

ILASS-Asia 2016



18th Annual Conference on Liquid Atomization and Spray
Systems
Nov 06 – 09, 2016 Chennai, India

ILASS – Asia 2016

book of abstracts of the

18th Annual Conference on Liquid Atomization and Spray Systems

organized by

Combustion Institute (Indian Section)

supported by

National Centre for Combustion Research & Development, IISc, and
National Centre for Combustion Research & Development, IIT Madras

November 6 – 9, 2016, Chennai, India

Organized by



Combustion Institute (Indian Section)

Supported by



National Centre for Combustion Research and Development



Indian Institute of Science,
Bangalore



Indian Institute of
Technology Madras, Chennai

Program Committee

Conference Chair:

Prof. R. V. Ravikrishna, Indian Institute of Science, Bangalore

Program Chair:

- Prof. Shamit Bakshi, Indian Institute of Technology Madras, Chennai

Local Arrangement Chair:

- Prof. T.N.C. Anand, Indian Institute of Technology Madras, Chennai

Advisory Committee

Chairman

- Dr. V. K. Saraswat, Chairman, Combustion Institute (Indian Section)

Members:

- Prof. Masataka Arai, Gunma University and Tokyo Denki University
- Prof. Min Xu, Shanghai Jiao Tong University
- Prof. B. N. Raghunandan, Indian Institute of Science, Bangalore
- Dr. P. K. Pandey, Secretary, Combustion Institute (Indian Section)
- 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
- Prof. Satya Chakravarthy, Indian Institute of Technology Madras, Chennai
- Prof. R. I. Sujith, Indian Institute of Technology Madras, Chennai
- Prof. Saptarshi Basu, Indian Institute of Science, Bangalore
- Prof. D. Sivakumar, Indian Institute of Science, Bangalore
- Prof. Mun Soo Chon, Korea National University of Transportation
- Prof. (Emeritus) Norman Chigier, Carnegie Mellon University
- Prof. Nobuyuki Kawahara, Okayama University
- Prof. Wei-Hsiang Lai, National Cheng Kung University
- Prof. Chang Sik Lee, Hanyang University
- Prof. Keiya Nishida, Hiroshima University
- Prof. Jiro Senda, Doshisha University
- Prof. Shijin Shuai, TsingHua University
- Prof. Muh Rong Wang, National Cheng Kung University
- Prof. Ta-Hui Lin, National Cheng Kung 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

Sponsors

teesscorn



ANSYS®



Preface

The Institute of Liquid Atomization and Spray Systems (ILASS) was established for the mutual exchange of scientific and technical information in the field of atomization and sprays, and promotion of activities in each of the three regional ILASS bodies - ILASS-Asia, ILASS-Americas, and ILASS-Europe. ILASS-Asia is coming to India for the first time with this conference and provides a venue for academic and industrial researchers and students to meet and share recent developments in this field. This conference also marks the formal beginning of ILASS activities in India, under the auspices of the Combustion Institute (Indian Section). We have around 110 papers being presented at this conference with delegates participating from several countries in Asia such as China, Japan, Korea, Taiwan and also from USA and Europe, in addition to India. I wish all delegates a very productive and enjoyable conference!

R. V. Ravikrishna
Conference Chair

Contents

Keynote Lecture 1		7 th November, 2016 (Monday)
	Modeling of Dilute Spray Combustion: Interaction of Evaporation, Turbulence and Chemical Reactions	P 5
	Eva Gutheil	
Spray Characterization - I		7 th November, 2016 (Monday)
Paper no 48	Quantitative Analysis Of Various Empirical Equations For Fuel Sprays	P 8
	Peng Gu, Min Xu, David L. S. Hung, Shengqi Wu and Xue Dong	
Paper no 34	Spray characteristics in IC engines fuelled with straight vegetable oil	P 9
	Soo-Young No	
Paper no 55	An approach for including evaporation in a model for predicting spray penetration	P 10
	Gautham Krishnan, Cg Narendiran and Shamit Bakshi	
Paper no 87	Characterization of Sprays Discharging from Gas- Centered Swirl Coaxial (GCSC) Atomizer using PDI	P 11
	Athul Joseph, R Sakthikumar and D Sivakumar	
Paper no 108	Liquid Phase Penetration of n-Hexane Sprays at different Vapour Concentration of the Ambient	P 12
	Manas Pal and Shamit Bakshi	
Paper no 136	Spray Characterization Of A Multi Hole Port Fuel Heated Tip Injector	P 13
	Chandrasekar Sundaravelu, Anand T N C and Ramesh A	
Air-Assisted Atomization		7 th November, 2016 (Monday)
Paper no 28	Experimental study of liquid jet breakup in co-axial air blast atomization	P 16
	Abhijeet Kumar and Srikrishna Sahu	
Paper no 16	Breakup of Conical Liquid Film with and without Coaxial Gas Flow	P 17
	Zhongtao Kang, Zhen-Guo Wang, Qinglian Li and Peng Cheng	
Paper no 90	Role of the precessing vortex core on wall filming and ligament breakup inside a prefilming airblast atomizer	P 18
	Shanmugasadas K.P. and Chakravarthy S.R.	
Paper no 91	Atomization Studies On Coaxial Circular Injector And Coaxial Profiled Injector	P 19
	Anjaneesh Shukla, Rajakumaran Nair, Kanthasamy M, Remesh G, Jayan N, Suresh M.S. and Narayanan V	
Paper no 18	An Experimental Study of Spray Characterization of Various Air Flow Configurations in a Hybrid Atomizer	P 20
	Amlan Garai, Shinjan Ghosh, Swarnendu Sen and Achintya Mukhopadhyay	
Paper no 29	Measurement of primary break-up of liquid jet in a cross-stream airblast atomizer	P 21
	Shirin Patil and Srikrishna Sahu	
UWS Sprays		7 th November, 2016 (Monday)
Paper no 126	Droplet Evaporation and Spray Formation of Urea Solution in High Temperature Atmosphere	P 24
	Tomohiko Furuhashi, Siti Nor Ain Musa, Yoshio Zama and Masataka Arai	
Paper no 123	Numerical Modeling of adBlue Droplet Evaporation in the Context of SCR-DeNOx	P 25
	Kaushal Nishad, Amsini Sadiki and Johannes Janicka	
Paper no 69	Experimental Studies on an Evaporating Droplet of Urea Water Solution; Effect of Ambient Temperature and Initial Urea Mass Fraction on Evaporation Rate	P 26
	Mikhail Surendran, Anand T N C and Shamit Bakshi	

Internal Flows in Nozzles		7 th November, 2016 (Monday)
---------------------------	--	---

Paper no 137	X-Ray High-Speed Visualization of Cavitation in Fuel Injector and Discharged Liquid Jet Akira Sou, Takashi Miwa, Rubby Prasetya, Takumi Kasahara, Seoksu Moon, Yoshitaka Wada and Hideaki Yokohata	P 28
Paper no 133	The New Cavitation Bubble Model inside the Nozzle in Consideration of the Bubble Dynamics with Bubble Nucleation Model Shingo Murakami, Vikas Semwal, Masahiro Matsuura, Eriko Matsumura and Jiro Senda	P 29
Paper no 45	Effects of Internal Deposits on Sprays of a Gasoline Direct Injector Haoyi Song, Jin Xiao, Kai Yu, Xianpei Yang, Xinqi Qiao, Dehao Ju, Zhen Huang, Qi Yin, Xiaojin Bai and Xuewei Pan	P 30
Paper no 32	Influence of Pressure Oscillation Induced Klystron Effect on the Inner Flow and Spray Characteristics of Pressure Swirl Injector Peng Cheng, Zhongtao Kang, Huiyuan Chen, Xinqiao Zhang and Qinlian Li	P 31
Paper no 145	Study of Cavitation and 3D Needle Movement due to Erosion in Fuel Injection Nozzles Using Coupled Simulation Tools Bejoy Mandumpala Devassy, Wilfried Edelbauer and David Greif	P 32

Collision and Coalescence		7 th November, 2016 (Monday)
---------------------------	--	---

Paper no 44	High fidelity simulations of binary collisions of liquid drops Sarang Holay, Shashank Jaiswal, Rajesh Reddy and Raja Banerjee	P 34
Paper no 83	Experimental measurement of droplet coalescence in three-dimensional turbulence M Shyam Kumar, S.R Chakravarthy and M Mathur	P 35
Paper no 79	Droplet coalescence statistics in a low pressure multi-hole injector using high-speed imaging Rohit Singh Pathania, Satyanarayanan R. Chakravarthy and Pramod S. Mehta	P 36
Paper no 27	Experimental investigation of droplet collision and dispersion in interacting sprays Manish M and Srikrishna Sahu	P 37

Engine Sprays - I		7 th November, 2016 (Monday)
-------------------	--	---

Paper no 153	Microscopic Spray Investigation of Karanja Biodiesel and Its Effects on Engine Performance and Emissions Chetankumar Patel, Nikhil Sharma, Nachiketa Tiwari and Avinash Agarwal	P 40
Paper no 73	Characteristics of High Pressure Ethanol Spray Flames in Diesel Engine-Like Environment using Optical Diagnostics Madan Mohan Avulapati, Radboud Pos, Athanasios Megaritis, Jun Xia and Lionel C. Ganippa	P 41
Paper no 152	Spray Behavior of a GDI Injector at Constant Fuel Injection Pressure and Varying Engine Load Nikhil Sharma, Chetankumar Patel and Avinash Kumar Agarwal	P 42

Numerical Sprays - I		7 th November, 2016 (Monday)
----------------------	--	---

Paper no 116	Validation of Discrete Spray Models for a Movable Pintle Injector of Liquid Rocket Engines Kanmaniraja Radhakrishnan, Min Son, Keonwoong Lee and Jaye Koo	P 44
Paper no 116	Numerical Modeling on Liquid Fuel Injected Spray Break-up and Evaporation with Elevated Pressures in BERL Burner Ramesh Repaka and Gurunadh Velidi	P 45
Paper no 24	Numerical Simulation Of Extinction Conditions Of Counterflow Methane Diffusion Flames Using Polydisperse Water Spray Akaashdeep Singh, A. Harish and Vasudevan Raghavan	P 46
Paper no 7	Numerical Investigation Of Extinction Condition Of Flat And Curved Counterflow Laminar Diffusion Flame With Polydisperse Water Sprays Sourav Sarkar, Achintya Mukhopadhyay and Swarnendu Sen	P 47
	Numerical Study Of The Effect Of Fine Polydisperse Water Spray	

Paper no 9	On Laminar Flame Speed Of Hydrogen/Air Premixed Flames Stabilized In The Stagnation Flow Field Joydeep Munshi, Sourav Sarkar, Achintya Mukhopadhyay and Swarnendu Sen	P 48
Droplet Evaporation, Combustion and Impact		
7th November, 2016 (Monday)		
Paper no 20	Spreading Dynamics of Shear-Thinning Fluid Drop Impact on Hydrophobic Substrate Purushothaman Nandagopalan, Jerin John, Ankur Miglani and Seung Wook Baek	P 50
Paper no 43	Effect of ambient gas on pendant droplet evaporation Radhakrishnan S, Anand T N C and Shamit Bakshi	P 51
Paper no 139	Experimental investigation of droplet evaporation under forced convective conditions Saurabh Markandeya and R. V. Ravikrishna	P 52
Paper no 75	Study on Flame Spread Behavior in Two dimensional and Three-dimensional Droplet Clouds Considering Flame-spread limit Based on Microgravity Experiments of Flame Spread of Droplet Arrays Herman Saputro, Masato Mikami and Takehiko Seo	P 53
Paper no 119	Disruptive Combustion Behavior of Gelled Ethanol Fuel Droplets Ankur Miglani, Purushothaman Nandagopalan, Jerin John and Seung Wook Baek	P 54
Engine Sprays - II		
7th November, 2016 (Monday)		
Paper no 118	Droplet diameter measurement near a nozzle exit of a gasoline DISI injector Nobuyuki Kawahara and Eiji Tomita	P 56
Paper no 15	The Effect of Fuel Injection Timing on the Macroscopic Spray Characteristics in a Gasoline Direct-Injection Engine with Optical Access Ashish Bisht, Mayank Mittal and Harold Schock	P 57
Paper no 25	Liquid and Vapour penetration length estimation for GDI spray of Iso-octane and alcohols at elevated pressure and temperature conditions Rakesh Kale and Raja Banerjee	P 58
Paper no 67	Numerical Prediction of Purging of a GDI nozzle at the End of Injection Sandeep Mouvanal, Axel Burkhardt, Dhiman Chatterjee and Shamit Bakshi	P 59
Paper no 131	Experimental analysis of spray characteristics using fuel heating for D.I.S.I. engines Eriko Matsumura, Keitaro Imori, Yudai Sakai and Jiro Senda	P 60
Keynote Lecture II		
8th November, 2016 (Tuesday)		
	Artificial Control in Spray Combustion Process Based on Fuel Design Concept for Engine Systems Prof. Dr-Eng Jiro Senda	P 65
Spray Characterization – II		
8th November, 2016 (Tuesday)		
Paper no 35	Design and Analysis of a High Pressure Swirl Atomizer Abhijit Kushari, Suresh Lal and Pranay Matkar	P 68
Paper no 36	Spray Characteristics from a Twin-Fluid Atomizer with Internal Impactor Abhijit Kushari, Deepak Kumar, Tushar Sikroria, Pramod Kumar, P. V. C. Rao and N. V. Choudary	P 69
Paper no 51	Spray cross-sectional pattern analysis of a SIDi multi-hole injector under various flash boiling conditions Shangze Yang, Min Xu, David L. S. Hung and Shengqi Wu	P 70
Paper no 61	Experimental investigations on spray characteristics of potential fuels for advanced combustion engines M Murugesu Pandian and K Anand	P 71
Paper no 109	Experimental and numerical characterization of fatty acid and methyl ester sprays Rajan Lanjekar and Devendra Deshmukh	P 72

