperature. Viscosity only changes significantly when the temperature falls below the critical point.

*Corresponding author: zxhe@ujs.edu.cn

Applications - II

An innovative solution to prevent airborne transmission of infectious respiratory droplets inside a classroom

Riddhideep Biswas¹, Anish Pal¹, Sourav Sarkar^{1*}, Achintya Mukhopadhyay¹

¹Department of Mechanical Engineering, Jadavpur University Kolkata-700032, West Bengal, India.

Abstract

The education sector has suffered a catastrophic setback due to ongoing COVID-pandemic, with classrooms being closed indefinitely. The current study aims to alleviate the existing daunting condition by extensively examining COVID -transmission inside a classroom and giving long-term sustainable solutions. In this work, a standard 5m X 3m X 5m classroom is considered where 24 students are seated, accompanied by a teacher. A CFD approach using an Eulerian-Lagrangian framework, utilizing OpenFOAM software has been used to simulate the above situation. The infection risk in the classroom for the students considering the teacher is infected have been evaluated based on the stochastic dose response framework. A quiescent hot dry ambience reports to be most risky scenario, the infection probability reaching 100% for certain students. Furthermore, investigation reveals that wearing a surgical mask does not provide complete protection against the infection, the infection probability reaches 100% for certain students even in the case of surgical masks. A novel solution in the form of screens suspended from the ceiling has been proposed that entraps the droplets and this reduces the infection risk by 85% from the no screen scenario.

*Corresponding author: souravsarkar.mech@jadavpuruniversity.in

Numerical Investigation of Water-assisted Atomization of Molten Metal Jets for Additive Manufacturing Applications

A. Chakravarty¹, S. Sarkar^{2*}, A. Mukhopadhyay² and S. Sen²

¹School of Nuclear Studies & Application

²Department of Mechanical Engineering

Jadavpur University, Kolkata, West Bengal, INDIA

Abstract

Metal powder generation by water atomization has become an emerging method for powder production in additive manufacturing due to its cost effectiveness. Even with certain limitations, water-assisted atomization can achieve desired powder characteristics when certain conditions are met. The fundamental understanding of the mechanism of water-assisted atomization is, however, still lacking. In this regard, the present study attempts to fundamentally analyze the basic mechanism of water-assisted atomization of molten metal jets through computations. A computational fluid dynamics simulation is performed using the Volume of Fluid approach to model primary atomization of molten metal jet in presence of atomizing water jets. The effect of change in water jet apex angle is investigated by considering two different apex angles. An in-house image processing algorithm is used to determine the droplet and ligament size generated from primary atomization. It is found that the Sauter Mean Diameter of the droplets decreases with increasing apex angle. However, the droplet sphericity decreases with increasing apex angle.

*Corresponding author: souravsarkar.mech@jadavpuruniversity.in

Rewetting On Horizontal Tube Surface During Air-Atomized Spray Cooling: An Experimental Investigation

Bhuwanesh Kumar*, Ravi Kumar and Akhilesh Gupta

Department of Mechanical & Industrial Engineering, Indian Institute of Technology Roorkee - 247667,
India

Abstract

Rewetting the Calandria tube's surface is essential for keeping a safe temperature in a low-frequency event such as a loss-of-coolant accident in a pressurized heavy water nuclear reactor. The rewetting study was conducted on SS-304 tubes at a set initial temperature with constant heat flux. The rewetting temperature, rewetting velocity, and maximum heat flow phenomena that occur when horizontal tubes are cooled with an air atomized spray were examined in this work. The rewetting on horizontal tube surfaces using air-atomized water spray is still a poorly understood phenomenon. Rewetting provides an immediate indication of the cooling rate achieved during the cooling process. To determine the rewetting velocity in the peripheral direction of the circular tube surface, a high-speed video camera and thermocouples were used. At $P_w=0.5$ bar and $P_a=3$ bar, the greatest rewetting velocity was measured 9.95 mm/s.

*Corresponding author: 347bhuwanesh@gmail.com

The study of coal particle combustion in oblique-impinging methane/air premixed flames

Yueh-Heng Li*, Vadlakonda Sirisha, Chia-Wei Chang

Department of Aeronautics and Astronautics, National Cheng Kung University, Tainan, 701, Taiwan.

*Prof. Yueh-Heng Li

Department of Aeronautics and Astronautics, National Cheng Kung University, Tainan, 701, Taiwan.

Phone: 886-6-2757575 ext 63632

Fax: 886-6-238-9940 (Taiwan)

E-mail: yueheng@mail.ncku.edu.tw

Abstract

A novel coal-firing burner with two oblique-impinging flames was used to improve combustion efficiency and reduce particulate matter emissions. Two oblique jet burners and one slot jet burner make up an oblique-impinging jet burner. The recirculation zone was created by an oblique jet burner with a preset deflection angle and a stoichiometric methane/air flame. The coal particles were delivered through a central slot jet burner. Two impinging flames would form two flow structures from two premixed impinging flames, an impinging jet zone and an inner recirculation zone. As a result, coal particles would come into touch with the devolatilization process, resulting in the production of combustible gases such as hydrogen, methane, and carbon monoxide. The microscopic coal particles would be carried downstream to the impinging jet flame and burn away. To examine the combustion characteristics, some parameters are assessed in terms of the deflection angle (30°, 45°, 60°) of the oblique-impinging jet nozzle and the equivalence ratio of two premixed flames. The flow structure in the proposed burner was analyzed using particle imaging velocimetry, and the surface temperature of coal particles was measured using two-color pyrometry. SEM was employed to observe the shrunken coal particles at various heights.

Keywords: coal-firing burner; oblique-impinging flames burner; recirculation zone; PIV; two-color pyrometry; kriging model

Investigation of Particle number concentration emission characteristics according to damage type of DPF for diesel vehicle

S. Park, G. Park, T. Hwang, H. Ryu, S. Lee*

Graduate School of Automotive Engineering Kookmin University, Department of Automotive Engineering

University of Kookmin

Seoul, 02707, Korea

Abstract

As a measure to improve air pollution in the metropolitan area, the Ministry of Environment of the Republic of Korea carried out a project to attach an exhaust gas reduction device to old vehicles, and by attaching DPF to old vehicles, the reduction efficiency of PM and NO_x was improved by more than 80%. However, there are reports that the number of vehicles equipped with damaged DPFs is increasing due to insufficient performance maintenance and inspection management of DPFs. Therefore, it is time to introduce technology to check whether the vehicle's DPF performance and original shape are maintained.

Conventional DPF diagnostic methods, such as soot concentration test, OBD (On-Board Diagnostics) diagnosis, and visual inspection after DPF separation, are difficult to clearly confirm the state of DPF and take a lot of time.

Therefore, the previous study "Study on whether the DPF was damaged using CR X-ray imaging technique" was conducted, and using CR-X-ray, it was possible to determine whether the internal substrate of the DPF was damaged in cracks, melts, and hollows, and whether soot and ash were accumulated. The technical feasibility of acquiring and verifying images through X-rays was confirmed. However, there are not many studies on the characteristics of exhaust gas depending on whether the DPF attached to the vehicle is damaged.

In this study, a non-destructive technique was used to analyze the correlation between the DPF damage diagnosis for a moving vehicle and the exhaust gas. The DPF classified by type of damage was attached to the test vehicle, and through the KD-147 test, the particle number emission characteristics and soot concentration characteristics of particulate matter were identified for each type of DPF damage. The test was conducted 10 times each according to the type of DPF damage. A particle counting counter (NPET) was used to determine the number of particles of particulate matter according to the type of damage DPF, and a smoke density meter (OPA-102) was used to characterize the soot concentration according to the type of damage DPF.

In the case of the measured soot concentration, both the normal DPF and the damaged DPF satisfied the domestic soot concentration regulation of less than 10%. And in the case of the number of particles of particulate matter, it was confirmed that the damaged DPF exceeded the limit of 1,000,000 particles per cubic centimeter at PTI in the Netherlands.

Keywords: diesel particulate filter (DPF); particulate matter (PM); soot; ash; computed radiography (CR); X-ray; Image J; Nanoparticle Emission tester (NPET); KD-147 test mode (Korean Diesel Test mode); Smoke opacity (OPA-102)

Experimental Atomization - III

Liquid Sheet Break-up Characteristics of Gel Propellant Simulant

K Vivek* and Kabiraj L

Department of Mechanical Engineering

Indian Institute of Technology Ropar, Rupnagar, Punjab, 140001, INDIA

Saurabh A

Department of Mechanical Engineering

Indian Institute of Technology Kanpur, Kanpur, UP, 208016, INDIA

Abstract

Liquid atomization experiments on a like-on-like impinging jets configuration were conducted for water and water-Carbopol gels. A comparison of a liquid sheet break-up process of gel and water under similar operating conditions is presented here. Variation in sheet break-up dimensions for the shear-thinning gels and water for a range of Weber number 100–2000 is reported. Waves visible in the liquid sheet were observed to be the major cause of sheet disintegration. For the highest gel concentration (0.15 wt.%) investigated, sheet break-up length was found to reduce substantially with an increase in Weber number compared to 0.10 wt.% and water, both of which showed only a marginal change in sheet break-up length.

*Corresponding author: 2018mez0025@iitrpr.ac.in

Experimental study on the spray structure and nozzle tip wetting with LPDI injector under flash boiling

Young Soo Yu^a, Sungwook Park^{b,*}

^aDepartment of Mechanical Convergence Engineering, Graduate School of Hanyang University, Seoul 04763, Republic of Korea

^bSchool of Mechanical Engineering, Hanyang University, Seoul 04763, Republic of Korea

Abstract

Due to stricter exhaust emission regulations, research on the alternative fuels vehicles and eco-friendly vehicles is in process. LPG fuel has the advantage of having a lower C/H ratio than gasoline, which reduces carbon dioxide emissions and is cheaper. However, the boiling point of LPG is lower than room temperature, resulting in flash boiling phenomenon. Flash boiling can improve the engine combustion efficiency by promoting the atomization of spray, but there is a disadvantage that the spray hits the wall inside the cylinder under superheated conditions, increasing the particulate matter. In addition, the particulate matter is generated on the surface of injector nozzle directly exposed inside the combustion chamber. In this study, the far-field and near-field visualizations and nozzle tip wetting visualization were conducted using n-heptane and LPG summer under flash boiling conditions. As a results, in near field, the spray width increases due to spray plume expansion as the fuel temperature increases. In addition, in far-field, spray collapse was generated due to increasing fuel temperature and thus the spray was developed to center. Finally, in the nozzle tip wetting, it was found that the nozzle tip wetting decreased due to evaporation of test fuels under flash boiling conditions.