Paper no 110	Effect of biodiesel-ethanol blends on spray atomization Vasudev Chaudhari, Aniket Kulkarni and Devendra Deshmukh	P 73
Impinging Sprays		
	8th November, 2016 (Tuesday)	
Paper no 88	Like-on-Like Impingement and Atomization of Highly Viscous Fluids Irene Del Carmen Ruiz-Rodriguez , Madan Avulapati and Lionel C. Ganippa	P 76
Paper no 54	Experimental investigation of behavior and drop size of impinging spray at ultra-low temperature conditions Huijia Lv, David L. S. Hung, Min Xu, Xue Dong and Hujie Pan	P 77
Paper no 100	Drop size distribution effects in intersecting sprays Dhivyaraja Kumaran, Sri Vallabha Deevi and Mahesh V Panchagnula	P 78
Paper no 81	Mixing field and penetration of impinging twin liquid jets injected obliquely against super-sonic air crossflow Venu Ganapuram, Dr. Venkat S. Iyengar, Sathiyamoorthy K, Srinivas J, Prof. Muruganandam T M and Prof.Sathyanarayana R Chakravarthy	P 79
Paper no 22	Numerical Simulation Of Atomization By The Oblique Collision Of Two Newtonian Jets Navaneeth Krishnan, P S Rajagopal, Madhukar .M. Rao and Akshai K Runchal	P 80
Paper no 149	Numerical Simulation of Impinging Jets with Newtonian and Non-Newtonian Fluid using 3-D unsteady Finite Element VOF solver Atish Ezhuthasan, Jayakumar J S, T Jayachandran and Ajith M	P 81
Air-Assisted Atomization, Supersonic flows		
	8th November, 2016 (Tuesday)	
Paper no 58	A Spray Structure Model of Liquid Jet being injected into Supersonic Gaseous Cross-flow Qinglian Li, Liyin Wu, Chun Li and Yeming Zeng	P 84
Paper no 31	Investigations on Droplet Sizes of Liquid Jet Atomization in Supersonic Crossflow Yeming Zeng, Qinglian Li , Liyin Wu and Chun Li	P 85
Paper no 10	Experimental study on cavity influence on liquid jet penetration in supersonic crossflow Chenyang Li, Qinglian Li, Chun Li and Liyin Wu	P 86
Paper no 77	Investigation of forward injection of impinging liquid jets in supersonic cross flow Sathiyamoorthy K, Venkat S. Iyengar, Panchabudhe Lakhan M, Muruganandam T. M and Satyanarayanan R. Chakravarthy	P 87
Engine Sprays - III		
	8th November, 2016 (Tuesday)	
Paper no 132	Analysis in Unsteady Evaporating Gasoline Spray with LIEF Method Masahiro Matsuura, Vikas Semwal, Shingo Murakami, Eriko Matsumura and Jiro Senda	P 90
Paper no 134	Verification of Breakup Model for Direct Injection Gasoline Spray Tomoyuki Mukayama, Hirotaka Tabata, Eriko Matsumura and Jiro Senda	P 91
Paper no 140	Spark-Ignited Combustion Characteristics of Gasoline Spray Injected by Hole-Type Nozzle for Direct Injection Engine Keiichi Katayama, Kishorkumar Gnanasekaran, Hongliang Luo, Keiya Nishida and Yoichi Ogata	P 92
Paper no 143	Air Entrainment of High Pressure Methane Gas and Effects of Nozzle Diameter and Injection Pressure Hiroshi Tashima, Daisuke Tsuru and Yusuke Shintani	P 93
Paper no 146	Schlieren, Shadowgraph, Mie-scattering visualization of diesel and gasoline sprays in high pressure/high temperature chamber under GDCI engine low load condition Donghoon Kim, Sungsan Park and Choongsik Bae	P 94
Swirl Spray		
	8th November, 2016 (Tuesday)	
Paper no 122	Experimental study on the effect of swirl chamber geometry on the dynamic characteristics of closed-type swirl injector Yunjae Chung, Seok-Kyu Jeong and Youngbin Yoon	P 96

Paper no 95	Breakup dynamics of liquid sheet in coaxial swirling gas flow field Kuppuraj Rajamanickam, Vishnu Goodwill and Saptarshi Basu	P 97
Paper no 70	A study on the flow characteristics of bi-swirl injectors for a thrust chamber Wonjae Yoon, Bom Lee and Kyubok Ahn	P 98
Paper no 21	Spray Dynamics in a Swirling Jet Airblast Atomizer Abhijit Kushari, Srihari Dinesh Kumar Juvva and S Mariappan	P 99

Image Processing

8th November, 2016 (Tuesday)

Paper no 78	Prediction of windward trajectory of liquid jet in cross flow using image processing technique G Muthu Selvan, Muralidhara H S, Suriyanarayanan P, Venkatakrishnan L, Jeyaseelan A R and Rajeshwari N	P 102
Paper no 4	Development Of Algorithm For Determining Particle Size In PDIA Tryambak Gangopadhyay, R. V. Ravikrishna and Achintya Mukhopadhyay	P 103
Paper no 84	POD and DMD of circular liquid sheet originated from a simplified pendent type fire sprinkler Chetankumar S. Vegad, Ramgopal Sampath, S. R. Chakravarthy and Amit Kumar	P 104
Paper no 93	Design, Development and Experimental Validation of an Open Source Program for Particle/Droplet Image Analysis Aravindh Babu and Anand T N C	P 105

Numerical Sprays - II

8th November, 2016 (Tuesday)

Paper no 97	LES of particle laden flows Sri Vallabha Deevi and Joseph Mathew	P 108
Paper no 114	Computational study of liquid jet breakup in swirling cross flow Surya Prakash Ramesh, Suhas Jain Suresh, Gaurav Tomar, Ravikrishna R V and Raghunandan B N	P 109
Paper no 62	A glance at Omega-Y and VOF-DPM hybrid spray models using studies to demonstrate their industrial Applicability Vivek Kumar, Markus Braun and Jochen Schuetze	P 110
Paper no 147	Simulating a mechanical fuel injection system to improve its capabilities Balasubramanian N., Vivekkumar U. and Jayabalan S.	P 111
Paper no 64	Numerical Simulation of Transport of Polydisperse Polykinetic Spray in Spatio-temporally Fluctuating Crossflow using Quadrature Method of Moments Sudeepta Mondal, Aravindhan Ramesh and Achintya Mukhopadhyay	P112
Paper no 150	A Rudimentary Model for Breakup Length in Effervescent Sprays Hrishikesh Gadgil and Raghunandan B.N.	P113

Misc-I

8th November, 2016 (Tuesday)

Paper no 57	Microscopic Observation of Fuel Spray Injected into Cross-Flow and Impinged on Flat Wall Min Guo, Zhanbo Si, Nagisa Shimasaki, Keiya Nishida, Youichi Ogata, Julien Manin and Chenglong Tang	P 116
Paper no 125	Experimental investigation on flow characteristics of diesel spray impinging on flat wall Yoshio Zama, Yotstugu Odawara and Tomohiko Furuhashi	P 117
Paper no 68	Intensification of hydrogen production from methanol partial oxidation using sprays Wei-Hsin Chen	P 118
Paper no 68	Production of Medical Microspheres with Hybrid Atomization Processes for Embolization Applications Po-Yen Chen, Pei-Yu Ho and Muh-Rong Wang	P 119
Paper no 76	A study on atomization process for carbon black reactor Madhavi Gupta, Raj Gopal Tripathi, Manoj Kandakure and Ranjan Ghosal	P 120
Paper no 23	Jet characteristics of high pressure gas under argon atmosphere Yinchun Gong, Xuehai Gong, Jun Deng and Liguang Li	P 121

Misc-II			8 th November, 2016 (Tuesday)
Paper no 42	Suppression of Obstructed Fire using Water Mist Generated by a Twin-Fluid Nozzle Abhijit Kushari and Suresh Lal		P 124
Paper no 5	Effect of Side-Tube on the Agglomeration of Carbon Dioxide Snow Tien-Chu Lin, Shih-Hua Liu, Yu-Hao Li and Mu-Rong Wang		P 125
Paper no 129	Characteristics of zirconia-ethanol suspension spray with thin coaxial twin-fluid atomizer for suspension plasma spraying Norihiro Iki, Masato Suzuki, Mohammed Shahien and Takashi Okada		P 126
Paper no 154	Effect of Acoustic excitation on the Spray Characteristics of a Hollow Cone Spray Aravind I B and S R Chakravarthy		P 127
Paper no 135	Electrostatic Application to Ethanol Based Fuel Injected through Multiple Nozzles Juan Carlos González Palencia, Yasutaka Watanabe, Akito Onozato, Mikiya Araki and Seiichi Shiga		P 128
Paper no 60	Experimental Investigation of Melt Delivery Tube Position on Liquid Metal Atomization Prashanth Satchidanandam, Supriya Sarkar, Anand T N C, Sabin Lal and Shamit Bakshi		P 129
Keynote Lecture III			9 th November, 2016 (Wednesday)
	Simulating Atomization in Complex Injector Geometries Prof. Dr. Marcus Herrmann		P 135
Spray Characterization – III			9 th November, 2016 (Wednesday)
Paper no 112	Experimental Studies on Evaporating Sprays of Diesel & Jatropa Methyl Ester (JME) Prasad Boggavarapu and Ravikrishna R.V.		P 138
Paper no 138	Experimental analysis on the swirl spray formation under low injection pressure Shogo Sakatani, Akihiro Niwa, Jun Kaniyu and Eriko Matsumura		P 139
Paper no 148	Spray Characteristics Of High Rotating Speed Slinger Of Verti-Cal Distribution Mohammad Aqib Khan, Yu-Ting Chen and Wei-Shiang Lai		P 140
Paper no 74	Investigation of High Pressure Diesel Spray Characteristics with Computational Fluid Dynamics Approach. D. Siva Krishna Reddy, Shamit Bakshi and Anand T.N.C.		P 141
Paper no 89	Effect of liquid properties on the oscillation of elliptical liquid jets K.R. Rajesh, R. Sakthikumar, M.V. Narasimha Prasad and D. Sivakumar		P 142
Paper no 99	Identifying Non-Markov effects in sprays Bhaumik Shah, Sri Vallabha Deevi, Arun Tangirala and Mahesh Panchagnula		P 143
Atomization and Breakup			9 th November, 2016 (Wednesday)
Paper no 121	A Consideration on Cavitation and Flash Boiling Atomization Masataka Arai, Jiro Senda and Min Xu		P 146
Paper no 128	Atomization Characteristics of Rotary Bell Cup Atomizer with Shaping Air Flow Masatoshi Daikoku, Souta Nyui, Tatsuya Soma, Yasuhiro Saito, Yohsuke Matsushita, Hideyuki Aoki, Minori Shirota, Takao Inamura, Shin Ogasawara and Junichi Fukuno		P 147
Paper no 52	Breakup of a Liquid Jet under Rigid-Body Rotation Hsin-Wei Huang, Chun-Chia Chang, Chun-Kuei Chen and Ta-Hui Lin		P 148

Paper no 13	Numerical Investigation of Molten Fuel Jet Breakup inside a Liquid Pool Tuhin Maitra, Priyanka Datta, Aranyak Chakravarty, Achintya mukhopadhyay, Koushik Ghosh and Swarnendu Sen	P 149
Paper no 117	Effect of Density Ratio on the Secondary Breakup of Spherical Drops in a Gas Flow Suhas Jain S, Neha Tyagi, Gaurav Tomar, Surya Prakash R and R V Ravikrishna	P 150
Paper no 92	Studies on Secondary Atomization of Molten Gallium Droplets at Nearly Critical Weber Numbers Sumit Joshi and Anand T N C	P 151

Misc-III

9th November, 2016 (Wednesday)

Paper no 2	Numerical Analysis of Pre- Detonation chamber properties with varying mass flow rates using hydrogen/air mixture and its effects in Pulse Detonation Engine. Dheeraj Kumar Singh C T and Gurunadh Velidi	P 154
Paper no 49	Influence Of Swirl Ratio On Fuel Distribution And Cyclic Variation Under Flash Boiling Conditions Jie Yang, Min Xu, David L.S. Hung, Qiang Wu and Xue Dong	P 155
Paper no 104	Relating In-Cylinder Crank-Angle-Resolved Flame Temperature Measurement to Air-Fuel Ratio in a PFI Spark-Ignition Engine using Two-Colour Method V Balaji, R Pradheep, Mayank Mittal, Pramod S Mehta, S R Chakravarthy and V Lakshminarasimhan	P 156
Paper no 85	Imaging of Fuel Distribution in a PFI Spark-Ignition Engine using Planar Laser-Induced Fluorescence R Pradheep, V R Rajagopalan, V Balaji, Pramod S Mehta, Mayank Mittal and S R Chakravarthy	P 157
Paper no 17	Numerical Simulation of Aerosol Transport and Dynamics in a Mixing Chamber Janki Shinde, Pranav Dave, P. S. Rajagopal, Manish Joshi, B K Sapra, S Anand, Madhukar .M. Rao and Akshai K Runchal	P 158
Paper no 3	Experimental investigation of the Atomization Behavior of Ethanol and Kerosene in Acous-tic Fields Dehao Ju, Xiaoxu Jia, Zhong Huang, Xinqi Qiao, Jin Xiao and Zhen Huang	P 159

Monday

7th November 2016

Plenary Lecture 1

Modeling of Dilute Spray Combustion: Interaction of Evaporation, Turbulence and Chemical Reactions

Eva Gutheil

Interdisciplinary Center for Scientific Computing

Heidelberg University

69120 Heidelberg, Germany

gutheil@iwr.uni-heidelberg.de

Abstract

The presentation concerns an overview of models for dilute spray combustion with special emphasis on the interaction of the spray evaporation, the turbulent mixing, and the chemical reactions. In particular, transported joint pdf (probability density function) models are presented where different gas-phase variables act as dependents of the pdf. The solution of the pdf transport equation requires the closure of molecular mixing which is achieved through stochastic difference equations where different mixing models are investigated that are extended to account for spray evaporation. The marginal distribution of the transported joint pdf then are used to identify a generalized presumed β probability density function to account for spray evaporation. Moreover, models to account for incorporating detailed chemical reactions that are important to predict pollutant emission are addressed. A new spray flamelet equation is presented that accounts for the spray evaporation in dilute spray flames. Different regimes in the spray flame are identified where evaporation may determine the spray flame structure and non-premixed or partially premixed regions may exist. Numerical simulations of ethanol/air spray jet flames are presented and compared with experimental data from A. Masri, Sydney, Australia to evaluate the performance of the various models involved. Available experimental data comprise droplet size and velocity distributions in both axial and radial directions as well as gas temperature. The simulations include profiles of gas-phase species such as formaldehyde and carbon monoxide that are major pollutants as well as acetylene that is a precursor of soot formation.

Spray Characterization - 1

Quantitative Analysis of Various Empirical Equations for Fuel Sprays

Peng Gu¹, Min Xu^{1*}, David L. S. Hung^{1,2}, Shengqi Wu¹, Xue Dong¹

1. School of Mechanical Engineering, Shanghai Jiao Tong University, National Engineering Laboratory for Automotive Electronic Control Technology, Shanghai, China
2. University of Michigan-Shanghai Jiao Tong University Joint Institute, Shanghai Jiao Tong University, Shanghai, China

Abstract

Fuel sprays in direct injection engines have been studied extensively through both experimental and numerical methods. A large amount of previous work was focused on acquiring the empirical equations for fuel spray structures based on experimental results. However, one important limitation of the existing empirical equations is that most of them are only applicable to a small range of operating conditions, and they are based on specific assumptions. Furthermore, numerical results from certain empirical equations do not agree with each other. In this study, various empirical equations obtained from existing literatures were quantitatively analyzed and compared based on the experimental results. In addition, limitation of each equation was assessed from the comparison of predictions at specific operating points. Though the analysis under diesel and gasoline engine operating conditions, useful advice in selecting rational equations in practice is provided.

Keywords: Fuel spray, empirical equations, quantitative analysis

*Corresponding author: mxu@sjtu.edu.cn

Spray characteristics in IC engines fuelled with straight vegetable oil

Soo-Young No
Department of Biosystems Engineering
Chungbuk National University
Cheongju, 361-763, Korea

Abstract

In the application of straight vegetable oil (SVO) produced from the triglycerides based biomass to internal combustion (IC) engine, spray characteristics of SVO is the one of important characteristics of engines. Therefore, this review will be concentrated on the spray characteristics of SVO obtained from the triglycerides based biomass to IC engines, particularly compression ignition (CI) engines. SVO used as diesel substitute can be classified into two groups: edible and inedible vegetable oils. Among the edible SVOs, the spray characteristics in IC engines fueled with rapeseed oil and sunflower oil were mainly studied. Even though the studies on the application of rapeseed SVO are mainly concentrated on diesel engine, sunflower SVO was studied in air-assisted pressure –swirl atomizer for micro-gas turbine application and in hollow cone pressure atomizer or twin-fluid atomizer for furnace application. The spray characteristics of the inedible SVOs obtained from *Jatropha* oil and *Karanja* oil were investigated. Of the three spray characteristics; i.e. macroscopic, microscopic and atomization characteristics, spray angle and spray penetration included in the macroscopic spray characteristics were mainly investigated. In the microscopic characteristics, Sauter mean diameter (SMD) was discussed. The study of SMD variation in CI engines fueled with rapeseed SVO is required. Generally, higher values for spray penetration, SMD and lower values for spray angle than those of petrodiesel were obtained. The many different definitions of spray angle and spray penetration were used in the studies. The validation of the newly proposed correlation for the prediction of spray penetration for *Karanja* SVO is required. The viscosity of *Karanja* SVO can be reduced to the level of diesel either by heating the SVO beyond 120 °C or blending less than 20% of the SVO with diesel.

Keywords: straight vegetable oil (SVO), compression ignition (CI) engine, spray angle, spray penetration, SMD

An approach for including evaporation in a model for predicting spray penetration

Gautham Krishnan^{*}, Narendiran C G^{*} and Shamit Bakshi
Department of Mechanical Engineering,
Indian Institute of Technology, Madras.
Chennai - 600036, India.

Abstract

Fuel sprays have been a key topic of research in the field of IC engines, as it influences the efficiency and emission and hence the performance of the engines. But it is fairly tasking to deduce an exhaustively descriptive model of spray mechanics, because of the complexity of the physics involved and the number of parameters that affect the spray, which includes unsteady in-chamber conditions, phase change in the spray and the internal flow in the injector. Nevertheless many models with modest assumptions have been proposed, which reasonably capture the physics of fuel spray. The approach of fuel being considered as *isolated droplets* travelling in the spray and subsequent *evaporation effects* have been studied by a few. It has been thought that the effect of evaporation is negligible in many cases. In this work, we shall examine the mechanics using a fundamental approach and observe how droplet evaporation could affect the temporal evolution of high speed fuel spray in a pressurized chamber. An earlier model based on differential control volume analysis for the mass and momentum conservation of fuel and entrained ambient gas, is extended to account for the evaporation of the droplet.

Keywords: evaporative droplet spray, atomization, drop size, penetration length.

^{*}Corresponding author: gauthamkrishnan1996@gmail.com

^{*}Corresponding author: cgnarendiran@gmail.com

Characterization of Sprays Discharging from Gas-Centered Swirl Coaxial (GCSC) Atomizer Using PDI

Athul Joseph^{*1}, R. Sakthikumar², and D. Sivakumar¹

¹ Department of Aerospace Engineering, Indian Institute of Science, Bangalore 560012, India

² Interdisciplinary Centre for Energy Research, Indian Institute of Science, Bangalore 560012, India

Abstract

This work presents experimental results on the characteristics of sprays discharging from gas-centered swirl coaxial (GCSC) atomizer employed in liquid propellant rocket engines operating under a staged combustion cycle. The atomizer discharges a central gas (oxidizer) jet surrounded by an annular swirling liquid (fuel) stream. The experiments are conducted in a spray test facility with air and water as experimental fluids. The morphology of spray from the atomizer is analyzed by capturing images of spray with the aid of an image acquisition system comprising a digital camera and a lighting device. The measurements of spray droplet characteristics are obtained using phase Doppler interferometry (PDI). The measurements include mean drop size (arithmetic mean diameter and Sauter mean diameter), mean drop velocity, and liquid volume flux. A systematic spatial mapping of spray droplet measurements (11 radial locations $\times$ 5 axial locations) is obtained to describe the evolution of spray droplet characteristics both along the radial and axial directions. For a given flow condition, characterized in terms of gas-to-liquid momentum flux ratio (J), the smaller sized droplets are concentrated near to the spray axis whereas the bigger droplets to the periphery of spray. Evidences on the role of secondary atomization in GCSC atomizer spray are presented

Keywords: Gas centered swirl coaxial atomizer, drop characteristics, spray measurements, phase Doppler interferometry

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

Liquid Phase Penetration of n-Hexane Sprays at different Vapour Concentration of the Ambient

Manas Kumar Pal*^{1,2}, Shamit Bakshi^{1,2}

¹Internal Combustion Engines Laboratory, Department of Mechanical Engineering, IIT Madras.

²National Centre for Combustion Research and Development, IIT Madras.
Chennai 600036, India

Abstract

Evaporation of liquids is influenced by the temperature, pressure, density and transport properties of the surrounding gas. Presence of liquid vapour in the surrounding environment also influences the evaporation processes due to change in saturation level in the surrounding medium. As sprays consists of droplets, evaporation of the sprays is likely to get influenced by the vapour concentration of liquids in the ambient. In the present work, liquid phase penetration of an evaporating single component fuel (n-hexane) spray is studied at different vapour concentration of the ambient in a high pressure –high temperature chamber with optical access. The experiments have revealed that the ambient vapour concentration has an effect on the liquid length of an evaporating spray. Liquid length increases with increase in mass fraction of vapour in the surrounding gas.

Keywords: Spray, evaporation, liquid length, vapour concentration

*Corresponding author: manaspal2015iitm@gmail.com

Spray Characterization and Cold start study of heated fuel using a Multi-hole port fuel heated tip injector

S. Chandrasekar^{1,2*}, T. N. C. Anand¹, and A. Ramesh¹

¹ Department of Mechanical Engineering and National Centre for Combustion Research and Development

Indian Institute of Technology Madras
Chennai 600036, India

² Hyundai Motor India Ltd, Chennai, India

Abstract

Improvement of atomization is one of the effective strategies to improve cold-startability in internal combustion engines. One of the methods to improve fuel spray atomization is by heating the fuel. In this work, the effect of heating commercial gasoline fuel on atomization and cold startability was studied using a four hole port fuel injector. Fuel spray characteristics of unheated fuel at 30° C and heated fuel at 60° C were studied using optical methods. The fuel was electrically heated using a heater built internally within the injector. Liquid droplet surface area per unit volume in a radial plane at various distances from the injector tip was obtained at a frequency of 10 kHz using a planar line of sight extinction tomography based six axes optical patternator. The Influence of fuel pressure and fuel temperature on penetration length and included angle are also reported using back light high-speed imaging technique. Cold start tests were carried out at unheated and heated fuel conditions in a single cylinder, air cooled 200 cc engine. From the results it has been observed that heated fuel surface area is significantly higher compared to unheated fuel at the same injection pressure and at the same injection duration. No significant difference between heated and unheated spray penetration length and included angle was observed. With heated fuel injection, the engine started 4 cycles earlier than for the unheated fuel case and the peak unburnt hydrocarbon emission for a 100 cycles engine run was reduced by 22 %.