Keywords: Far-field, near-field, nozzle tip wetting, LPDI injector, flash boiling

*Corresponding author: parks@hanyang.ac.kr

Experimental investigations of nozzle pitch circle impact on tip wetting

Kaixiang Li, Di Xiao, Shangning Wang, Shuyi Qiu, Xuesong Li, Min Xu*

School of Mechanical Engineering, Shanghai Jiao Tong University, 800 Dongchuan Road, Shanghai,
China 200240

Abstract

Fuel films deposited on the injector tip, or so-called nozzle tip wetting, have been acknowledged as an essential source of engine emissions. Recent research found that nozzle geometry parameters would affect the generation of tip films. However, relevant investigations are still limited to emission measurements and corresponding analysis. Due to the lack of visualization investigations, mechanisms of nozzle tip wetting generation and its link with geometry parameters are still unclear. To elucidate the effect of nozzle pitch circle on tip wetting and its underlying mechanism, the dynamic formation process and fuel film area evolution of nozzle tip wetting were directly visualized using Laser-induced Fluorescence technique and photomicrography technique. Experiments were carried out in a constant volume chamber, using injectors of different pitch circles, under sub-cooled and flash boiling conditions. It was found that reducing the pitch circle resulted in a small decrease in tip wetting under sub-cooled conditions, however, under flash boiling conditions, reducing the pitch circle resulted in a sharp increase in tip wetting. The phenomena aforementioned could be well understood according to the motion of spray plumes and the fuel films, which would contribute to the control strategies of nozzle tip wetting and the optimal design of fuel injection systems.

Keywords: Tip wetting; Laser-induced Fluorescence; Photomicrography; Flash boiling

*Corresponding author: mxu@sjtu.edu.cn

Experimental Study on Wall Impingement Characteristics of Hydrogen / Diesel Dual Fuel Spray

Yan Lei*, Wenbo Jin*, Tao Qiu, Chenxi Liu

Faculty of environment and life, Beijing University of Technology, Beijing 100124, People's Republic of China

Abstract

Hydrogen/diesel dual fuel injection reduces carbon emissions of conventional diesel by adding hydrogen, a kind of non-carbon fuel. During the fuel injection, the spray impingement occurs, which directly affects the air-fuel mixture formation and finally the engine power and emissions. The wall impact characteristics of hydrogen/diesel dual fuel high-pressure jet were investigated on the dual-fuel injection test bench by using high-speed photography. The diesel-hydrogen dual-fuel spray impingement area and its impingement time were measured under the condition of various injection pressure. The results show that both the dual-fuel spray impingement area and its impingement time increase with the increase in the hydrogen injection pressure. When the hydrogen injection pressure reaches 16 MPa, the dual-fuel spray impingement area is 129% higher than that of a single diesel jet, and the dual-fuel spray impingement time is 46% higher than that of a single diesel jet. These results reveal that the increasing hydrogen injection pressure is good for improving the diesel spray and liquid-gas mixture.

*Corresponding author: leiyan@bjut.edu.cn

Comparative Study of Microscopic Spray Characteristics of Dimethyl Ether (DME) with Diesel in Atmospheric Ambient Conditions

Vikram Kumar, Shanti Mehra, Hardikk Valera, Avinash Kumar Agarwal*

Engine Research Laboratory, Department of Mechanical Engineering,

Indian Institute of Technology Kanpur, Kanpur, 208016, India

Abstract

Increasing global energy demands and limited fossil fuel reserves have led to sky-rocketing crude oil prices. Hence, there is a need to research and develop green and clean e-fueled engines that meet stringent emission norms and are also cost-effective. Dimethyl ether (DME) is a biofuel that emits very low emissions compared to baseline mineral diesel. Its physical, thermal and chemical properties are ideal for engine applications. Its fuel spray characterization is necessary to understand the spray morphology and atomization. The spray characteristics of fuel determine the fuel-air mixing, an essential requirement for efficient combustion. In this experimental study, a comparative analysis of the microscopic spray characterization of DME and diesel fuels was done. Spray analysis was done inside a constant volume spray chamber (CVSC) to assess the distributions of spray droplet velocity and droplet size using a phase doppler interferometer (PDI). Results showed that DME spray droplet sizes were smaller than diesel due to differences in properties such as surface tension and viscosity. Smaller droplets of DME spray showed higher maximum droplet velocity than diesel. Also, the number of smaller droplets in DME spray was higher than in diesel. This study showed that DME has superior spray evaporation/ atomization characteristics, enhancing the fuel-air mixture's homogeneity. It is also helpful in understanding DME-fueled engines' combustion, performance, and emissions characteristics.

*Corresponding author: akag@iitk.ac.in

Large-scale flow features in swirl airblast atomization

Surendra Kumar Soni^{1*}, Pankaj S. Kolhe²

¹Dept. of Mechanical Engineering, Indian Institute of Technology Kanpur, India

²Dept. of Mechanical and Aerospace Engineering

Indian Institute of Technology Hyderabad, India

Abstract

The direct substitution of the atomizing liquid with the alternative fuel influences the combustion performance, among which the atomization characteristics are the main ones. In the present experimental work, we have contrasted the liquid jet breakup process when the water spray is replaced with vegetable oil. The nearfield region of spray is illuminated by diffuse LED light and laser sheet in the shadowgraph and Mie-scatter imaging mode, respectively. The captured images are used to investigate the liquid jet breakup process and are further utilized for Proper Orthogonal Decomposition analysis to extract the coherent structure. The swirl airblast atomizer is employed to fragment the central liquid jet with the fast-moving annular swirl atomizing air. A detailed comparison is made for the various conditions. The results show that the fundamental instabilities that develop in the breakup process are flapping, wave, and explosive instabilities, which primarily depend upon the atomizer's working principle and operating conditions rather than the liquid fuel properties.

*Corresponding author: surends@iitk.ac.in

Modelling and Computations - III

A computational analysis of hydrodynamic instability during co-current airflow over a thin liquid film

S. Saha¹, S. Sarkar², A. Chakravarty^{1*}, A. Mukhopadhyay²

¹School of Nuclear Studies & Application

²Department of Mechanical Engineering Jadavpur University
Kolkata, West Bengal, India

Abstract

Hydrodynamic instabilities generated due to interaction of airflow over thin liquid film is a phenomenon of considerable importance that is encountered during natural and industrial processes. One major example is during the time of breathing when air flows over an underlying thin mucus layer in human respiratory tract. Such interaction can destabilize the mucus layer due to hydrodynamic instabilities and may result in formation of respiratory droplets within the tract. These respiratory droplets play an important role in disease transmission within the lung as well as in drug delivery to various regions of the lung. A fundamental investigation of this phenomenon, thus, becomes extremely important. A computational analysis is carried out using the Volume of Fluid (VOF) approach in OpenFOAM considering a two-dimensional representative channel which approximates a region of the respiratory tract. Air and mucus are selected as the two interacting phases. The analysis is carried out for different airflow velocity and mucus film thickness considering co-current flow of air and mucus, and focuses on understanding the underlying physics behind destabilization of the mucus layer under different physiological conditions.

*Corresponding author: aranyak.chakravarty@jadavpuruniversity.in

Numerical study on atomization and combustion characteristics of multi-jet gas-liquid pintle injector

Xinyao Lei, Xiaodong Li, Shuai Zhou, Hongjun Liu*

Affiliation

Northwestern Polytechnical University Xi'an, Shaanxi, China

Abstract

The pintle injectors are widely used in liquid rocket engines. In order to further improve the atomization effect of the pintle injector and solve the ablation problem of the head of the pintle injector, multi-jet gas-liquid injection form is proposed, which adds oblique jets under the radial jets and centrifugal jets at the bottom of the plug head. Based on the VOF and DPM models, the two-phase flow process and SMD distribution with different oblique injection angles were analyzed, and the combustion chamber temperature distribution and combustion efficiency were compared with the non-premixed combustion simulation. The results show that as the angle between the oblique jet and the axial direction increases, the SMD distribution decreases more rapidly along the radial direction, which significantly improves the atomization performance. The oblique jet promotes the development of combustion along the axial direction and improves the combustion efficiency. The centrifugal jet blocks the high temperature gas under the plug head and forms a low temperature area at the bottom of the plug head, which plays a thermal protection effect.

Keywords: Two-phase flow, sauter mean diameter, combustion, pintle injection, liquid rocket engine

*Hongjun Liu: liuhj@nwpu.edu.cn

Investigation of Spray in Crossflow using Automatic Coupled Volume of Fluid and Lagrangian Particle Tracking within Large Eddy Simulation Framework

Yaquan Sun^{1,2,*}, Kaushal Nishad^{1,2}, Yongxiang Li^{1,2}, Louis Dreßler^{1,2} and Amsini Sadiki^{1,2}

Technische Universität Darmstadt, Department of Mechanical Engineering

¹Institute of Reactive Flows and Diagnostics, Otto-Berndt-Straße 3, 64287 Darmstadt, Germany

²Institute of Energy and Power Plant Technology, Otto-Berndt-Straße 3, 64287 Darmstadt, Germany

Abstract

The spray evolution process can be divided into a dense and a dilute region. With regard to a complete numerical modeling of such a system, the high-resolution Volume-of-Fluid (VOF) simulation can consider suitable in resolving the wide range of length and time scales involving in sprays, especially in the near-nozzle flow and spray atomization characteristics. However, the associated high computation cost considerably limits among other its applications. For a complete spray simulation, a good trade-off between accuracy and required computational cost can be achieved when the primary atomization in the dense region is solved by the VOF method, while the Lagrangian particle tracking (LPT) method is used for the dilute spray region. In this paper an automatic coupling of both methods is applied to numerically describe the complete spray atomization of a liquid jet in a cross flow. It is found that the calculated spray characteristics, including spray penetration jet trajectory and droplet size distribution agree well with experiment. The obtained numerical results further reveal that the computational costs are significantly reduced since the VOF droplets in the Eulerian framework are converted into the Lagrangian framework in a safe way, and thus the number of cells are reduced due to the adaptive mesh refinement. The analysis of the number-based probability density functions (pdf) of Lagrangian particles at six different axial planes shows that they follow a Gaussian form with a maximum at around 20 μm . All this evidence the applicability of the seamless automatic VOFLPT coupling approach in improving the description of the complete spray evolution including the liquid atomization using economical computational costs.