Keywords: Optical patternator, Tomography, Heated-tip injector, High-speed imaging

* Corresponding author: sundarachandrasekar@gmail.com

Air-Assisted Atomization

Experimental Study of Liquid Jet Breakup in Coaxial Air Blast Atomization

Abhijeet Kumar, Srikrishna Sahu^{*}
Department of Mechanical Engineering
Indian Institute of Technology
Chennai600036, India

Abstract

Optical connectivity technique is used for measurement of primary breakup length of a water jet in a coaxial air-blast atomizer. The application of the non-intrusive technique is based on imaging the fluorescent light from the liquid (doped with a dye) in the continuous core of the jet when illuminated with a laser beam. The paper focuses on measurement of both mean and fluctuations of liquid break-up length, and also, the effect of air swirl on jet break-up. The experiments were conducted for a wide range of momentum flux ratio. In agreement with some of the previous literature on jet break-up, the momentum flux ratio was found to be the single parameter describing the break-up length. Though, the fluctuations of jet break up were low in comparison to the mean value, it may contribute to downstream spray unsteadiness. As a consequence of addition of air swirl, the mean break-up length was found to reduce in comparison to the case of no air swirl, while the fluctuations were higher. The proper orthogonal decomposition (POD) analysis of the OC images was performed in order to capture the unsteady dynamics of the jet break-up process. The 2nd POD mode was found to correspond to the jet flapping. The corresponding eigen contribution was higher with addition of the air swirl.

Keywords: Optical connectivity, atomization, measurement, breakup length, swirl

^{*}Corresponding author: ssahu@iitm.ac.in

Breakup of Conical Liquid Film with and without Coaxial Gas Flow

Zhongtao Kang^{1,2}, Zhen-guo Wang^{1,2*}, Qinglian Li^{1,2} and Peng Cheng^{1,2}

1. Science and Technology on Scramjet Laboratory
National University of Defense Technology
Changsha 410073, China
2. College of Aerospace and Engineering
National University of Defense Technology
Changsha 410073, China

Abstract

To understand breakup of conical liquid film with and without coaxial gas flow, a back-lighting photography technique has been employed to capture the instantaneous conical liquid film with a high speed camera. The spray cone angle, breakup length, fractal dimension are calculated by image processing. The wave length and wave frequency are extracted from the proper orthogonal decomposition. Breakup of conical liquid film with and without coaxial gas flow was analyzed and discussed. The results show that the conical liquid film have five spray patterns and breakup regimes with the increase of liquid mass flow rate: onion shape, tulip shape, fully developed wavy cone shape, perforated cone shape and turbulent cone shape. The breakup regimes of these sprays include “impact wave” induced (Onion), surface wave induced (tulip and fully developed wavy cone), perforation induced (perforated cone) and turbulence induced (turbulent cone). The spray cone angle increases with the increase of liquid mass flow rate. While the breakup length initially increases and then decreases with the increase of liquid mass flow rate when the Reynolds number is large enough. This phenomenon is caused by the competition of the increase of liquid velocity and the promotion of breakup. The atomization of conical liquid film is fractal process, and the conical liquid film is self-similarity. The fractal dimension increases when the spray pattern varies from Onion shape to fully developed wavy cone shape, and suddenly decreases when the spray develops from fully developed wavy cone shape to perforated cone shape. The reason is that the perforation significantly increases the degree of fragmentation, and the larger the degree of fragmentation, the smaller the fractal dimension. Wave frequency increases when the spray pattern varies from tulip to fully developed wavy cone, perforated cone and finally turbulent cone with the increase of liquid mass flow rate, while the power spectral density decreases with this variation. What's more, the wave frequency bandwidth increases with the variation of the spray pattern. The coaxial gas flow decreases the spray cone angle and breakup length, and change the surface instability from Rayleigh-Taylor instability to Kelvin-Helmholtz instability. The frequency of the Rayleigh-Taylor instability is about 1400 Hz, and that of the Kelvin-Helmholtz instability is about 2000 Hz. The power spectral density of Kelvin-Helmholtz instability is similar with that of Rayleigh-Taylor instability, while the wave frequency bandwidth of Kelvin-Helmholtz instability is smaller than that of Rayleigh-Taylor instability.

Keywords: Conical liquid film, breakup, spray pattern, spray cone angle, POD, surface wave

*Corresponding author: wangzhenguo_wzg@163.com

Supported by the Chinese Program for New Century Excellent Talents in University (Grant No.NCET-13-0156), the National Natural Science Foundation of China (No. 11472303 and 11402298), the National Basic Research Program of China (Grant No. 613239).

Role of the precessing vortex core on wall filming and ligament breakup inside a prefilming airblast atomizer

K. P. Shanmugadas*, S. R. Chakravarthy
National Centre for Combustion Research and Development
Department of Aerospace Engineering
Indian Institute of Technology Madras
Chennai 600036, India

Abstract

The present paper investigates the role of precessing vortex core (PVC) of the swirling air flow field on the wall filming and atomization process inside a simplified typical gas turbine fuel injector. For this, a coaxial counter-swirl prefilming airblast injector with necessary optical access is developed, and experiments are conducted using different laser diagnostic techniques. The internal flow field is captured using time-resolved laser induced fluorescence (LIF) imaging and droplet size distribution is obtained from phase Doppler interferometer (PDI) measurements. The air PVC induces a precessing motion to the pilot nozzle spray, which in turn causes a periodically non-uniform wall filming on the prefilmer surface. Accordingly, the smaller droplets are sheared off periodically from the film at the prefilmer tip, apart from the bigger ligaments produced from an accumulating rim of liquid. Spectral analysis shows that the spray precession frequency matches with the droplet impingement frequency and the shedding frequency of the finer droplets, while the ligament shedding occurs at lower frequencies due to the liquid accumulation. However, the PVC has little effect on the ligament breakup process and on the final injector spray in case of the present geometry and flow conditions.

Keywords: Precessing vortex core, Prefilming airblast atomizer, LIF, PDA

*Corresponding author: shanmugadaskp8@gmail.com

Atomization studies on coaxial circular injector and coaxial profiled injector

Shukla Anjaneesh^{1*}, Nair Rajakumaran², M. Kanthasamy³, G. Remesh⁴,
N. Jayan⁵, M. S. Suresh⁶, V. Narayanan⁷

¹⁻⁷Liquid Propulsion Systems Centre (LPSC), ISRO,
Thiruvananthapuram, Kerala, India-695547

Abstract

The performance of Liquid Rocket Engines (LRE) combustion chamber is characterized by efficient atomization, mixing and combustion of propellants. The mixing of propellants is influenced mainly by the injector type and its configuration. Coaxial shear type injectors are used in many high thrust cryogenic engines worldwide. They have the advantage of uniform distribution of propellants and high throttling ability. Coaxial shear type circular injector element has the advantage of simplicity in geometry and construction; however the fineness level of droplets is poor. This is due to limited shearing area available for the propellants. This can be enhanced by use of profiled exit section in injector element.

Atomization studies on a coaxial circular injector element and coaxial 4-lobed profiled injector element are carried out. Flow tests are done in the experimental facilities at LPSC, wherein the studies are carried out with gaseous nitrogen and water. Tests are conducted for different mass flux ratio and its effects on spray angle, velocity distribution and droplet distribution are investigated. Flow visualization is done with Malvern particle sizer equipment. The cold flow test results indicate that the profiled injector configuration gives shorter breakup length and finer atomization compared to the conventional coaxial circular injector configuration.

Keywords: Injector, Profile, Spray, atomization, drop size

*Corresponding author: anjaneeshactive@gmail.com

An Experimental Study of Spray Characterization of Various Air Flow Configurations in a Hybrid Atomizer

Amlan Garai^{*}, Shinjan Ghosh, Swarnendu Sen, Achintya Mukhopadhyay
Department of Mechanical Engineering
Jadavpur University
Kolkata 700032, India

Abstract

A hybrid atomizer is a modification of an air blast atomizer. Air blast atomizers employ a fuel stream and an outer high velocity air stream to produce effective atomization. However in a hybrid nozzle, an internal air stream is also incorporated to synergize the effect of the outer air stream. In the current study, an annular fuel sheet is sandwiched by two sheets of air in the atomizer. A cylindrical sheet of air from the inside, and an annular air sheet externally. Various configurations have been employed to study the resultant spray characteristics using kerosene as the fuel. The fuel sheet is fed tangentially to produce a swirling effect and the air sheets are given a swirling motion in either directions by using tangential inlets. The inner air sheet can be fed axially as well. These arrangements have been combined to produce the different configurations. The resulting spray characteristics such as breakup length, spray cone angle and sheet-width are being measured with a variation in the flow rates of air and fuel sheets using all the above mentioned configurations. A high speed camera is being used to capture the sprays at a number of frames for image processing.

Keywords: Spray, breakup length, cone angle, sheet width

^{*}Corresponding author: amlangarai@gmail.com

Measurement of Primary Break-up of Liquid Jet in a Cross-stream Airblast Atomizer

Shirin Patil, Srikrishna Sahu*
Department of Mechanical Engineering
IIT Madras
Chennai 600036, India

Abstract

Due to their significance in gas turbine engine combustors, cross-stream injection of fuel jets into incoming air flow has gained attention since last few decades. Especially, primary breakup of the cross-stream jet is of prime importance for optimal engine performance, reduction of combustion instability and emission. While most of the literatures on this subject are confined to wind-tunnels, available data for atomizers are limited. The current research work focuses on measurement of primary breakup of liquid jets in an airblast atomizer, which allows radial injection of a jet of water in to axial flow of air at much higher velocity. The Optical Connectivity (OC) technique is employed to visualize and measure instantaneous primary break up length of the liquid jet. The images of the liquid jet are captured for side view as well as front view. Experiments were performed for a range of Weber number (We), Momentum Flux Ratio (MFR) and liquid Reynolds number (Re). The variation of mean horizontal and vertical breakup lengths with respect to We and MFR for the same Re are studied. For large Re the breakup length was found to be insensitive to MFR and We . The fluctuations of breakup lengths were quantified and found to be significant for high We flows. Experiments were also conducted in presence of swirling air flow, which could deflect the jet in the direction of swirl and also twist it such that the jet breakup length was smaller in comparison to no swirl case.

Keywords: Breakup length, Swirling air, Optical connectivity, Momentum flux ratio, Weber number

*Corresponding author: ssahu@iitm.ac.in

UWS Sprays

Droplet Evaporation and Spray Formation of Urea Solution in High Temperature Atmosphere

Tomohiko Furuhashi*, Siti Nor Ain Musa, Yoshio Zama
Division of Mechanical Science and Technology
Faculty of Science and Technology, Gunma University
1-5-1 Tenjin-cho, Kiryu, Gunma 376-8515, Japan
Masataka Arai
Department of Mechanical Engineering
Tokyo Denki University
5 Senju Asahi-cho, Adachi-ku, Tokyo 120-8551 Japan

Abstract

Selective Catalytic Reduction using urea as reductant is the aftertreatment technology being adopted by the majority of diesel bus and truck manufacturers for meeting NO_x emission regulations. This technology is known as urea-SCR system. Urea-SCR system works by introducing urea solution into the automobile exhaust to convert the NO_x to harmless nitrogen and water, over an SCR catalyst. In this study, the evaporation behavior of a single urea solution droplet and urea solution spray was investigated. In order to study the evaporation behavior of a single urea solution droplet, two methods were used where first, droplet was impinged onto a heated surface, and second, droplet was suspended in an electric furnace. In experiment using the heated surface, evaporation behavior of urea solution was classified into two patterns when temperature exceeded the maximum evaporation point. These patterns were short lifetime evaporation and long lifetime evaporation. In the experiment using the electric furnace where droplet was suspended at the end of a quartz fiber, evaporation behaviors were divided into three stages. Urea solution droplet showed similar behavior as water only at the first stage, where the square of droplet diameter decreased linearly with time, following the *d-square law* (*d² law*). In the second stage, formation of bubbles and foams, and expansion of droplet size were observed. The last stage was crystallization stage. The differences between water and urea droplet evaporation characteristics were assumed to be caused by the thermal decomposition of the urea. Spray formation of urea solution was studied by injecting urea solution into a hot air stream inside a wind duct which acted as a substitute apparatus for the real engine exhaust pipe. When the hot air temperature T_a was 200°C, crystallization of urea spray into white substance occurred faster than at $T_a = 500^\circ\text{C}$.

Keywords: Urea-SCR, droplet evaporation, spray behavior, decomposition, crystallization, deposit

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

Numerical Modeling of adBlue Droplet Evaporation in the Context of SCR-DeNO_x

K. P. Nishad * , A. Sadiki, and J. Janicka

Institute of Energy and Power-plant Technology, Department of Mechanical Engineering
Darmstadt University of Technology
Darmstadt, 64287, Germany

Abstract

Apart from improving the in-cylinder design technology and favourable engine combustion, the injection of the urea-water solution (UWS) commercially known as adBlue inside the exhaust port is one of the widely adopted technologies for NO_x reduction in diesel engine to meet environment regulations. With the help of available heat from exhaust gas, the injected UWS is transformed into gaseous NH₃ by evaporation of mixture and thermal decomposition of urea. Subsequently, NH₃ reacts with NO_x and converts into harmless compound that can be finally released to the environment. The reaction and conversion efficiency mostly depend upon the evaporation and subsequent mixing of the NH₃ into the exhaust gas, which in turn depends upon the engine loading conditions. Considering the individual and peculiar thermo-physical properties of water and urea (solid state at room temperature), the model development for adBlue evaporation is challenging.

In this paper, a bi-component evaporation model is developed and validated by comparison with available experimental data. Two distinct evaporation stages are observed in both experiments and simulations. The first stage represents the initial evaporation of water due to its low boiling point, while a second slope accounts for the urea thermolysis. A satisfactory agreement is achieved for a wide range of operating temperatures of the adBlue droplets. At low exhaust gas temperature, crystallization or solidification of urea is generally observed due to its higher sublimation temperature. In SCR context urea particle solidification should be avoided as it blocks the SCR circuit, and also the solidified urea takes longer time to convert into gaseous phase making it difficult to optimize the adBlue injection process. To address these aspects requires extra effort which is left for future work.

Keywords: adBlue, evaporation, multi-component.

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

Experimental Studies on an Evaporating Droplet of Urea Water Solution; Effect of Ambient Temperature and Initial Urea Mass Fraction on Evaporation Rate

S. Mikhail*, T. N. C. Anand, and S. Bakshi
Department of Mechanical Engineering
Indian Institute of Technology
Chennai 600036, India

Abstract

Climate change has never before, in the history of mankind, attracted the attention of researchers' world-wide, like it does today. Stringent emission norms and depleting fossil fuel reserves have, in general, led to the development of cleaner technologies and optimization of existing technologies for stationary/mobile power generation. Even though some technologies like selective catalytic reduction (SCR) devices, for reduction of nitrogen oxides have been in existence and widely used in stationary diesel engines for many decades, their adaptation to automobile diesel engines have posed new challenges and this in turn has led to a renewed interest in this area among engineers and scientists alike. In order to optimize the chemical process of reduction of NO_x in an SCR device, attempts have been made to better understand the evaporation process of urea water solution (UWS) at temperatures typical of those in an automobile exhaust gas stream. The current work attempts to further improve our understanding about these processes, by measuring the droplet evaporation rate of a suspended UWS droplet using shadow images. This work also attempts to develop an empirical model for UWS droplet evaporation which is computationally less expensive and easily implementable as a droplet evaporation model for UWS spray simulations.

Keywords: UWS, Single Droplet Evaporation, Micro Fibre Suspended, SCR, Sherwood Number Correlation

* Corresponding author: mikhilsuren@gmail.com

Internal Flows in Nozzles

X-Ray High-Speed Visualization of Cavitation in Fuel Injector and Discharged Liquid Jet

A. Sou*, T. Miwa, R. Prasetya, T. Kasahara,¹
Kobe University, Kobe, Japan

S. Moon,²
Japan, National Institute of Advanced Industrial Science and Technology, Tsukuba, Japan

Y. Wada, and H. Yokohata,³
MAZDA Motor Corporation, Hiroshima, Japan

Abstract

Cavitation in fuel injectors plays an important role in fuel spray characteristics for internal combustion engines. However, our understanding on cavitation in the injectors is not enough. In this study we carried out a phase contrast high-speed imaging of cavitation in some liquid injectors using a synchrotron x-ray in SPring-8 with a short exposure time. As a result, we found that (1) cavitation inception sometimes occurs in the bulk liquid or on the side wall, *i.e.*, heterogeneous nucleation, and cavitation usually consists of non-spherical bubbles, and (2) the sizes of cavitation bubbles in the nozzle are almost proportional to the thickness of the recirculation zone, which is proportional to the nozzle size.

Keywords: Cavitation, fuel injection, high-speed visualization, x-ray, diesel fuel injector

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

The New Cavitation Bubble Model inside the Nozzle in Consideration of the Bubble Dynamics with Bubble Nucleation Model

S. Murakami^{*}, V. Semwal, M. Matsuura, E. Matsumura, J. Senda

Department of Mechanical Engineering

Doshisha University

1-8, Tatara, Miyakodani, Kyotanabe-shi, Kyoto, 610-0394, Japan

Abstract

Spray characteristics are significantly affected by the turbulence of the flow such as cavitation inside the nozzle. The objective of the work reported in this study is to clarify the effect of cavitation phenomenon on the flow inside the nozzle and the spray characteristics. This report deals with the effect of cavitation phenomenon both in experiment and analysis. In the experiment, the purpose is to investigate the effect of cavitation phenomenon on the liquid jet, on cavitation bubble morphology and cavitation behavior. In the analysis, our original cavitation model applied turbulence induced by the energy generated with cavitation bubble shrinkage or collapse was incorporated in commercial code of Star-CCM+ scheme. The effects of cavitation phenomena on the flow conditions and the liquid jet were simulated by this model. To validate this model, this numerical simulation results same as experimental conditions were compared with the experimental results. In addition, it was confirmed that the number of bubble nuclei gave to the flow inside the nozzle by changing the number of bubble nuclei artificially.

Keywords: D.I.S.I, Cavitation, Bubble Dynamics, Nozzle Flow, Liquid Jet

^{*}Corresponding author: dup0544@mail4.doshisha.ac.jp

Effects of Internal Deposits on Sprays of a Gasoline Direct Injector

Haoyi Song¹, Jin Xiao^{1*}, Xianpei Yang¹, Kai Yu¹, Xinqi Qiao¹, Dehao Ju¹, Zhen Huang^{1*}, Qi Yin², Xiaojin Bai², Xuewei Pan²

1. Key Lab. for Power Machinery and Engineering of M. O. E
Shanghai Jiao Tong University
200240, Shanghai, P. R. China.
2. SAIC Motor Cooperation Limited
201804, Shanghai, P. R. China.

Abstract

Internal Deposit formation in the injector nozzle can both restrict the fuel flow and alter the spray characteristics of the injectors. High levels of flow restriction and any level of spray distortion cannot be adequately controlled electronically yet. The decrease of the fuel flow rate results in the decrease of power and the fuel spray pattern distortion impacts combustion quality, which can lead to loss in power and fuel economy, as well as increased exhaust emissions. In the present paper, an experimental facility that simulates the heat environment when a GDI injector is working in the engine is employed to make internal deposits form inside the injector nozzle and to study the effects of injector tip temperature on deposits formation. Besides, an experimentally investigation to the spray behaviors before and after internal deposits forming is reported. Results shows that a proper temperature interval exists, beyond which it's not easy for deposits to form inside the injector. In addition, the positions of deposits accumulating in the injector are not the same under different injector tip temperature and a rule is concluded that the higher that the temperature is, the deeper the positions of deposits are. Moreover, synchrotron X-ray micro-tomography is used to identify the configuration of nozzle, which contributes to understand the experimental results.

Keywords: GDI, Deposits, Injector tip temperature, Spray

*Corresponding author: xiaojin@sjtu.edu.cn

Influence of Pressure Oscillation Induced Klystron Effect on the Inner Flow and Spray Characteristics of Pressure Swirl Injector

Peng Cheng, Zhongtao Kang, Huiyuan Chen, Xinqiao Zhang, Qinglian Li^{*}
Science and Technology on Scramjet Laboratory
National University of Defense Technology
Changsha 410073, China

Abstract

Klystron effect is a possible mechanism of low and medium frequency combustion instability in liquid rocket engines. Therefore, a better understanding of klystron effect is needed. Present work focuses on the influence of klystron effect on the inner flow and spray characteristics of pressure swirl injector. Film thickness near the injector exit, spray angle and size distribution of droplets are analyzed and discussed. The results indicate that the klystron effect of pressure swirl injector occurs when the fast fluid catches up with the slower one injected previously. And the frequency of its occurrence is consistent with that of pressure oscillation. When klystron effect initially occurs in the injector, liquid is accumulated in swirl chamber. Subsequently, the axial velocity of the liquid at the swirl chamber exit increases. After that, the axial-velocity-increased liquid film catches up with the slower one, producing a thicker and faster liquid film. When klystron effect moves outside the injector, the spray angle suddenly decreases. As the faster and thicker film catches up with slow one, the conical film is folded. This folded film further breaks up into ligaments and droplets, producing a spray cluster with the smallest light transmittance. Instead, the SMD (Sauter Mean Diameter) of this spray cluster is the largest, because the folded thick film produces more big drops. Above all, when the pressure oscillates, klystron effect is observed in the inner flow and the breakup process of liquid film periodically. Accompanying with the klystron effect, the film thickness at the injector exit, spray angle, transmittance of the spray, SMD and diameter distribution oscillate with the same frequency. The klystron effect reduces the uniformity of spray and deteriorates the atomization performance.

Keywords: Combustion instability, klystron effect, pressure swirl injector, inner flow, spray characteristics

^{*}Corresponding author: peakdreamer@163.com

Supported by Chinese Program for New Century Excellent Talents in University (Grant No. NCET-13-0156) , the National Natural Science Foundation of China (No. 11472303 and 11402298) and National Basic Research Program of China (Grant No. 613239)

Study of Cavitation and 3D Needle Movement due to Erosion in Fuel Injection Nozzles Using Coupled Simulation Tools

B. Mandumpala Devassy* and W. Edelbauer
AVL LIST GmbH, Graz, Austria

D. Greif
AVL-AST d.o.o., Maribor, Slovenia

Abstract

Internal combustion engines still remain one of the greatest inventions of the modern world. Current developments lean towards lower fuel consumption, more performance at high efficiency and low tail-pipe emissions. Studying and understanding of different involved physical phenomena requires sophisticated tools, regardless if computational or experimental. Present study focuses on the computational study of fuel injection events of a typical piezoelectric common-rail mini-sac Diesel injector. The importance of liquid compressibility, viscous heating effect and cavitation are addressed. Besides, the influence of cavitation erosion on the 3D needle movement (and thereby its effect on the emerging spray) are examined. The adopted methodology involves the coupling of 1D and 3D simulation tools, AVL BOOST HydSim and AVL FIRE™ respectively. The 3D case consists of a 3phase system (liquid fuel, fuel vapor and air) with a 3D computational domain comprising a 6-hole injector attached to a high pressure gas chamber replicating a real Diesel fuel injection system. The results indicate pronounced flow asymmetry due to cavitation erosion which in turn will affect inhomogeneity in spray and mixture preparation inside the combustion chamber. Moreover, such flow asymmetry will also lead to radial needle movement adding further irregularity in the cavitation cloud thereby producing asymmetric erosion on the nozzle-hole surfaces.

Keywords: Atomization, Injector, Compressible Multiphase flow, Cavitation, Spray Formation

*Corresponding author: bejoy.mandumpaladevassy@avl.com

Collision and Coalescence

High fidelity simulations of binary collisions of liquid drops

H. Sarang¹, S. R. Reddy^{2*}, J. Shashank³, and R. Banerjee⁴

Department of Mechanical and Aerospace Engineering
Indian Institute of Technology Hyderabad
Hyderabad 502285, India

Abstract

The phenomena of binary droplet collision is seen in dispersed phase systems such as sprays in internal combustion engines, gas turbines, etc. The present work aims at understanding off-center collision dynamics of two droplets of unequal sizes. High fidelity simulations are performed using three dimensional, two phase- finite volume based solver in OpenFOAM platform. Volume of fluid (VOF) method is used for interface capturing. Navier-Stokes equations are solved using a projection algorithm. Simulations are done with MPI parallelization to meet the computational demand. The solver is validated against droplet splashing benchmark test case. The present numerical results study the effect of impact parameter and diameter ratio on droplet collision dynamics.