*Corresponding author: sun@rsm.tu-darmstadt.de

Numerical Modelling of Fuel Spray Autoignition using Open-Source Code

C. -Y. Wu*, T. -C. Hsu, X.-X. Chen.

Department of Aeronautics and Astronautics

National Cheng Kung University

Tainan, 70101, Taiwan

Abstract

This paper was intended to present a numerical study of fuel spray ignition in a control volume at high temperature and pressure. It has been known that the operation and control of low-temperature combustion and homogeneous charge compression ignition engines depend on the autoignition properties of fuels. In order to study this topic systematically, the OpenFOAM, which is an open-source code for numerical simulation, was utilized in the present study. The spray pattern in terms of the spray angle, spray penetration depth, and the ignition and burning of fuel spray were delineated. In addition, the calculated ignition delay time was also compared with the data obtained from the experiments in the constant volume chamber. The results showed that the numerical results have fairly well agreement with the experimental data.

Keywords: autoignition of fuel spray; ignition delay; open-source code

*Corresponding author: cywu@gs.ncku.edu.tw

Experimental Verification of OpenFOAM-based Simulations of Secondary Atomization

S. Deti, K. G. Kshetri, S. Dighe, K. Bodi, H.Gadgil*

Department of Aerospace Engineering

Indian Institute of Technology Bombay

Mumbai, 400076, India

Abstract

Fragmentation of drops or ligaments into much finer droplets while moving relative to the ambient gas is known as secondary atomization, and it is indeed instrumental in dictating the final shape of the drop size distribution. The interplay between aerodynamic, viscous, and surface tension forces generates a variety of droplet deformation morphologies leading to various breakup regimes studied extensively in the past. The scope of the present work is to simulate the phenomenon of secondary atomization using the open-source CFD tool (OpenFOAM). OpenFOAM has gained popularity for simulating a variety of fluid flows in recent years. We assess the capability and the reliability of OpenFOAM solvers in simulating multiphase, particle-laden flows using the relatively simple problem of secondary atomization. The simulations are being performed over a range of Weber numbers to encompass all the regimes of droplet breakup. Experiments are also simultaneously performed by injecting water droplets in the air jet facility for similar non-dimensional numbers and capturing the high-speed visualization. The comparative assessment of the simulation results and the experimental observations being conducted using estimation of breakup regimes and droplet deformation morphologies show promising trends.

*Corresponding author: gadgil@aero.iitb.ac.in

Numerical Investigation of Atomization Characteristics of AdBlue Injection in SCR-DeNOx system using VOF-LES

Kaushal Nishad^{1,*}, Yaquan Sun¹, Yongxiang Li¹ Christian Hasse², and Amsini Sadiki¹

Technische Universität Darmstadt,

¹Reactive Flows and Diagnostics, Otto-Berndt-Str. 3, 64287 Darmstadt, Germany

²Simulation of reactive Thermo-Fluid Systems, Otto-Berndt-Str. 2, 64287 Darmstadt, Germany

Abstract

In a diesel engine, the (selective catalytic reduction) SCR-DeNOx system utilizes NH₃ obtained during the thermal decomposition of AdBlue as a reducing agent for NO_x reduction. Since AdBlue is injected at moderate pressure (5-10 bar), the pressure-swirl AdBlue injector can provide AdBlue spray with adequate atomization characteristics. Based on the engine loading conditions and measurements from exhaust gas sensors upstream to SCR, various AdBlue dosing strategies are followed, namely, the injection mass, duration and injection frequency. Moreover, during the nozzle opening and closing phases of each injection event, achieving a good atomization of AdBlue jet is still challenging. In this work, a detail numerical investigation of the pressure-swirl AdBlue injector is carried out using the volume of fluid (VOF) method within a large eddy simulation (LES) framework to include the transient evolution of spray at early injection stage. Thereby, an adaptive mesh strategy is applied in order to adequately resolve the near-nozzle flow characteristics. In particular, the primary atomization due to formation and breakup of hollow-cone liquid-jet into ligaments and subsequent droplets. It turns out that, the liquid mass retained in nozzle orifice from previous injection and gradual development of flow-swirl leads to formation of bigger droplet and connected ligament at early injection stage. This can contribute to the accumulation of liquid film and formation of solid deposit in the SCR duct. The compared droplet size distribution also highlights the importance of fine mesh to resolve the smaller droplets.

*Corresponding author: nishad@rsm.tu-darmstadt.de

Optical Diagnostic Methods

Diffused Back Illumination (DBI) Experimental Technique on Non-Flash Boiling and Flash Boiling Sprays Using a Single-hole Gasoline Direct Injection (GDI) System

Anurag Gaur, Kaushik Saha*

Engines and Unconventional Fuels Laboratory
Department of Energy Science and Engineering
Indian Institute of Technology Delhi
New Delhi (110016) India

Abstract

The present work focuses on spray visualization with the following fuels – gasoline, a blend of methanol and gasoline (M85), using a single-hole gasoline direct injection system in a constant volume chamber. GDI engines often operate at a high (100 to 120 bar) injection pressure to achieve better atomization. In addition, fuel injection takes place against low and high chamber pressure under early and late injection strategies, respectively, which affects spray morphology. Due to the direct injection of fuel into the combustion chamber, the fuel residing in the fuel injector experiences high heat flux due to combustion in the chamber. The fuel temperature rises to 90°C, consequently its saturation pressure also increases. When the heated fuel is injected in the early mode at part load condition, it attains superheating, leading to flash boiling. Macroscopic morphology of spray is being investigated under non-flashing and flash boiling conditions. An LED-based diffused back illumination (DBI) technique is used in the current work to analyze the liquid phase of the spray, such as liquid penetration, spray cone angle, and spray width.

Keywords: GDI, gasoline, methanol, M85 blend, flash boiling, DBI

*Corresponding author: kaushiksaha@dese.iitd.ac.in

Liquid Jet Breakup Characterization in a Crossflow Airblast Atomizer by Time -Resolved Fluorescence Imaging Technique

Siddhivinayak Rampurkar*, Shirin Patil, Srikrishna Sahu

Department of Mechanical Engineering

Indian Institute of Technology, Madras

Chennai 600036, India

Abstract

The atomization of liquid jet by a crossflow air within wind tunnel has received wide attention in literature. The focus of the present research work is to study breakup characteristics of a single liquid jet within an injector which is relevant to Lean Premixed Prevaporized (LPP) gas turbine combustors. The aim of the research is to experimentally characterize the instabilities in liquid jet breakup process using high-speed fluorescence imaging by optical connectivity (OC) technique. The technique facilitates exclusive visualization of the jet core by capturing the fluorescing signal from the jet such that the breakup parameters can be defined accurately. The atomizer is operated over a wide range of Weber number (We_g) and momentum flux ratio (MFR) in the absence and presence of swirling crossflow of air. The frequency of surface instability of liquid-air interface is measured by time tracking the cross-stream location of windward interface and thereafter applying FFT analysis over the obtained signal. The surface instability frequency is found to be governed by We_g rather than MFR . The breakup parameters such as jet breakup and penetration length, jet width and spread angle are reported for different MFR and We_g which are vital for the design of fuel injectors. The mean jet penetration and breakup length and spread angle are dependent on both We_g and MFR rather than MFR alone. On the other hand, the jet spread angle is mostly dictated by We_g .

*Corresponding author: siddhivinayakr29@gmail.com

Digital In-line Holography and backlight illumination imaging of sprays using levitated droplets

Arpit Joglekar^{1*}, Devendra Deshmukh¹, Yogeshwar Nath Mishra^{1,2}

¹*Department of Mechanical Engineering, Indian Institute of Technology-Indore, 453552, India*

²*Department of Physics, University of Gothenburg, SE 41296, Gothenburg, Sweden*

Abstract

Digital In-line Holography and backlight illumination imaging are two of the most common techniques in two-phase flow diagnostics. While backlight illumination imaging has been mostly used for two-dimensional (2D) mapping, digital in-line holography provides three-dimensional (3D) information from a scene. Using both techniques, in this work, we investigate atomizing sprays in terms of droplet size, droplet velocity and their location in 3D field. A digital in-line holography setup is developed by taking four droplets as calibration. An acoustic trap is used to levitate four droplets in air. The levitator consists of an emitter and a reflector, and the droplets can be levitated in the nodes and antinodes generated by two standing waves. For the holography setup calibration, the levitated droplets are placed in between the collimated laser beam and the recording sensor. The interference caused by the droplets has been recorded on a CMOS sensor as a digital hologram. Thirteen holograms are generated by linearly translating the levitator stage with a step size of 2mm on either side of the focal plane. Out of these three holograms were selected to see the transition in the edge sharpness by moving away from the focal plane. The behavior of the interference pattern is recorded along the depth direction. Also, 3D droplet visualization at different depths is performed.

*Corresponding author: arpitjoglekar@iiti.ac.in

Development of an Algorithm for Structure Identification and Droplet Distribution Reconstruction in Liquid Sprays

Zuhaib Nissar*, Steven Begg and Oyuna Rybdylova

Advanced Engineering Centre, School of Architecture, Technology and Engineering University of Brighton

Brighton, East Sussex, BN2 4GJ, UK

Abstract

The present study aims to deliver insights into droplet generation in sprays and develop a suitable methodology to evaluate the droplet distribution. The goal is to aid the development of a novel simulation framework that is based on the Fully Lagrangian Approach (FLA), which is capable of modelling polydisperse evaporating droplets (gFLA) as well. Unlike more commonly used methodologies which study the overall droplet size distribution in sprays, the development and validation of the new modelling approach essentially requires characterising droplet size and velocity distributions at all locations. A commercial flat fan nozzle was adopted in our study to inject water under steady flow conditions using a low-pressure pump. The atomisation takes place such that the liquid sheet and droplets that break out from the spray do not deviate significantly from a 2-D plane. The spray is visualised at 5000 frames per second via a high-speed camera with a long-distance microscopic lens, the optical setup includes a pulsating LED for backlight illumination. The experimental data was collected from 2000 consecutive images at steady state conditions and a 2D spray analysis was performed using an in-house image processing code. The code includes a functionality to distinguish between structures in the spray and to locate the region where the two-phase flow transits from a continuous liquid film to ligaments and then to individual droplets. In the down-stream atomised region, the image processing was applied to obtain the position and size of the droplets in consecutive frames and across all sequences of images. The data was collected across the spray domain to acquire the local droplet size distributions. Finally, we mapped the spray characterisation parameters such as Sauter mean diameter, level of polydispersity, and number density.

Keywords: Optical diagnostics, atomisation, image processing, fully Lagrangian approach, droplet distribution

*Corresponding author: z.nissar@brighton.ac.uk

Evolution of Flow Patterns on Laminar Jet Impingements Using Image Processing

Surya B, Bupesh Kumar R, Arun M S, S N Sridhara, P Nandagopalan*

Department of Aeronautical Engineering,

Hindustan Institute of Technology and Science, Padur Chennai-603103, Tamil Nadu, India.

Abstract

In this study, we experimentally investigate the flow patterns originated upon the impingement of two laminar jets of water. The water jets eject from the nozzle of diameter $d_{exit} = 0.51mm$. An experimental setup was developed to avoid the bubble formation. The variation of jet velocity ratios (J_i) influences the dynamics of the cold flow patterns. The fluid used is glycerol-water mixture which is combined at 6:4 ratio. We use backlit light combined with high-resolution digital photography to study the evolution of the instabilities, post to which the image is analyzed with the help of image processing. The validation of the experimental observations was carried out on the basis of the jet patterns. The instabilities occur here, are observed in the form of onset of the daughter droplets from the rim, sheet break up and the penetration length alterations. These instabilities were found to vary with the impingement angle, and the velocities of liquid jet column. For the different values of J_i , we capture the patterns such as Transition phase (TP), Fishbone Pattern (FB) and Fierce Flapping (FF) occur at different time scales. The study is extended for the various orifices such as Square (SQ) and elliptical (EP). It is observed that the three distinct patterns are the global phenomena irrespective of the orifice shapes. However, the sudden axis switching modes enable the onset of FB instability to occur earlier than the Circular orifice in the SQ and EP orifices.

Analyzing Near-nozzle Turbulence Intensity of Fuel Sprays Using X-ray Phase-Contrast Imaging with MHz Frequency

Omer Faruk Atac, Hyunsu Lee, Seoksu Moon*

Department of Mechanical Engineering, Inha University

Michuhol-gu, Incheon, 22212, South Korea

Abstract

It has been well-recognized that nozzle- and cavitation-generated turbulence can initiate the near-field breakup of discharged liquid jets. In literature, spray simulations have been performed based on the turbulence-induced flow breakup models to predict spray formation downstream. However, due to the lack of turbulence measurement data for fuel injectors, the validity of the spray simulations has been difficult to be examined and discussed. In this study, ultrafast X-ray phase-contrast imaging (XPCI), which can provide a clear image in the near-field using a high-energy X-ray source, was applied to observe the fluctuation of flow velocity in the near-field to obtain the turbulence intensity of the discharged jet. To capture the ultrafast variance of flow velocity originating from the nature of turbulence, the high imaging frequency was applied up to 1.2 MHz. With the implemented methodology, turbulence intensity distributions of discharged liquid jets were measured for various injection pressures and nozzle geometries. The obtained results were compared and discussed with well-established knowledge from previous spray observations. The experimental method introduced in this study can provide important insights into turbulence characteristics of spray and provide valuable validation data and boundary conditions for reliable spray simulations.

Keywords: Near-Field Fuel Spray, Turbulence Intensity, X-ray Phase-Contrast Imaging (XPCI), MHz Imaging Frequency.

*Corresponding author: ss.moon@inha.ac.kr

Cavitation and Flash Boiling - I

Single String Cavitation and Swirling Flow in a Nozzle and a Hollow-Cone Spray

Dongping Shen¹, Takashi Miwa¹, Akira Sou^{1*},

Yoshitaka Wada², Yoshiharu Ueki², Hideaki Yokohata²

¹Graduate School of Maritime Sciences, Kobe University, Kobe, Japan

²Powertrain Engineering Analysis Group No.1, Mazda Motor Corporation, Hiroshima, Japan

Abstract

Single string cavitation and a swirling flow around it may occur in a fuel injector of Diesel engines at the moment of low needle lift if the tip of the needle valve is close to the orifice entrance. The swirling flow in an orifice of the injector is known to induce a hollow-cone spray with a very large spray cone angle and to enhance spray atomization, which reduces the emission of particulate matters. However, the complicated gas-liquid two-phase swirling flow phenomenon and its effects on the hollow-cone spray have not been clarified yet. In this study, the mass and momentum conservation equations of the annular swirling liquid film flow around a single string cavitation in an orifice and the hollow-cone liquid film flow in a stagnant air are derived with the correlations on wall friction and gas-liquid interfacial friction. In order to obtain the boundary condition at the entrance and exit of the orifice and to verify the validity of the swirling film flow model, high-speed visualization of a string cavitation in a transparent mini-sac nozzle by visible light with refractive index matching for diesel as well as high-speed X-ray phase contrast imaging of a hollow-cone spray are conducted to measure the diameter of a single string cavitation spray cone angle and the thickness of the swirling liquid film along the axis. Then the circulation upstream of the orifice entrance is measured by PIV analysis. As a result, we conclude that (1) the proposed model on the swirling film flow with single string cavitation can quantitatively predict the motion of an annular swirling film flow in an orifice and a hollow-cone spray, and (2) spray cone angle is determined by the ratio of the axial and azimuthal velocities of the annular swirling film flow in an orifice.

Keywords: String Cavitation, Hollow-Cone Spray, Mini-Sac Nozzle, Diesel Fuel Injector, Visualization, X-Ray

*Corresponding author: sou@maritime.kobe-u.ac.jp

Vortex Flow Pattern with Single or Twin String Cavitation in Multi-Hole Mini-Sac Diesel Injector and Sprays

Samsu Dlukha Nurcholik¹, Takashi Miwa¹, Akira Sou^{1*},
Mikimasa Kawaguchi², Yuhei Matsumoto², Keiya Nishida²,
Yoshitaka Wada³

¹Graduate School of Maritime Sciences, Kobe University, Kobe, Japan

²Graduate School of Advanced Science and Engineering, Hiroshima University, Hiroshima, Japan

³Engineering Analysis Group. No.1, Mazda Motor Corporation, Hiroshima, Japan

Abstract

Several patterns of vortex flows accompanied by single or twin string cavitation may occur in multi-hole mini-sac diesel fuel injectors, which promote greatly the atomization and dispersion of fuel sprays. However, the mechanism and condition of the formation of the vortex flows with single or twin string cavitation have not been clarified yet. In this study, high-speed visualization and PIV analysis of the flows in various transparent injectors were performed to investigate the effects of the injector geometry, needle lift, and fuel flow rate on the flows. We manufactured a lot of injectors with different orifice numbers, needle shapes, sac diameters, and needle lifts to reveal these effects. The results showed that (1) there are three types of vortex flow and string cavitation, these are single cavitation and swirling flow pattern, twin cavitation and toroidal vortex flow pattern, and simple downstream flow pattern without string cavitation; (2) low needle lift promotes strong vortex flow due the large local momentum in the sac; (3) short distance between needle surface and orifice inlet induces single string cavitation by the bathtub flow effect; (4) smaller sac enhances strong vortex flow and increases spray angle by larger inlet momentum and smaller diffusion; (5) appearance of single and twin string cavitation flows can be predicted using the string cavitation regime map based on the new dimensionless parameters, Re_s and Re_1 ; and (6) we can roughly estimate the spray angle using Re_s and Re_1 and modified cavitation number σ_c .

Keywords: Diesel Injector, String Cavitation, Visualization, PIV Analysis, Dimensionless Parameters, Sprays

*Corresponding author: sou@maritime.kobe-u.ac.jp

Breakup Mechanism of Flash Boiling Sprays Based on Rim Structures

Shangning Wang¹, Shangze Yang², Yijia Zhang¹, Xuesong Li¹, David L.S. Hung³, Min Xu^{1*}

¹School of Mechanical Engineering, Shanghai Jiao Tong University

²Institute for Multiscale Thermofluids, School of Engineering, University of Edinburgh

³University of Michigan-Shanghai Jiao Tong University Joint Institute, Shanghai Jiao Tong University
Shanghai, 200240, China

Abstract

Flash boiling is an important phenomenon in internal multiphase flow and could lead to unique spray structures. Many researchers have investigated this phenomenon using optical diagnostic methods, and progress has been made in spray morphology studies as well as bubble dynamics inside the nozzle. However, the detailed transition from in-nozzle bubble to out-nozzle spray droplets has not been experimentally observed, and present theories lack solid proof. To illustrate the true transition procedure from bubble to droplets of flash boiling bubbles, high-speed imaging technique is adopted on a specially designed optical nozzle with a two-dimensional exit to investigate the transition from innozzle flash boiling bubbles to out-nozzle spray droplets. A unique rim structure has been observed for the first time, quantitative and qualitative analyses are made to figure out the true self of the rim. Based on the study, the driving force, as well as the transition process of flash boiling bubbles, has been illustrated in detail, it is found that under high superheated degrees, the growth speed limit would also influence the bubble burst process. Based on the observed phenomena and theoretical analysis, a new flash boiling spray breakup model that coincides with the actual physical process has been established.

*Corresponding author: mxu@sjtu.edu.cn

Experimental investigations of the phase change impacts on flash boiling spray propagations and impingements

Shuyi Qiu¹, Di Xiao¹, Xuan Zhang¹, Shangning Wang¹, Tongyang Wang¹, David Hung²,
Xuesong Li^{1*}, Min Xu¹

¹School of Mechanical Engineering, Shanghai Jiao Tong University Dongchuan Road 800, Shanghai 200240, China

²University of Michigan-Shanghai Jiao Tong University Joint Institute, Shanghai Jiao Tong University, 800 Dongchuan Road, Shanghai 200240, China

Abstract

Flash boiling atomization technique has been considered a potential method for spray breakup performance enhancement utilizing the micro-explosion of superheated liquid. But the strong evaporative features of flash boiling atomization are not thoroughly investigated and understood. Therefore, this work focuses on the phase change impacts of flash boiling atomization and measures the volume flux of the liquid component during flash boiling atomization using phase Doppler interferometry. This work further investigates the consequence of flash boiling spray collapse in the aspect of spray impingement, with the two-dimensional thickness and temperature distribution measured by laser-induced exciplex fluorescence technique. This investigation verifies that due to the phase change of flash boiling atomization, the liquid component inside the plume drops significantly so that the wall impingement performance cannot be evaluated with spray penetration length only. Such observations also hold that even spray collapse takes place using a multi-hole fuel injector.

Keywords: Flash boiling sprays; Spray impingement; Wall film formation; Laser diagnostics; Volume flux

*Corresponding author: xuesonl@sjtu.edu.cn

Atomizer Design - I

Influence of swirl number and air flow rate on the spray characteristics of a microchannel based rotary atomizer (MCR)

Anjith Kumar*, T. Sundararajan

Thermodynamics and Combustion Engineering Lab and

National Centre for Combustion Research and Development (NCCRD-IIT-Madras),

Indian Institute of Technology-Madras,

Chennai, Tamil Nadu-600036, India

Abstract

A novel rotary atomizer with multiple micro-channel (MCR) grooves for fuel injection is developed. The width and height of the micro-channels are approximately 0.4mm. The injector is tested for a range of flow conditions at different rotational speed (N) values. The near field jet break-up characteristics have been presented for $N = 4000$ RPM at the air flow rates of 1000 LPM and 2000 LPM. High speed shadowgraphy has been employed to capture the time resolved images of jet break-up. Secondary atomization characteristics have been obtained from phase Doppler particle analyzer for different aerodynamic Weber number (We_g) and air swirl number (S) values at different axial planes. Arithmetic Mean diameter and probability density functions illustrate a nearly uniform spray of fine droplets obtained from the MCR rotary injector.