Keywords: droplet collision, two-phase flow, Volume of fluid method

*Corresponding author: me10p006@iith.ac.in

Experimental Measurement of Droplet Coalescence in Three-Dimensional Turbulence

M. Shyam Kumar^{*}, S. R. Chakravarthy,

Department of Aerospace Engineering, and National Centre for Combustion Research & Development, Indian Institute of Technology Madras, Chennai, India

Manikandan Mathur

Department of Aerospace Engineering, Indian Institute of Technology Madras, Chennai, India

Abstract

Droplets in a turbulent flow are a common occurrence; and, they undergo coalescence or shattering upon colliding with each other. While binary droplet collision experiments delineate regimes of coalescence and shattering, statistics of such events with droplets in a turbulent flow field are lacking. An active turbulence grid (ATG) is adopted with a spray downstream of it, to focus on the spray-turbulence interaction far downstream where the droplets have attained nearly constant diameter and velocities over large radial distances. The ATG introduces a near-isotropic turbulence, with anisotropy in the 1.1-1.2 range over a wide range of intensities spanning 5-20% to an otherwise near-laminar uniform convective flow with velocities ~ 2 m/s. A rate of coalescence is calculated based on the spatial growth of droplet sizes (measured using PDPA-Phase Doppler Particle Analyzer) and the droplet residence time in the turbulent flow field (measured using LDV- Laser Doppler velocimetry). The coalescence rate is observed to increase with turbulent intensity up to some level, beyond which it mildly decreases with further increase in turbulent intensity. At each turbulence intensity, we identify a cross-over diameter above/below which the number density of droplets of a given size increases/decreases with the residence time in the turbulent flow. We discuss some potential physical mechanisms that may describe the observed coalescence behavior. Further high-speed tomographic visualization experiments are needed to substantiate the coalescence or shattering of droplets as the above data indicates.

Keywords: Spray, Kolmogorov scale, coalescence, turbulence intensity

^{*}Corresponding author: shyamkuttamath@gmail.com

**Droplet coalescence statistics in a low-pressure multi-hole injector
using high-speed imaging**

R. S. Pathania^{*}, S. R. Chakravarthy^{+,#}, P. S. Mehta^{*}

^{*}Department of Mechanical Engineering, ⁺Department of Aerospace Engineering
National Centre for Combustion Research and Development
Indian Institute of Technology Madras, Chennai 600036 India

[#]Corresponding author – email: src@ae.iitm.ac.in

Abstract

This study concerns the measurement of droplet diameter before and after coalescence of droplets during atomization of spray from a low-pressure (2-6 bar) multi-hole injector. The droplet diameter is measured by high-speed imaging along the spray axis at 30, 50, and 70 mm from the injector tip. The configurations studied are a single hole and two adjacent holes of a four-hole injector, where such coalescence was previously suspected to cause growth in droplet size along the spray downstream as measured by PDPA, whereas spray from the two diagonal holes and all four holes did not exhibit this behaviour. Droplet pairs of different size ranges collide to form bigger droplets with different probabilities for the single and two-hole cases at different injection pressures. In both cases, droplets of 130-170 microns size lead to ones of 230-260 microns at low pressures (2-3 bar). At high pressures (4-6 bar), however, droplets >150 microns collide to form those >200 microns, more frequently for the single-hole than the two-hole case, whereas coalescence at smaller sizes, i.e., >80 microns leading to 150-180 microns, occurs more frequently for the two-hole rather than the single-hole case.

Keywords: Droplet coalescence, port fuel injector, high speed imaging, droplet diameter

Experimental investigation of droplet collision and dispersion in interacting sprays

Manish M, Srikrishna Sahu*
Department of Mechanical Engineering,
IIT Madras,
Chennai 600036, India

Abstract

While binary droplet collision has been well studied in past, which constitutes the basis for modelling collision outcomes in current spray computations, experimental report on major effects of droplet collision in sprays is not abundant. The present paper aims to quantify the effect of droplet collisions on spray characteristics and dispersion of droplets in overlapping region of two interacting sprays, whose axes are parallel to each other. Planar measurement of droplet size is achieved by ILIDS (Interferometric Laser Imaging for Droplet Sizing) technique, while droplet velocity is measured by Particle Image Velocimetry (PIV). The droplet number density is measured by identifying individual droplets on PIV image and then droplet counting within an interrogation window. The results show considerable effect of spray interaction on droplet characteristics for the considered flow conditions, for example, droplet velocity increases in the overlapping region, while droplet number density reduces suggesting droplet coalescence. The paper includes a comprehensive report on the effect of droplet collision on droplet size, number density, velocity and liquid volume flux in the spray overlapping region.

Keywords: Spray, droplet collision, coalescence, ILIDS, number density, clusters.

*Corresponding author: ssahu@iitm.ac.in

Engine Sprays - 1

Microscopic Spray Investigation of Karanja Biodiesel and Its Effects on Engine Performance and Emissions

Chetankumar Patel, Nikhil Sharma, Nachiketa Tiwari and Avinash Kumar Agarwal*
Engine Research Laboratory, Department of Mechanical Engineering
Indian Institute of Technology Kanpur, Kanpur-208016, India

Abstract

Diesel engines are widely popular for small to large size vehicles and agricultural/ construction/ power generation machines. Biodiesel from non-edible oils such as Karanja can be used as an alternative fuel without modifications in the engine hardware. Fuel atomization is largely affected by higher viscosity of biodiesel. Microscopic spray investigations are conducted to understand the spray behavior of the biodiesel. This investigation carried out to measure spray droplet's sauter mean diameter (SMD), droplet velocity and its variations with respect to time at lower injection pressure of 200 bar (Typical fuel injection pressure in mechanical fuel injection equipment equipped engines) at atmospheric condition for Karanja biodiesel (KB100) and mineral diesel. Higher SMD and slightly lower velocity were observed for KB100 vis-à-vis baseline mineral diesel. Engine performance and emissions were measured for these test fuels at constant engine speed of 1500 rpm. Higher CO emission and lower HC and NO_x emissions were observed from KB100, primarily due to differences in spray characteristics and fuel properties.

Keywords: Microscopic Spray Characteristics; SMD; Droplet Velocity; Biodiesel; Emissions.

* Corresponding author: akag@iitk.ac.in

Characteristics of High Pressure Ethanol Spray Flames in Diesel Engine-Like Environment using Optical Diagnostics

M.M. Avulapati*, R. Pos, A. Megaritis, J. Xia and L. C. Ganippa
Center for Advanced Powertrain and Fuel Systems
College of Engineering, Design and physical Sciences
Brunel University London
United Kingdom UB8 3PH

Abstract

Ethanol is considered as one of the clean alternative fuels for IC engine applications. Due to its properties, ethanol can be used directly in spark ignition (SI) engines with very little modifications. However, using ethanol to fuel compression ignition (CI) engines has its own challenges including poor ignitability and lubricity. These challenges can be overcome by using additives and by developing new combustion systems by modifying the engine geometry and components. The use of ethanol as an alternative fuel in diesel engines has the potential to reduce particulate emissions significantly. In the present study, spray combustion of pure ethanol from a multi-hole common rail diesel injector was studied using high speed optical diagnostics. Fuel was injected into an optically accessible constant volume chamber maintained at high temperature and high pressure conditions. Mie scattering and natural soot luminosity images were acquired and analysed. Under similar ambient conditions of temperature and pressure, ethanol exhibited higher soot lift-off length, longer ignition delay and lower soot luminosity when compared to diesel. Variations in soot lift-off lengths and ignition delays between sprays from different orifices were attributed to anomalies during initial part of injection.

Keywords: Ethanol spray combustion, lift off length, Spray combustion, Ethanol spray.

*Corresponding author: madan.avulapati@brunel.ac.uk

Spray Behavior of a GDI Injector at Constant Fuel Injection Pressure and Varying Engine Load

Nikhil Sharma, Chetankumar Patel, and Avinash Kumar Agarwal*
Engine Research Laboratory, Department of Mechanical Engineering
Indian Institute of Technology Kanpur, Kanpur-208016, India

Abstract

In order to meet stringent emission norms, it becomes necessary to understand the behavior of atomized fuel spray droplets of a GDI engine. An experimental study was carried out to find droplet size and velocity distributions using phase droplet interferometry (PDI). Experiments were performed in ambient condition using a gasoline direct injection (GDI) injector at a fixed fuel injection pressure (FIP) of 80 bar. Two fuel injection quantities of 12mg and 20mg per injection were compared for spray droplet size and velocity distributions. Shadowgraphy of the spray plume was performed with the help of a high speed CCD camera to determine primary and secondary spray breakups. Experiments were performed 30 mm downstream of the GDI injector nozzle. Maximum spray droplet velocity distribution was found to be almost similar at same FIP, irrespective of fuel injection quantity. The three channels measured the droplet velocity distributions in mutually orthogonal planes. Increasing the fuel injection quantity while keeping constant fuel injection pressure, didn't make noticeable difference in velocity or diameter distributions. It was observed that lower fuel injection quantity exhibited larger droplet surface area in contact with ambient air at constant pressure therefore it led to superior fuel-air mixing.

Keywords: Gasoline direct injection, spray, atomization, droplet size distribution, and droplet velocity distribution.

* Corresponding author: akag@iitk.ac.in

Numerical Sprays 1

Validation of Discrete Spray Models for a Movable Pintle Injector of Liquid Rocket Engines

Kanmaniraja Radhakrishnan, Min Son and Keonwoong Lee,¹
Jaye Koo*²

¹ Graduate School, Department of Aerospace and Mechanical Engineering,
Korea Aerospace University
Goyang 10540, Republic of Korea

² School of Aerospace and Mechanical Engineering
Korea Aerospace University
Goyang 10540, Republic of Korea

Abstract

Pintle injectors are used in bipropellant rocket engines to control the mass flow rate and allow for deep throttling to obtain high combustion efficiency. Movable pintle tip allow a variety of spray characteristics. To study this effect, a numerical study was conducted on a 2D axisymmetric model with two different of pintle-opening distances. Water and air were used as propellants. The liquid sheet was broken by injecting water from a center gap and air from an annular gap of the pintle injector. The discrete spray model was used to simulate the breakup, with secondary breakup being performed using a wave model. To validate the spray models, analyses were conducted with a breakup time constant of $B_1 = 1.73$ with an initial droplet diameter equivalent to the pintle-opening distance in the wave-breakup model. The Sauter mean diameter was the primary performance parameter considered for comparison, and validation was performed throughout experiment. Further, the effects of gas phase on liquid droplet breakup and the secondary atomization of droplets were observed from the spray-particle images. Recirculation in flows was investigated in terms of spray angles and mixing of propellants. Finally, numerical results were obtained using $B_1 = 1.73$ and found to agree with the experimental data concerning SMDs.

Keywords: Pintle injector, Water–air spray, Sauter mean diameter, Simulation, Liquid rocket engine

*Corresponding author: jykoo@kau.ac.kr

Numerical Modeling on Liquid Fuel Injected Spray Break-up and Evaporation with Elevated Pressures in BERL Burner

Repaka Ramesh*, Dr Gurunadh Velidi

*M. Tech in Computational Fluid Dynamics

Assistant Professor, Department of Aerospace Engineering

University of Petroleum and Energy Studies

Dehradun 248007, India

Abstract

The present study demonstrates the n-Heptane liquid fuel injected spray break-up process and evaporation with elevated pressures in BERL Burner by performing computational analysis using Eulerian-Lagrangian multiphase scheme. Spray combustion is a complex process which involves two phases as gaseous phase and liquid phase. Here, the liquid in the form of droplets as discrete/dispersed phase and gas as continuous phase. The computational approaches such as Lagrangian is formulated for discrete phase and Eulerian approach formulated for continuous phase. In the current spray combustion simulations four Cone Nozzle Injectors used from which the liquid fuel n-heptane is injected into high temperature (850K) gaseous environment through $3 \times 10^{-4}m$ cone nozzle diameter. In this study the total six simulations were carried out such as simulations of high pressure fuel injection (10bar, 15bar and 20bar) combined with two different Burner chamber pressures (3bar and 5bar). In this research the numerical investigation for liquid spray break-up process and evaporation is done using the open-source software package OpenFoam.

Models like Break-up, Heat Transfer, turbulence and evaporation models used in the sprayFoam solver which is in-built solver in OpenFoam. Each model has its own sub-model like for evaporation studied with Spalding sub-model, for heat transfer studied with RanzMarshall sub-model and for break-up studied with ReitzDiwakar sub-model.