*Corresponding author: me18b001@iittp.ac.in

Single Orifice Effervescent Atomizer in simulated Counter-flow Conditions

Ondrej Cejpek^{*1}, Milan Maly¹, Miloslav Belka¹, Louis Decanay², Madan Mohan Avulapati³, Jan Jedelsky¹

¹Energy Institute, Brno University of Technology, Brno 616 69, Czech Republic

²Advanced Engineering Center, University of Brighton, Brighton BN2 4AT, United Kingdom

³Indian Institute of Technology Tirupati, Renigunta Road, Tirupati – 517 506, India

Abstract

CO₂ is the most severe greenhouse gas released into the atmosphere and is responsible for the global temperature increase. The amount of CO₂ released can be reduced through deployment of post-combustion CO₂ capture. Spray columns, as simple gas cleaning contractors, utilise different types of atomizers e.g. pressure atomizers, twin fluid atomizers and rotary atomizers. The CO₂ capture process requires well-tailored spray characteristics, such as mean drop size, drop size distribution, droplet density, spray cone angle (SCA) and droplet velocity. Effervescent atomizers are beneficial, in that way they can be used for spraying viscose chemical solutions to counterflowing flue gasses. However, they typically create a wide droplet size distribution. Small droplets from this wide distribution are carried away by the ambient flow and cause significant sorbent losses. The total surface area, which determines the rate of mass transfer of CO₂, is significantly reduced by this droplet drift. In this study, the effervescent atomizer was tested under realistic flow conditions relevant to spray columns. A small-scale vertical wind tunnel was used to simulate flow velocity in the range from 0 to 10 m/s with a turbulent intensity lower than 3%. Phase Doppler anemometry and high-speed visualization were used to capture the droplet behaviour at several axial locations from the atomizer exit orifice. The liquid discharge was not influenced by the ambient flow. The effect of ambient flow was more prominent far from the atomizer, where the droplet momentum rapidly decreased. Droplets smaller than 20 µm were completely carried away by the ambient flow.

^{*}Corresponding author: Ondrej.Cejpek@vutbr.cz

Characterization of primary atomization processes of like on unlike impinging injectors

Bikash Mahato¹, Vikram R.², Ganesh P.³, K.P. Shanmugasdas¹

¹Department of Mechanical Engineering, IIT Jammu, India-181221

²National Centre of Combustion Research Centre (NCCRD), IIT Madras, Chennai, India- 600036

³ISRO Propulsion Complex, Mahendragiri, India- 627133

Abstract

The present work focuses on the primary atomization processes of like-on-unlike impinging injectors, using various laser diagnostic techniques to understand the flow regimes, sheet formation, and breakup dynamics over a wide range of average weber numbers. Backlight imaging and time-resolved planar laser-induced fluorescence imaging (TRPLIF) techniques are used both in front and side view configurations to understand the physical characterization of the sheet breakdown process. The linear sheet parameters like sheet width, sheet thickness, breakup length, and sheet angle are quantified for average weber numbers varying from 188 to 2994. The breakup length increased at $We_{avg} \sim 270$ and then it decreases. At lower weber numbers, the impact waves dominated the sheet breakup process whereas at higher weber numbers, perforation dominated the sheet breakup is observed. The ligament shedding frequencies are correlated with the impact wave frequencies and major spatial modes are obtained using proper orthogonal.

¹Bikash Mahato: 2020pth0048@iitjammu.ac.in

Study on real time gasoline-water mixture fuel preparation and spray characteristics based on impingement method

Meng Ji^{a,b}, Zhijun Wu^{*b}, Alessandro Ferrari^a

^aSchool of Automotive Studies, Tongji University, China, 200092, Shanghai

^bEnergy Department Politecnico di Torino, University of Polietecnico di Torino, Italy,
10129, Torino

Abstract

With the aggravation of environmental pollution and the increasing energy consumption, it is urgent to develop internal combustion engine technology with high efficiency and low emission. Engine water injection technology has great potential in alleviating engine detonation and reducing emissions and has been widely used and studied. Gasoline-water mixed injection technology is a unique water spraying technology. It has great application potential because of its advantages of reducing modification cost and improving atomization. In this work, based on the principle of impinging flow, an real-time gasoline-water mixing preparation system for internal combustion engine had been designed and the preparation performance in different influencing factors were analyzed firstly; Than, based on mixture Flash Boiling Spray platform, spray characteristics of the spray angle, spray area and spray size distribution of gasoline-water mixture under different mixing conditions, different temperatures (30-160 °C) in 5 MPa pressures were analyzed. The experiment of gasoline-water mixture preparation showed that the impact pressure was the main factor affecting the emulsification performance of the real-time gasoline-water mixture, and the real-time mixing system could produce stable gasoline-water emulsion. The flash boiling spray experiment of gasoline-water mixture fuel showed that with the increase of temperature, the spray penetration distance decreased first and then increased, which corresponded to the flash boiling point of each gasoline-water mixture; The spray cone angle decreased first and then increased with the increase of temperature; The particle size of gasoline-water mixture was smaller than that of pure gasoline; Adding water can help to reduce the particle size under flash boiling conditions.

*Corresponding author: zjwu@tongji.edu.cn

Bubbles and Micro-explosions

Ammonia flash break-up modeling based on bubble dynamics with correlating analysis of surface tension and pressure force

Jisoo Shin^a, Sungwook Park^{b*}

^aDepartment of Mechanical Convergence Engineering, ^bDepartment of Mechanical Engineering

Hanyang University

Seoul, Seongdong-gu, 04763, Republic of Korea

Abstract

Role of ammonia as an alternative fuel is becoming important as carbon regulations tighten around the world. In the use of ammonia fuel as a fuel of an internal combustion engine, the use of dual-fuel and flash boiling phenomena should be considered due to the high self-ignition temperature and vapor pressure of ammonia. Therefore, in this study, the droplet breakup was modeled using KH-RT model for aerodynamic breakup and flash breakup model for thermodynamic breakup. CONVERGE v3.0 program was used for simulation and flash breakup model was inserted by userdefined function code. Based on bubble dynamics, the bubble inside the fuel droplet generation and growth was calculated. And the number density of bubbles in the droplet changes depending on the superheating degree. To reflect this effect on the model, the initial bubble size is determined for various superheating degree. The flash breakup occurred when the force by pressure difference between droplet inside and the ambient was greater than the surface tension force, and the change in the velocity of the droplet was predicted through the energy change. As a result, the results using flash breakup model more accurately predicted the spray development process for different ambient temperature conditions.

*Corresponding author: parks@hanyang.ac.kr

Study of air bubble entrapped droplet evaporation on oil-coated surfaces

D. Ramajayam and M. Vadivukkarasan*

Department of Mechanical Engineering

National Institute of Technology Puducherry Karaikal, 609609, India

Abstract

Understanding entrapped bubbles are ubiquitous in industrial and environmental phenomena. Evaporative cooling of microelectronic devices with a ternary system attracts study on evaporation of liquid droplets with air bubbles trapped on heated substrates. An air bubble inside the liquid droplet leads to rupture in such applications to form several tiny drops. The primary interest of the present study is to identify how the oil-coating surface affects the evaporation rate, growth, ageing, and death of air bubbles entrapped in the liquid droplets at specified temperatures. We systematically investigate the controlled environment using the customized temperature-controlled contact angle meter. This experimental study explores the lifetime of the bubble ebullition occurrence at (i) solid-liquid and (ii) liquid-liquid interfaces in the heated substrates. This study attempts to reveal the lifetime of the various sizes of the liquid bubble (say 5 microliters) on the oil-coated heated substrates. Our study is directly relevant to the ternary evaporative system (oil-air bubble–liquid droplet) over the binary evaporative system (air bubble–liquid droplet). This work can be regarded as the extension of predicting the lifetime of the bubble on the solid surface at room temperature [Poulain, Villiermaux and Bourouiba, J. Fluid Mech. (2018)].

Keywords: air bubble, oil coated substrate, life time of a multi-component droplet, evaporation

*Corresponding Author: m.vadivu@nitpy.ac.in

A model for prediction of inflight micro explosion in bicomponent fuel droplets

Abhiram Gummula*, Himanshu Singh, Madan Mohan A

Department of Mechanical Engineering

Indian Institute of Technology Tirupati

Tirupati, Andhra Pradesh, 517507, India

Abstract

Microexplosion is a phenomenon where a parent droplet is disintegrated into multiple child droplets due to evaporation of volatile components inside a multi component liquid droplet. Fuel droplets tend to undergo micro explosion when there is a large difference in the volatility of different components. With use of alternative fuels such as Ethanol blending with diesel, large differences in the boiling points tend to create microexplosion of the droplets when sprayed into high temperature conditions inside the engine. Blending of water with diesel to reduce the emissions can also create microexplosion inside the droplets. In spray combustion processes like CI engines, droplets will undergo heating as well as evaporation during their flight. Predicting the time required for microexplosion will be helpful in understanding whether droplets will undergo an explosion before they completely evaporate or encounter combustion chamber surfaces. In this paper, a droplet micro explosion model is integrated with a droplet trajectory model for predicting the time taken for microexplosion during the flight of the droplet. The basic model is taken from the work of Sazhin et al and incorporates changes in properties as well as droplet surface temperature with time along with droplet kinetics to find the trajectory. The integrated model is observed to predict the micro-explosion time closer to the experimental values reported in the literature for the droplet range 50 μm to 150 μm . The results showed that the time for puffing increases with increase in droplet size and decreases with increase in surrounding temperature, initial droplet temperature and initial water concentration.

*Corresponding author: me18b001@iittp.ac.in

Droplet Evaporation Characteristics of Nanobubble-Liquid Suspensions

K Vivek* and Kabiraj L

Department of Mechanical Engineering

Indian Institute of Technology Ropar, Rupnagar, Punjab, 140001, INDIA

Saurabh A

Department of Mechanical Engineering

Indian Institute of Technology Kanpur, Kanpur, UP, 208016, INDIA

Abstract

We report experimental observations on the comparison between the evaporation of suspended droplets of stable nanobubble suspensions (oxygen bubbles in water) and water. Single suspended droplet evaporation of pure water and oxygen-gas nanobubbles-water was studied through experiments. The droplets were exposed to a laminar stream of air at Reynolds number 1700-1800. Results show that with the presence of nanobubbles, a few of the tested nanobubble water droplets underwent faster evaporation.

*Corresponding author: 2018mez0025@iitrpr.ac.in

Cavitation and Flash Boiling – II, Other topics

Study on Laminar Combustion Characteristics of Flash Boiling Spray under Cold Start Environment

Mingli Cui¹, Zhe Sun², Yansong Li³, Xuesong Li^{4*}, Min Xu⁵

School of mechanical and engineering

Shanghai Jiao Tong University

Shanghai, 200240, China

Abstract

Due to poor fuel atomization and evaporation effects in cold engine startup environment, gasoline direct injection engines tend to generate a large number of soot particles under the startup conditions. Flash boiling spray by elevating the fuel temperature is a potential way to solve the problem. In this paper, the spray is decoupled from the in-cylinder dynamic environment, and the combustion characteristics of the fuel-air mixture produced by the flash boiling spray under cold start environment are investigated in a constant volume combustion chamber. The results show that the spray morphology is significantly changed by elevating the fuel temperature. It is easier for the flash boiling spray to be ignited and form a stable flame kernel and the overall flame propagation speed of the flash boiling spray at the early stage is faster. The Markstein length of the flame generated by the flash boiling spray mixture is larger, which means more stable flame propagation process of the flash boiling spray. Compared with subcooled spray, the flash boiling spray effectively reduces soot particulate generation.

Investigations of string cavitation and air back suction during injection duration in the scaled-up diesel fuel injection nozzle

Chen Li^{a,b}, Zhixia He^a, Wei Guan^{a,b}, Genmiao Guo^a, Tianyang Bai^a, Bihe Hu^a

^aInstitute for Energy Research, Jiangsu University, Zhenjiang 212013, China

^bSchool of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China

Abstract

The flow inside the diesel fuel injector nozzle is directly associated with the subsequent jet atomization and fuel-air mixture performance, and finally affects the combustion efficiency and emission. In this study, high-speed microscope imaging technology is applied to capture the complex multiphase flow and near-field spray during the process of fuel injection. Besides normal optical experiments for nozzle internal flow and spray, the submerged jet experiment for comparison is also carried out to investigate the phenomenon of air back suction. Combined with the simulation of a three-component multiphase model, the difference between air and cavitation vapor is analyzed. Results show that the submerged jet experiment will not change the internal flow field but can eliminate the influence of air. The shaded area in the hole obtained by the shadow method may be the sucked air or vapor generated by string-type cavitation phase change. Both can coexist in the hole and are closely related to the swirling intensity and the spray structure, respectively. When the needle lift is low, the internal flow field has larger swirling intensity resulting in a kind of hollow spray. The low-pressure region exists in the vortex core, nearly at the axis of the nozzle orifice. The ambient air will be sucked into the orifice to form an air column under the action of pressure difference, which is also easy to occur under low injection pressure. When the needle lift is high, the weak swirl intensity leads to hole-to-hole string cavitation which has weak cavitation intensity and obvious periodicity. At this time, the results of non-submerged and submerged jet experiments are basically the same. In addition, the cavitation bubble near the orifice outlet was observed, for the first time, to move upstream under the action of the pressure difference inside and outside the orifice in the submerged jet experiment. It can be predicted that in the non-submerged jet experiment, the ambient air will be sucked into the orifice to form an air column.

Keywords: Diesel injection nozzle, String cavitation, Air back suction, Multiphase simulation, Submerged jet experiment

The Cavitation Flow and Spray Characteristics of Gasoline-diesel Blends in the Nozzle of a High-pressure Common-rail Injector

Hu Bihe, Wang Jianquan, Guo Genmiao, He Zhixia*, Feng Zehao

Jiang Su University

Zhen Jiang, Jiang Su, 212013, China

Abstract

In compression-ignition engines, gasoline-diesel blends has been widely concerned for its potential to obtain 50% thermal efficiency and ultra-low emissions. Due to physical property difference between the two fuels, the spray atomization characteristics of the blended fuels are different from that of a single fuel, and the cavitation flow characteristics of the blends in the high-pressure nozzle and their impact on the spray are not well understood. In this paper, the visualization of cavitation flow and spray characteristics of five different proportion blended fuels (D100G0, D80G20, D60G40, D40G60, D20G80) being injected through the taper hole nozzle with the outlet diameter of 0.22mm under different injection pressures was carried out based on the high-pressure common rail fuel injection system, in which the real-size transparent nozzle tip was employed. The optical study reveals the influence of fuel properties corresponding to the gasoline-diesel mixing ratio on the initial generation of vortex-induced cavitation in the nozzle, the development of transient cavitation flow process and the injection rate and spray characteristics. For this tapered orifice nozzle, string cavitation is captured mainly concentrating in the needle opening initial period and closing end period, which is the main factor of spray instability and fluctuations. In the initial period of injection, the addition of gasoline into the diesel fuel resulted in a significant increase in string cavitation strength compared to the pure diesel, even if the percentage of gasoline added is small. For the same injection conditions, the injection rate of gasoline-diesel blends is greater than that of diesel. The different gasoline addition ratios which mainly affects the initial moment of cavitation have little effect on the maximum intensity of string cavitation and the duration of string cavitation in the nozzle. Accordingly, the addition of gasoline increased the overall spray cone angle and spray fluctuations throughout the injection process. At the same time, as the injection rate and the single injection volume increased, the injection rate curve in the initial process is flat at first and then become steep which was precisely related to the cavitation of the vortex line in the nozzle. The findings of this paper are important guidelines to the determination of fuel injection strategies and spray analysis of gasoline-diesel blends for compression-ignition engines.

*Corresponding author: zxhe@ujs.edu.cn

Prediction of Cavitation Erosion in Diesel Fuel Injector

Natsuha Okami¹, Akira Sou¹, Toshiyuki Saito²

¹Graduate School of Maritime Sciences, Kobe University, Kobe, Japan

²NICO Precision Co., Inc., Niigata, Japan

Abstract

Due to the increase in injection pressure of the recent Diesel fuel injection system, cavitation erosion may take place on the surfaces of the orifices, sac and needle valve of the fuel injectors, which has to be avoided for stable fuel injection. However, the occurrence mechanisms of the cavitation erosions at these positions have not been clarified yet and the method to predict local cavitation has not been established yet. In this study we develop a simple prediction model of cavitation erosion at the orifice entrance and the sac entrance in the mini-sac nozzles by calculating the local pressure based on the Bernoulli's equation. In order to verify the validity of the model we manufacture a transparent large-scaled multi-hole mini-sac nozzle and visualize cavitation at various needle lifts and fuel flow rates by using refractive index matching technique. Then, we simulate cavitation erosion during the transient injection process of a real-scale fuel injector with a moving valve by using the proposed model. As a result, we observe cavitation inception and development at the orifice and sac entrances in the transparent injector. We confirm that the model can quantitatively predict cavitation inception at these positions with various needle lifts and fuel flow rates, by which we verify the validity of the model. Finally, we find that cavitation erosion at the sac entrance occurs at the moment of low needle lift and that at the orifice entrance takes place at the moment of full needle lift.

Keywords: Cavitation erosion, Fuel injector, Visualization, Refractive index matching

*Corresponding author: sou@maritime.kobe-u.ac.jp

Exploring evaporation and atomization characteristics of DMM_n and ethanol/DMM_n binary droplets during combustion

Bingyao Huang¹, Xiaoyuan Yang¹, Yi Zhang¹, Haodong Zhang¹, Wei Li¹, Yuyang Li^{1*}

Institute of Aerospace Propulsion, School of Mechanical Engineering

Shanghai Jiao Tong University

Shanghai 200240, P. R. China

Abstract

Polyoxymethylene dimethyl ethers (DMM_n) is a promising renewable synthetic fuel family from CO₂ feedstock, while their droplet combustion characteristics are still insufficiently understood. This work reports an experimental investigation on the evaporation and atomization characteristics of DMM_n ($n = 1-4$) and ethanol/DMM_n binary droplets, with special attentions on the influence of concentration, light component and boiling point difference. History of normalized square of droplet diameter is recorded during the combustion process, while two stages including stable evaporation and atomization are observed. Three different behaviors, i.e., puffing, ejection and micro-explosion, are observed in the atomization stage of the studied binary fuel droplets with the boiling point difference (ΔT) generally greater than 50 K, while only puffing and ejection can be observed as ΔT becomes lower. For D1/D3 and D1/D4 binary droplets, the maximum atomization indices are observed at 70 vol.% of DMM₁, while for the ethanol/D4 binary droplet, the maximum atomization index happens at 50 vol.% of ethanol. For both DMM_n and ethanol/DMM_n binary droplets, the atomization index is proportional to ΔT while the atomization delay time and micro-explosion intensity reach a limited value at higher ΔT . And with the same concentration, ethanol/DMM_n binary droplet have a higher micro-intensity and thus a higher atomization index, indicating their stronger atomization propensities than the DMM_n binary droplets.

*Corresponding author: yuygli@sjtu.edu.cn

Atomizer Design - II

Semi-empirical Prediction of SMD Spatial Distribution Downstream of Swirl cups at Different Air Pressures

Donghui Wang¹, YongHuang^{1*}

National Key Laboratory of Science and Technology on Aero-Engine Aero-thermodynamics,

School of Energy and Power Engineering

Beihang University

Beijing, 100191, China

Abstract

The spatial distribution of fuel droplets downstream of swirl cups plays a crucial role in combustor performance such as combustion efficiency, exit temperature profile, smoke, combustor dynamics, etc. At present, experimental studies provide a valuable tool in the development stages of the combustor components, and the semi-empirical methods are greatly needed to optimize designs with limited hardware testing. However, the empirical/ semi-empirical models proposed over the past few decades are focused primarily on the mean droplet sizes. The objective of this work is to establish a semi-empirical model of Sauter mean diameter (SMD) spatial distribution, which could be helpful to the design of the spray from a swirl cup. The normalized method is used to obtain the semi-empirical prediction model of SMD spatial distribution. And the effect of air pressure on the SMD spatial distribution is reproduced by the empirical constants in the model. The spray diagnostics were carried out by a two-component phase Doppler particle analyzer technology to verify the model. The experimental and predicted results show that with the increase of air pressure, the local SMD in the area close to the central axis increases first and then decreases, while the local SMD in the outer periphery decreases. In the center area, the local SMD is kept nearly unchanged for different air pressures. The model predictions agree well with the experimental results of the SMD spatial distribution, and the correlation coefficients are more than 0.81.