Keywords: SprayFoam, Evaporation, Break-up, Droplets

*Corresponding author: repakaramesh14@stu.upes.ac.in

Numerical Simulation of Extinction Conditions of Counter-Flow Methane Diffusion Flame with Polydisperse Water Spray

A. Harish, Akaashdeep Singh and Vasudevan Raghavan^{*}

Department of Mechanical Engineering, Indian Institute of Technology Madras, Chennai,
India 600036

Abstract

This study numerically investigates the effect of sauter mean diameter of water droplets on the extinction of counter-flow methane diffusion flame. Fully developed methane and oxidizer streams flow from top and bottom inlets, respectively. Nitrogen is supplied through a co-flow annular path outside both ducts to reduce the interference between the reactants and ambient gas. Poly dispersed water droplets having average sizes (SMD) of $38.5\ \mu\text{m}$ and $60.2\ \mu\text{m}$ are introduced from the lower duct at a steady rate along with the air stream. The mass fraction of water droplets is kept as 0.022. For a given water droplet diameter, oxygen mole fraction in the oxidizer stream is decreased from a value of 0.3 till a point where the flame eventually extinguishes. For a given SMD, the oxygen mole fraction at which the flame extinguishes is found to be in good agreement with the experimental results. As the oxygen concentration is reduced from 0.3, the temperature and extents of the flame zone decrease and finally the flame extinguishes. When temperature and reaction zone extents change, the path of the water droplets also changes. The case with SMD of 60.2 microns interacts directly with the flame in heterogeneous mode. As a result, the flame is quenched at a higher oxygen mole fraction than that in the case of 38.5 microns.

Keywords: counter flow methane diffusion flame, water droplets, oxygen mole fraction

^{*}Corresponding author: aqibkhn86@gmail.com

Numerical Investigation of Extinction Condition of Flat and Curved Counterflow Laminar Diffusion Flame with Polydisperse Water Sprays

Sourav Sarkar^{*}, Achintya Mukhopadhyay, and Swarnendu Sen
Department of Mechanical Engineering,
Jadavpur University,
Kolkata 700032, India

Abstract

A numerical investigation of the influence of polydisperse spray of water droplets and flame curvature on the extinction of counterflow laminar diffusion flames is presented. A practical atomizer generally produces a range of sizes of the droplets. The size distribution of the water droplet plays a crucial role in the effectiveness of the water spray. The common practice of representing the polydisperse spray by some suitable mean diameter (e.g., Sauter mean diameter) may not be effective in the present situation as the dynamics and evaporation characteristics of the individual droplets are different due to their different sizes. The physics of flame extinction by water droplets is complex due to interaction between flame stretch, flame curvature and heat loss due to evaporation. The present numerical study uses the commercial CFD code ANSYS Fluent with Laminar finite rate model and reduced CHEMKIN chemistry for numerical simulations. The discrete phase, namely the water spray, was simulated using Lagrangian DPM approach. In the present simulation the extinction condition was obtained by decreasing the oxygen concentration of the oxidizer stream for different water droplet size distributions. Different numerical models used for simulation have been validated with experimental data. Extinction conditions of both flat and curved flame have been explored. An optimum droplet diameter was obtained for which the effectiveness of the spray was maximum.

Keywords: Counterflow diffusion flame, Water spray, Extinction condition, Polydisperse spray, Discrete phase modeling, CHEMKIN

^{*}Corresponding author: souravsarkar.iitm@gmail.com

Numerical study of the effect of fine polydisperse water spray on laminar flame speed of hydrogen-air premixed flames stabilized in the stagnation flow field

Joydeep Munshi^{1*}, Sourav Sarkar², Achintya Mukhopadhyay², Swarnendu Sen²

¹School of Nuclear Studies and Applications, Jadavpur University, Kolkata

²Department of Mechanical Engineering, Jadavpur University, Kolkata

Abstract

Water spray systems are effective hydrogen hazard mitigation technique during severe accidents in nuclear power plants. Water spray helps in fire suppression in two ways; reducing temperature by absorbing the latent heat of evaporation and diluting the oxygen level due to vapor generated by evaporation. Flame speed reduction by using fine water spray can be a measure of efficacy of water as a fire suppressant. In the present study, effect of fine water droplets on the flame speed of Hydrogen-air premixed flame was numerically investigated using a single jet-plate configuration. The present numerical study uses the commercial CFD code ANSYS Fluent 14.5. The discrete phase, namely the water spray, was simulated using Lagrangian DPM approach. Dependence of Laminar flame speed on strain rates for stoichiometric mixture has been studied without water spray and was compared for varied spray situations. The chemical reaction rates involving different radicals like OH are also observed to be affected when the water spray is present.

Keywords: Laminar flame speed, Stagnation flow, Fire suppression, Water spray, Stretch Rate

*Corresponding author: joydeep3163@gmail.com

Droplet Evaporation, Combustion and Impact

Spreading Dynamics of Shear-Thinning Fluid Drop Impact on Hydrophobic Substrate

Purushothaman Nandagopalan^{*+}, Jerin John⁺, Ankur Miglani⁺ and Seung Wook Baek

Department of Aerospace Engineering,
Korea Advanced Institute of Science and Technology, KAIST,
291 Daehak-ro, Yuseong-gu, Daejeon-34141, Republic of Korea

Abstract

Droplet impact dynamics of the hydrogels on the hydrophobic surfaces have been studied experimentally and compared with water and ethanol as the reference fluids. Hydrogels are prepared by lading water with Polyacrylamide polymer at three loading rates 1, 2, and 3 wt. %. The resultant hydrogel phase behaves as a shear thinning fluid with altered rheophysical properties. The presence of polymer additive modifies the entire spreading dynamics by reducing the maximum spread which occurs due to the presence of non-zero yield stress and also by eliminating the well-known bouncing effect observed in case of water. Despite the effect of inertial spreading of the droplet, dominant viscous forces act as dampers during both the spreading and the receding regimes. In particular, viscous forces prolong the receding period by an order of magnitude while completely suppressing secondary spreading. Finally, it is shown that the shear-thinning fluids follow the classical Tanner's law ($R \propto t^{\frac{n}{7+3n}}$; R : spread radius, t : time and n : shear thinning index), however, with a simple modification involving the substitution of shear thinning index (obtained from rheology) with the spreading exponent that is estimated from drop impact experiments.

Keywords: Drop Impact, Shear-thinning fluid, Yield stress fluid, Hydrogel

^{*}Corresponding author: purushoth@kaist.ac.kr

⁺ Equal Contributions by the Authors

Effect of Ambient Gas on Pendant Droplet Evaporation

Radhakrishnan S¹, Anand TNC¹, Shamit Bakshi^{1*}

¹Department of Mechanical Engineering and National Centre for Combustion Research and Development, Indian Institute of Technology Madras, India

Abstract

This experimental work investigates the evaporation rates of pendant droplets of liquids (water, ethanol) in different ambient gases (nitrogen, carbon dioxide and sulfur hexafluoride). The evaporation rates of liquids are studied at ambient pressure, in a sealed chamber. A steel needle is used to suspend droplets. The evaporation rates of droplets are measured using backlight shadow imaging technique. Bulk and surface temperatures of droplets are also measured during evaporation. A comparison of measured and diffusion driven rates shows that factors other than the diffusion influence the evaporation rates. The rate of evaporation of an ethanol droplet is highest in nitrogen gas and lowest in sulfur hexafluoride and a similar trend is observed in the case of evaporating water droplets. Bulk and surface temperatures of evaporating droplets in nitrogen gas are lower than those measured in sulfur hexafluoride. Evaporative cooling is more pronounced in the case of droplets evaporating in nitrogen.

Keywords: Droplet evaporation, bulk temperature, surface temperature, evaporative cooling, heat balance

*Corresponding author: shamit@iitm.ac.in

Experimental investigation of droplet evaporation under forced convective conditions

S. S. Markandeya* and R. V. Ravikrishna

Department of Mechanical Engineering

Indian Institute of Science

Bangalore 560012, India

Abstract

Droplet evaporation rate plays an important role in many chemical and combustion applications. In this experimental investigation, droplet size and velocity distributions of a sparse spray under convective flow conditions are measured using laser imaging techniques. The experimental setup involves use of a nebulizer which injects a sparse spray into a hot co-flow environment mimicking a practical spray evaporation environment. The nebulizer generates liquid sprays with droplets between 5- μm to 40- μm . The hot co-flow is generated using a $\text{H}_2\text{-O}_2\text{-N}_2$ flat flame, where the co-flow temperatures can be varied from 473 K to 1073 K. Laser diagnostic techniques such as PIV, PTV and Shadowgraphy are used to characterize the convective environment and the droplet size distribution in the spray. Classical models for droplet drag and evaporative mass transfer are used to predict the downstream droplet size distribution using experimental data at a lower height. A reasonable match is observed between predictions and experimental data on droplet size-velocity correlation.

Keywords: Evaporating sprays, Drop size measurement, Shadowgraphy, PTV, PIV, Forced convection

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

Study on Flame Spread Behavior in Two-dimensional and Three-dimensional Droplet Clouds Considering Flame-spread limit Based on Microgravity Experiments of Flame Spread of Droplet Arrays

H. Saputro^{1*}, T. Seo², and M. Mikami²

¹ Department of Mechanical Engineering Education, Sebelas Maret University
Jl. Ahmad Yani No.200 Surakarta, Jawa Tengah, Indonesia

E-mail: hermansaputro@staff.uns.ac.id

² Graduate School of Science and Engineering, Yamaguchi University
2-16-1 Tokiwadai, Ube, Yamaguchi 755-8611, Japan

Abstract

This research presents the numerical simulation of flame-spread behavior in randomly distributed large-scale droplet clouds by using a percolation approach. The percolation model was created based on so-called “Mode 3” flame-spread with paying attention to the flame-spread limit distance $(S/d_0)_{\text{limit}}$ which was obtained in microgravity experiments of flame-spread droplet arrays. The simulation applies a simple flame-spread rule that the flame can spread to droplets existing within the flame-spread limit distance $(S/d_0)_{\text{limit}}$. Droplets are arranged on the lattice points in two-dimensional (2D) and three-dimensional (3D) lattice. The group combustion of droplet clouds appears if the flame spread starting from a side in 2D droplet arrangements and a face in 3D droplet arrangements reaches the other sides in 2D droplet arrangements and the other faces in 3D droplet arrangements. The occurrence probability of group combustion (*OPGC*) was calculated as a function of the mean droplet spacing $(S/d_0)_m$. The $(S/d_0)_m$ for 0.5 *OPGC* is defined as the critical mean droplet spacing $(S/d_0)_{\text{critical}}$, which separates the droplet cloud into two groups if the lattice size becomes infinity; relatively dense droplet clouds in which the group combustion is excited through flame spread and dilute droplet clouds in which the group combustion is never excited. The results show that the $(S/d_0)_{\text{critical}}$ in 3D droplet arrangement is greater than that in 2D droplet arrangement. Even when the group combustion occurs, a portion of the droplets remains unburned and the $(S/d_0)_{\text{critical}}$ can be a good index for the stable combustion and hydrocarbon emission.

Keywords: Droplet clouds, Flame-spread, Group combustion, 2D and 3D droplet arrangements, and Percolation approach.

*Corresponding author: hermansaputro@staff.uns.ac.id

Disruptive Combustion Behavior of Gelled Ethanol Fuel droplets

Ankur Miglani^{*†}, Purushothaman Nandagopalan[†], Jerin John[†] and Seung Wook Baek

Department of Aerospace Engineering
Korea Advanced Institute of Science and Technology (KAIST)
Daejeon, Republic of Korea - 34141

Abstract

Gelled fuel droplets are known to exhibit an unsteady combustion behavior with characteristic vapor jetting events that disrupt the flame structure. This study reports first observations on the combustion characteristics of organic-gellant based ethanol fuel droplet. It is observed that vapor jetting is a dominant process through which the unreacted fuel vapors are released from the droplet domain as opposed to vapor diffusion; dominant in case of hydrocarbon fuel droplets. Jetting events occur throughout the droplet lifetime and are governed by two competing mechanisms: First, bubble formation and growth within the droplet domain that leads to internal pressure build-up and promotes rupture. Second, the gellant layer formation at droplet-gas interface that tends to sustain this pressure through elastic stretching and increased thickness (with progression in droplet lifetime). The dynamic interplay between these processes results in four dominant jetting modes with distinct intensities: First, high velocity jets that break the flame envelope. Second, low velocity jets that tend to corrugate the flame envelope. Third, continuous jets that sustain for large timescales comparable to droplet lifetime and fourth, simultaneous jetting that results in flame break at multiple points. An understanding of the jetting phenomenon is crucial since it may influence rocket engine performance by varying local fuel-oxidizer ratio and gas phase mixing.