Keywords: swirl cups, SMD spatial distribution, Semi-empirical prediction, PDPA, air pressures

*Corresponding author: yhuang@buaa.edu.cn

Temperature Effect on the Breakup Characteristics of Ultra-high-pressure Spray of a Single-hole Injector

Tianhao Pu, Shengqi Wu*, Mingyu Xie, Yanshuai Pang

Shanghai Jiao Tong University

Shanghai, China

Abstract

In this paper, breakup characteristics of a single-hole gasoline direct injection injector were investigated in a wide range of fuel temperatures and injection pressures. High-speed Mie-scattering imaging technique was used to capture the macroscopic spray structure. Fuel pressure varied from 100 bar to 1000 bar, and fuel temperature ranged from 21 °C to 120 °C. Ambient pressure was adjusted from 0.5 bar to 4.5 bar to mimic a variety of actual engine conditions. Several semi-empirical models for predicting spray penetration were investigated, including the Dent's model, Hiroyasu and Arai's model, and Naber and Siebers' model. Results showed that the Hiroyasu and Arai's model demonstrated the best match with the experimental spray penetration at fuel temperature of 21 °C for all injection pressures. For evaporating conditions, one parameter dependent with fuel temperature was added into the models. As a result, the modified Hiroyasu and Arai's model provided the closest predicted penetration values at all conditions, and the breakup time and length derived from the modified model fitted well with that from the experimental data. Consequently, more information about the breakup characteristics of ultra-high-pressure spray was revealed.

Key Words: single-hole fuel spray, high speed Mie-scattering, empirical models, breakup characteristics, high fuel temperatures

*Corresponding author: pth123@sjtu.edu.cn (T. Pu), shengqiwu@sjtu.edu.cn (S. Wu)

Development and characterization of self-aspirating liquid fuel spray ejector for micro combustion applications

Mohammed Asad Khan*, Hrishikesh Gadgil, Sudarshan Kumar

Affiliation

Department of Aerospace Engineering, Indian Institute of Technology Bombay

Mumbai, Maharashtra, 400076, India

Abstract

This paper presents a novel liquid fuel injection strategy for microcombustion applications. The proposed injector exploits the vacuum generation aspect of expanding gas jet for liquid fuel self-spiration. The emerging annular liquid film manifests shear instability at the liquid/gas interface due to aerodynamic effects and subsequently breaks ligaments and fine micron droplets. The self-aspiration mechanism of the annular liquid sheet was explained using volume-of-fluid simulations. The computational fluid dynamics simulations have demonstrated the negative pressure inside the injector; therefore, the liquid flow rate decreases simultaneously with recess length and atomizing gas pressure. The results from the numerical study were employed for the development of a low flow rate spray injector. The characteristic of the spray injector was studied using time-resolved shadowgraphy techniques. The experiments show that an injector ejects 1.0 – 4.5 cc per minute of water with 0.62 – 1.78 bar gas pressure. The investigated spray injector provided water droplets with Sauter mean diameter of 74 – 130 μm range just below (10 mm) the injector exit. The present investigation suggests that the developed self-aspirating spray injector generates very fine droplets at a low flow rate with minimum air and energy requirement, which is beneficial for the development of standalone microcombustors for portable power applications.

*Corresponding author: masaadkhan@gmail.com

Geometrical Sensitiveness of a High Shear Injector Over Spray Flow Field: Mean and Dynamical characteristics

Sonu Kumar¹, Saptarshi Basu^{*2}

¹Interdisciplinary Centre for Energy Research

²Department of Mechanical Engineering

Indian Institute of Science

Bangalore, Karnataka, 560012, India

Abstract

The present study is conducted to understand the global evolution and dynamics of the spray flow field in a high shear injector with different geometrical parameters. High shear injector, used in gas turbine combustor, includes a series of radial air-entry swirlers (primary and secondary) with a diverging flare at the exit and fuel nozzle mounted at the centre of the swirler. We have investigated the effect of flow split-ratio (Δ), flare angle (θ), flow orientation of primary and secondary swirlers. Time-resolved PIV (~ 3500 frames/sec) has been employed to perceive the topological structure of the flow field. The study is further extended to examine the influence of the above-mentioned parameters on the dynamic behaviour of the spray flow field down the swirler exit using POD techniques. The spatial eigen modes and frequency signature show that change in split-ratio alter the size of recirculation zone (RZ). With increase in size of the RZ, recirculation strength decreases and dominating coherent structure shifts to low oscillation frequency. Keywords: Gas turbine injector, Spray flow field, Flow Split-ratio Time-resolved PIV, POD.

*Corresponding author: sbasu@iisc.ac.in

Insights to the Near-nozzle Flow Characteristics of Pressure Swirl Atomizers used in Aero Engines

J. K. Mulamootil^{a*}, M. Manish^b, B. Prasad^c and R. V. Ravikrishna^c

^a Mechanical Engineering Department, NIT Calicut, Calicut – 673601

^b Dept. of Mechanical Engg., Rajiv Gandhi Institute of Technology, Kottayam – 686501

^c Department of Mechanical Engineering, Indian Institute of Science, Bangalore - 560012 INDIA

Abstract

Fuel atomizers are highly specialized devices that play a crucial role in the functioning and performance of aero gas turbine engines. The development of tangential entry pressure swirl atomizers typically used for start-up requires careful design of the internal flow passages and orifice exit. Computational Fluid Dynamics (CFD) is very useful for design and optimization of the nozzle geometry. In the present work, detailed three-dimensional CFD simulations of the internal and near nozzle flow were carried out using a Volume-of-Fluid (VOF) approach with High Resolution Interface Capture (HRIC) to track the interface. The spray structure was analyzed for a range of flow rates and the effect of the outlet divergence angle and orifice shape were also studied. Numerical results provided insight to the flow physics and the evolution of the hollow cone structure. The resultant flow field was found to be strongly dependent on the internal flow passages, and particularly the orifice exit geometry. The numerical predictions of the spray cone angle showed excellent agreement when compared with experimental data obtained using laser-based shadowgraphy. To understand the importance of a sharp edge provided on the test nozzle, a simulation was performed by removing it. The results show a complete breakdown of the hollow cone structure owing to the Coanda effect demonstrating the extreme sensitivity of the near-nozzle flow field to the presence of a sharp edge at the nozzle exit.

*Corresponding author: jkmoshy@gmail.com

Droplets and Impact

Influence of Adjacent Droplet-plume Interaction on Spray Characteristics in Slinger Atomizers

Arshdeep Singh^{a,b,*}, Srikrishna Sahu^{a,b}, Dalton Maurya^c

^aThermodynamics and Combustion Engineering Laboratory, Department of Mechanical

Engineering, I.I.T. Madras,

^bNational Center for Combustion Research and Development (NCCRD), I.I.T. Madras, India

^cGas Turbine Research Establishment (GTRE), Bengaluru, India

Abstract

The slingers are a class of rotary atomizers that find application in small gas turbine engines. The slingers utilize the high rotational speed of the turbine to achieve liquid atomization. They contain an array of orifices on the periphery from which multiple liquid streams emerge due to the centrifugal force and disintegrate into small droplets due to interaction with the surrounding air. The present study experimentally investigates the effect of the interaction of droplet plumes from adjacent orifices on the overall spray characteristics in the slinger atomizers. Three slinger discs having a different number of orifices (2, 4, 18) are considered. For all cases, the orifices are drilled on the periphery of the disc so as to realize pairs of diametrically opposite orifices. All experiments are conducted using water as the working liquid under atmospheric conditions. The flow rate per orifice is maintained constant for all discs to compare the spray characteristics. The front-light illumination technique is used to visualize the primary liquid breakup close to the orifices. The Interferometric Laser Imaging for Droplet Sizing (ILIDS) technique is used for the droplet sizing. It is observed that the emerging liquid breakup structure is the same for all three discs for the same operating conditions. Droplet size results show variation in characteristic droplet size for all three discs for the same condition. It is observed that AMD increases with an increase in the number of orifices, which is an indication of the droplet-plumes interaction.

*Corresponding author: me19d053@smail.iitm.ac.in

Laser-induced breakdown in Acoustically Levitated Fuel Droplets

Vishal S. Jagadale¹, Yogeshwar Nath Mishra^{2,3}, Devashish Chorey¹, Mats Andersson⁴,
Dag Hanstorp², Devendra Deshmukh^{1*}

¹Department of Mechanical Engineering, Indian Institute of Technology-Indore, India

²Department of Physics, University of Gothenburg, Sweden

³NASA-Jet Propulsion Laboratory, California Institute of Technology, USA

⁴Department of Mechanics and Maritime Sciences, Chalmers University of Technology, Sweden

Abstract

The study of a single droplet in a quiescent medium has great significance to understand the fundamental physics of fluid dynamics, heat, and mass transfer. It results in a better understanding of liquid atomization in internal combustion engines, agriculture sprays, spray drying, and many other applications. However, fundamental research on laser-induced single droplet atomization and evaporation is limited. The present study is an attempt to understand the droplet fragmentation due to instabilities from the laser heating process. We focus on the plasma-induced breakup mechanism of single and multi-component fuel droplets from wavy sheet disintegration and instabilities. The plasma generation inside the droplets leads to intense breakup and fragmentation. A droplet is suspended in the air using an acoustic levitator where it is exposed to nanosecond or femtosecond laser pulses. The droplet sizes used are in the range of 0.52 mm in diameter, and laser energies per pulse are 1-20 mJ for a nanosecond and 1 mJ for femtosecond laser pulses. Initially, the shock wave propagation is observed due to the laser impact. Afterward, the droplet opens at one side and is subsequently stretched vertically where a liquid sheet is formed. The liquid sheet expands maximum (~500 μs) whereafter it shrinks due to surface tension. Small droplets and ligaments emerge from the circular edge of the sheet. Subsequently, these ligaments disintegrate into fine droplets called secondary atomization. Also, the fragments ejected from the rim impact the liquid sheet which creates capillary waves on the surface. The secondary droplet sizes and the number of droplets produced are deduced from image analysis where the Sauter Mean Diameter (SMD) is calculated for the different fuel droplets.

*Corresponding author: dldeshmukh@iiti.ac.in

Experimental Studies on Effect of Neighboring Droplets on Evaporation of Aqueous Ammonia and Monoethanolamine (MEA) Droplets

Gourav Parmar^{1*}, Vignesh Kumar D¹, Jan Jedelsky², Madan Mohan Avulapati¹

¹Indian Institute of Technology Tirupati, Tirupati, Andhra Pradesh, 517619, India

²Brno University of Technology, Faculty of Mechanical Engineering, Brno 616 00, Czech Republic

Abstract

The spraying of chemicals such as Monoethanolamine (MEA) and aqueous Ammonia (75% Water + 25% Ammonia) is widely used in spray columns for CO₂ removal from the exhaust and process gases. When the droplets in the spray interact with the flue gas, due to temperature difference, along with the absorption of CO₂ from the flue gas they undergo evaporation as well. Changes in the diameter as the droplet evaporates will change the surface area and gas concentration inside the droplets, influencing the rate of absorption of CO₂ into the droplet. Understanding the droplet evaporation and the influence of the surrounding droplets on the evaporation of droplets is one of the important aspects to address for developing reliable models for CO₂ capture from flue gases. This work focuses on studying single droplet as well as simultaneous evaporation of multiple droplets of MEA and aqueous Ammonia. The droplets suspended on a microfiber grid made up of 100 μ m glass fiber is introduced into the heating chamber and the evolution of the droplet size is recorded using backlight imaging. Images are processed using Matlab algorithms to obtain the droplets' evaporation rate. The presence of droplets in the neighborhood reduces the evaporation due to change in the ambient concentration of vapor. When the spacing between the droplets is larger, i.e. 5 mm (5 times the droplet diameter) in this case, increasing the number of droplets around the primary droplet has no significant effect on the rate of evaporation. However, the effect is pronounced when the spacing between the droplets is reduced to 2.5 mm (2.5 times the droplet diameter). Findings of these experimental results can be used in designing the atomizers for the removal of CO₂ from the flue gases.

Characterizing Sputtering Behaviors in Water/ethanol Bi-component Droplet Impact on a Heated Plate

X. Y. Yang, B.Y. Huang, Y. Zhang, Y. Y. Li*

Institute of Aerospace Propulsion, School of Mechanical Engineering

Shanghai Jiao Tong University

Shanghai, 200240, P. R. China

Abstract

Droplet impact on heated surfaces exists in various practical applications such as internal combustion engines, aeroengines, chemical engineering, and fire safety, where its significant role has been widely recognized. The complex flow, heat and mass transfer, and phase change processes in it have attracted extensive attention. Previous studies focused on mono-component droplet impact, while investigations on multicomponent droplets are still insufficient. This work experimentally investigates the dynamics of water/ethanol bi-component droplet impact on a smooth heated Al plate at a low Weber number (lower than the splashing threshold at room temperature) with a high-speed imaging system. The effects of surface temperature and mixture composition are explored. Five conditions of ethanol volume fraction are selected as 0%, 25%, 50%, 75%, and 100%, and the surface temperature ranges from 150°C to 360°C. Different from the mono-component droplet impact in which the parent droplet does not break up, the water/ethanol bi-component droplet impact exhibits pronounced breakup phenomena at specific temperatures and droplet compositions, including corona sputtering and prompt sputtering. Corona sputtering occurs at the ethanol volume content of 25% and 50% with a lower temperature range ($\leq 200^\circ\text{C}$). After impact, the edge of the droplet uplifts to form a corona, which undergoes a faster breakup process under the action of the upward lift force and the downward wall adsorption force. Prompt sputtering occurs when the volume fraction of ethanol is 25% and 50% with the wall temperature is higher than 220 °C, and the volume fraction of ethanol is 75% with the wall temperature is lower than 200 °C. After impact, the edge of the droplet violently breaks up into fragment droplets which eject almost horizontally, while the center of the droplet forms either a liquid lump filled with bubbles or a liquid jet that is launched vertically upward under the action of the bubble cloud and then disintegrates rapidly. The impact morphological differences between mono-component and bi-component droplets are compared. The counted number of the fragment droplets and the residence time of the two kinds of the droplet are discussed. Characteristic parameter analysis on the two specific sputtering phenomena of bi-component droplets is carried out. This work provides some new insights into the understanding of the dynamic characteristics of droplet impact and the regulation of secondary atomization.

*Corresponding author: yuygli@sjtu.edu.cn

Visualization and analysis of CNG wall-impingement jet characteristics under various injection pressures and impingement geometry conditions

Quangkhai Pham¹, Mengzhao Chang¹, Jeonghyun Park², Byungchul Choi^{3*}, Sungwook Park⁴,
Avinash Kumar Agarwal⁵, Suhan Park^{6**}

¹Department of Mechanical Engineering, Graduate School of Chonnam National University,
Gwangju 61186, Republic of Korea

²Department of Mechanical and Aerospace Engineering, Graduate School of Konkuk
University, Seoul 05029, Republic of Korea

³School of Mechanical Engineering, Chonnam National University, Gwangju 61186, Republic of Korea

⁴School of Mechanical Engineering, Hanyang University, Seoul 04763, Republic of Korea ⁵Department of
Mechanical Engineering, Indian Institute of Technology-Kanpur, Kanpur 208016, India

⁶School of Mechanical and Aerospace Engineering, Konkuk University, Seoul 05029, Republic of Korea

Abstract

Advanced injection strategies and alternative fuels for internal combustion engine have been extensively studied in recent decade. These are effective ways to enhance the mixture formation, combustion efficiency and engine performance to reduce harmful emission. Using CNG fuel make the engine cleaner and more efficiency when compared to conventional fuels, such as gasoline. This study focuses on investigating the effects of the low pressure ratio conditions ($4 < PR < 10$) and impingement geometry including impingement distance ($L_w = 20, 45$ and 50 mm) and angle ($\theta_w = 30, 60$ and 90°) on the CNG wall-impingement jet characteristics. Accordingly, the transient development of gaseous impingement jet was recorded by a high speed camera with Z-type Schlieren visualization techniques. The experimental results showed that impingement geometry has a mostly positive effect on the development of gaseous jet wall-impingement under low PR conditions compared with injection pressure. The pressure ratio is defined by ratio of injection pressure and ambient pressure. The effect of above factors on the jet structure were evaluated in a constant volume chamber at atmospheric conditions. This in turn resulted in the increase of jet tip penetration (STP) and jet height after the impingement with the increase of injection pressure. However, as the increase of impingement distance, STP increase while the spray height decrease. Additionally, the spray height increases and STP decreases on regardless of the impingement angle level, which has a great correlation with the increase of pressure ratios. Finally, the contribution index under these conditions on the structural parameters of wall-impingement jet were evaluated in this study.

*,**Corresponding author: suhanpark@konkuk.ac.kr, bcchoi@chonnam.ac.kr

Numerical study on splash of oblique hollow droplet impact on a liquid film

Tinglan Tang, Tai Jin*, Gaofeng Wang

School of Aeronautics and Astronautics, Zhejiang University

Hangzhou, Zhejiang, 310027, China

Abstract

Recently, there has been renewed interest in hollow droplets impaction since the great significance for thermal spraying, the 3D printing of foam material. Most of the existing studies are focused on the normal and oblique impingement of the continuous dense droplet on the liquid film, but only limited work has been done on hollow droplet impact. In order to understand the basic phenomena and the underlying mechanisms of the oblique hollow droplet impingement, we numerically study the dynamic characteristics of a hollow droplet obliquely impacting on the liquid film via volume of fraction (VOF) method in the present study. The formation and evolution of the splash during the process of oblique hollow droplet impact at different impact angles are investigated. The results demonstrate that, when the bubble diameter is 1/4 of the droplet diameter, the existing of the bubble has no apparent effect on the evolution of the crown morphology. With impact angle increasing, splashing is suppressed on the leeward side of the advancing droplet. There is a transition from a crown-shaped splashing to a non-crown-shaped splashing, and the crown height on both sides first increases and then decreases.

*Corresponding author: jintai@zju.edu.cn

PHANTOM[®]

BROADEST RANGE OF HIGH-SPEED CAMERAS

FOR SCIENTIFIC RESEARCH & DEVELOPMENT



- Camera models from 1 to 9 Megapixel Resolution
- Up to 75 Gpx/Second throughput
- Flex4K and VE04K for 4K up to 1000 fps
- TMX for ultrahigh-speed frame rates
- Workflow solutions to accommodate any environment



When it's too fast to see, and too important not to.[®]

PHANTOM[®]

PHANTOMHIGHSPEED.COM
+1.973.696.4500

ViSiON[®]
RESEARCH

AMETEK[®]



HIGH-SPEED CAMERAS FOR RESEARCH AND ENGINEERING



i-SPEED® 7 Series

2072 x 1536 @ 8,512 fps
3.2 Megapixel sensor
27+ GPx/s throughput



i-SPEED® 5 Series

Full HD resolution
@ 6,382 fps
13 GPx/s throughput



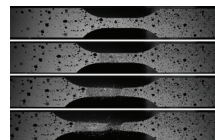
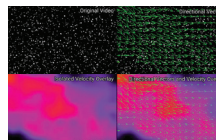
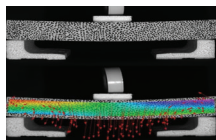
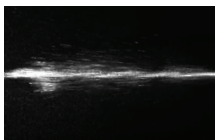
i-SPEED® 2 Series

1 MPx to 2.6 MPx sensors
up to 225,000 fps
Compact and portable
16GB internal RAM



CDUe

Industry-unique
handheld
camera control



Applications

Ballistics, PIV/DIC, Fluid Dynamics, Shock and Vibration, Material Science, Aerodynamics

ix-cameras.com

info@ix-cameras.com

Fluid Mechanics Systems

State-of-the-Art Fluid Mechanics Instrumentation

A photograph showing a man in a suit standing next to a large, yellow metal frame structure, likely part of a wind tunnel. Another person is visible in the background, and a model of an aircraft is being tested. The scene is dimly lit, with the primary light source coming from the wind tunnel's lighting system.

Tesscorn equips fluid mechanics and particle diagnostics researchers with the state-of-the-art hardware, software and complete measurement systems. These instruments provide diagnostics of fluid flow and particle/droplet sizing in a wide range of applications including: aerodynamics, hydrodynamics, spray diagnostics, multiphase flows, turbulent flows, and many more.

A Complete Portfolio of Fluid Mechanics Measurement Systems

Particle Image Velocimetry (PIV) Systems

- 2D & Stereo PIV
- Tomographic PIV
- Time-Resolved PIV & PTV
- Micro-PIV & Micro-PTV
- Thermographic PIV
- Underwater PIV
- Endoscopic PIV
- Educational PIV

Phase Doppler Interferometer (PDI) Systems

Laser Doppler Velocimetry (LDV) Systems

Laser-Induced Fluorescence (Mixing LIF)

- Planar Laser-Induced Fluorescence (PLIF) - Liquid & Gas
- Laser-Induced Incandescence (LII)
- Background Oriented Schlieren (BOS)

Shadowgraphy Systems

- 2D & 3D Shadowgraphy

Advanced Image Correlation

- 2D, 3D Deformation & Strain

High-Speed Imaging

Lasers

- Lamp-pumped Lasers
- Diode-pumped Solid State Lasers

Tesscorn Aerofluid Inc.

1172, 21st Cross, HSR Sector 3
Bangalore 560102

☎ +91.80.2572.9425

🌐 www.tesscorn-aerofluid.com

✉ info@tesscorn-aerofluid.com



Patalpani,

Thank you for your
participation in
ILASS-Asia 2022



Fort of Mandu,