Keywords: Droplet combustion, Gel fuels/propellants, Ethanol gel, Organic gellant, Jetting, Microexplosion.

^{*}Corresponding author: ankur.miglani@kaist.ac.kr

[†]Equal contribution by all authors

Engine Sprays - 2

Droplet diameter measurement near a nozzle exit of a gasoline DISI injector

N. Kawahara^{1*}, E. Tomita²

Department of Mechanical Engineering

Okayama University

Tsushima-Naka 3-1-1, Kita, Okayama 700-8530, Japan

Abstract

The purpose of this study is to investigate the unsteady spray structure formed by one hole of multi-hole injector using a phase Doppler anemometer. The primary spray breakup near the nozzle exit of DISI injector is important to understand the atomization process of direct injection spark ignition (DISI) injector. It is quite important to exit the appropriate stratified fuel mixture around the spark plug due to directly fuel injection inside the cylinder and a larger amount of recycled exhaust gas in DISI engine. The mixture equivalence ratio near the spark plugs at the time of the spark discharge is especially important for the ignition to succeed. Because the fuel atomization process has a dominant effect on the subsequent combustion process and directly impacts the power output, fuel consumption, and exhaust emissions, direct-injection systems into engine cylinder are increasingly the subjects of research and development projects.

In this experiment, droplet diameter and axial velocity in spray formed by one hole of multi-hole injector were measured with Phase Doppler Anemometer (PDA) to investigate atomization characteristics near the nozzle exit of the DISI injector. Firstly, droplet Stokes/ Weber number were evaluated using droplet diameter and axial velocity. It is shown that large droplets are broken up by being subjected to air resistance and are decreased. Therefore, the atomization is promoted in the central axis of the spray. Secondly, the measured droplet diameter and velocity were used as initial condition of numerical simulation. Experimental results correspond to those of numerical simulation in terms of penetration length and droplet diameter along the downstream. Three main conclusions were obtained from this research. It is possible to obtain droplet diameter and velocity distribution very close to the nozzle exit of DISI injector using HiDense PDA system. Smaller droplets under 10 μm can follow the entrainment vortex at the spray shell. On the other hand, larger droplets over 15 μm have larger velocity to penetrate the entrainment vortex. The atomization process is predominantly occurs at the center of the spray. Droplets larger than 15 μm shows higher Weber number near the nozzle, so these droplets would subject to further break up. Smaller droplets show small Weber number so these droplets are supposed to be stable.

Keywords: Phase Doppler anemometry, DISI injector, Droplet diameter, Droplet Stokes/ Weber number, Numerical simulation

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

The Effect of Fuel Injection Timing on the Macroscopic Spray Characteristics in a Gasoline Direct-Injection Engine with Optical Access

A. Bisht¹, M. Mittal^{1*}, and H. Schock²

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

²Department of Mechanical Engineering, Michigan State University, East Lansing, USA

Abstract

An optically accessible single-cylinder spark-ignition engine was used to study the influence of fuel injection timing on the macroscopic spray behavior. A multi-hole high-pressure direct-injection fuel injector was used with injection pressure of 5 MPa. Mie scattering technique was employed to visualize the liquid phase of the fuel dispersion. The obtained spray images were then used to study the macroscopic spray characteristics such as spray structure, spray tip penetration length and spray area. Considering different fuel injection timings, results showed that the spray tip penetration was longer at 210° before top dead center (BTDC) of compression compared to injection timings at 240° BTDC and 180° BTDC. Comparing different engine speeds, it was found that the spray areas were comparable at initial time stamps from the start of injection, but, it was enhanced considerably with the increase in engine speed at later time stamps.

Keywords: Gasoline direct-injection engine, high-speed imaging, macroscopic spray characteristics

*Corresponding author: mmittal.28@gmail.com

Liquid and Vapor Penetration Length Estimation for GDI Spray of Iso-Octane and Alcohols at Elevated Pressure and Temperature Conditions

Rakesh Kale^{*}, R. Banerjee
Department of Mechanical and Aerospace Engineering
Indian Institute of Technology Hyderabad
Hyderabad 502285, India

Abstract

Gasoline Direct Injection technology is gaining lot of interest because of its improved fuel efficiency and higher power. Due to environmental concerns and reduced dependency on fossil fuel, blending of alcohol based bio-fuels in the regular gasoline is becoming more common. Since the technology is completely spray controlled, changing the thermo-physical properties of the fuel can adversely affect the engine performance. In the present work Isooctane and two alcohols namely ethanol and n-butanol were selected to study their macroscopic spray behavior such as liquid penetration and the vapor penetration length. Six holes solenoid injector was used to study the fuels under the engine like elevated pressure and temperature conditions. Schlieren based visualization technique was used to track the vapor phase of the spray, whereas liquid portion was captured using Mie-scattering. Separate experiments were conducted to detect the liquid and vapor phase of the spray plume. Higher pressure and temperature was created by combusting a mixture of air and acetylene. Results show that thermo-physical properties such as boiling point and the latent heat of vaporization of the fuels play an important role in the spray penetration. The liquid penetration length of isooctane was less than the alcohols because its latent heat of evaporation is the least and therefore droplet cooling was correspondingly also less. Alcohols showed the higher liquid penetration due to their high latent heat of evaporation.

Keywords: GDI Spray, Schlieren, Mie-scattering, liquid penetration, vapor penetration, alcohols

^{*}Corresponding author: me13p1007@iith.ac.in

Numerical Prediction of Purging of a GDI nozzle at the End of Injection

S.Mouvanal^{1,2*}, A. Burkhardt², D.Chatterjee¹, and S.Bakshi¹

1.Department of Mechanical Engineering, Indian Institute of Technology Madras, India

2.Continental Automotive GmbH, Regensburg, Germany

Abstract

Recently there has been a raised interest on the internal nozzle flow, its transient behavior during needle closing stage. This also include estimation of the amount of residual fuel left in the sac volume at the end of the injection because of its direct impact on engine performance and emission. In the present study, numerical simulations were performed on a multihole GDI nozzle using ANSYS Fluent in order to understand the internal flow behavior during the fully open, closing and closed stages of needle. Multiphase model involving three phases with VOF method for interface tracking was used. Mass transfer was modeled using Schnerr-Sauer cavitation model and turbulence was modeled using $k-\omega$ SST with Reboud's correction on turbulence viscosity term. Moving mesh method which works on a user defined interpolation strategy was used to depict the prescribed needle motion. The simulation provides insight into the physics of internal flow dynamics in nozzle holes and the sac volume involving vapor generation, back flow of air from cylinder side to the sac volume and removal of residual liquid in the sac volume and nozzle. The result was compared with the X-ray measurements from the literature.

Keywords:GDI nozzles, EOI, cavitation,hydraulic flip, multiphase, purging

*Corresponding author: sandeep.mouvanal@continental-corporation.com

Experimental analysis of spray characteristics using fuel heating for D.I.S.I engines

E. Matsumura*, K. Imori, Y. Sakai, J. Senda

Department of Mechanical Engineering

Doshisha University

1-3, Tatara, Miyakodani, Kyotanabe-shi, Kyoto, 610-0394, Japan

Abstract

The flash boiling by fuel heating is a useful method to control the time spatial spray characteristics such as atomization of droplets, vaporization and air-fuel mixture density. It is one of the important phenomena for a direct injection gasoline engine (D.I.S.I) as a next generation powertrain. This report focuses on flash boiling spray, using fuel heating to understand its physical phenomena with scattered light method, schlieren photography and Super High Spatial Resolution Photography (SHSRP). Fuel was iso-octane and a single hole nozzle were used for the purpose of basic phenomenon analysis. The influence on spray shape can be shown by schlieren photography. Spray droplet diameter and spray dispersion at the nozzle exit are observed by super high spatial resolution photography which is our original development technique. As a result, spray dispersion and atomization were controlled and improved drastically by flash boiling. It was clarified that fuel spray shape greatly varied by heating more than saturated temperature and not only atomization of the whole spray is fine, but also a spray core region decreases. In the heating condition that is slightly lower than critical temperature, a shock wave “MACH DISK” was confirmed at nozzle exit.

Keywords: fuel heating spray, flash boiling, spray characteristics, D.I.S.I,

*Corresponding author: ematsumu@mail.doshisha.ac.jp

Tuesday

8th November 2016

Plenary Lecture 2

Artificial Control in Spray Combustion Process Based on Fuel Design Concept for Engine Systems

Jiro SENDA, Prof. Dr-Eng
Doshisha University, Mechanical Engineering Department
Spray & Combustion Science Laboratory, Energy Conversion Research Center
Kyotanabe, Kyoto 610-0321, JAPAN

jsenda@mail.doshisha.ac.jp

<http://comb.doshisha.ac.jp>

Abstract

I have proposed novel kind of fuel design researches for both Diesel and Gasoline engines by applying several kind of mixing fuels from the basis of just fuel side approach, as another selective way to get higher efficiency and lower emissions. Here, I am focusing on in-cylinder spray improvement and combustion control based on flash boiling spray. I will introduce flash boiling process of Gasoline spray and mixing fuel with gasoline and Diesel gas oil, as well as flash boiling spray model. Also, modelling and simulation of evaporation process in multi-component fuel sprays. My fuel design researches have been composed with CO₂ - Diesel gas oil mixture case, mixing fuel with Gasoline component and Diesel gas oil component and further mixing fuel with gas fuel and Diesel gas oil component. This talk is a kind of summary of these fuel design approach studies including the promising concept, practical studies and future extending research aspect. In this approach, the flash boiling spray is applied to control the physical evaporation process and some kinds of mixing fuels are used to control the chemical burning process. Thus, in the experiments, mixing fuel of liquefied CO₂ and *n*-tridecane (Gas oil) is used to obtain the simultaneous reduction both soot and NO_x, and mixing fuel of gas or Gasoline component and Diesel gas oil component to control both evaporation and ignition processes.

Spray Characterization - 2

Design and Analysis of a High Pressure Swirl Atomizer

Suresh Lal, Pranay Matkar
Center for Fire Explosive & Environment safety
DRDO, New Delhi, India

Abhijit Kushari*
Indian Institute of Technology Kanpur, Kanpur, India.

Abstract

This work was focused on design, development and characterization of a high-pressure swirl atomizer. The nozzle was designed for an inlet supply pressure of 80 bar of water flow. The spray characteristics were experimentally evaluated on the designed and fabricated nozzle. The characterization involved the measurement of liquid flow rate; spray cone angle and the droplet diameter using laser diffraction technique for different liquid supply pressures. Optimization of Sauter Mean Diameter of the droplets was achieved by varying the liquid supply pressure over a range of 40 to 100 bar. The experimental observations of SMD were compared with the analytical model developed by Wang and Lefebvre (1987). It was observed that the coefficient of discharge through nozzle was nearly 0.65, much better than that predicted by the theory proposed in the available literature. The spray cone angle was decreasing with the increase in pressure and the measured droplet SMD was within the same order as predicted by Wang and Lefebvre.

Keywords: Spray, atomization, high pressure, swirl, drop size, discharge coefficient; spray cone angle

* Corresponding Author: Email- akushari@iitk.ac.in, Ph. - +91-512-2597126

Spray Characteristics from a Twin-Fluid Atomizer with Internal Impactor

Deepak Kumar, Tushar Sikroria and A. Kushari*
Indian Institute of Technology Kanpur,
Kanpur -201806, UP, India

Pramod Kumar, P. V. C. Rao and N. V. Choudary
HPCL Corporate R&D Centre, Bangalore-560067, India

Abstract

In Fluidized Bed Catalytic Cracking (FCC) process, hydrocarbon feed undergoes vapour phase cracking in presence of hot regenerated catalyst to produce valuable products like LPG, Gasoline and Diesel. FCC feed injection system is most critical hardware component of FCC unit in order to get desired product yield by minimizing the undesirable dry gas and coke yield. Typically, twin-fluid nozzles (hydrocarbon and stream) are used to atomize the feed. In the present study, a twin-fluid injector, with an internal impactor to minimize the droplet size and velocity, is designed and developed. The performance of the feeding injector was evaluated using water and air as operating fluids and the droplet size and velocity were measured 150 mm downstream of the injector tip using a PDPA system for different water and air flow rates. The average droplet size (D32) showed an increase while the droplet velocity remained almost constant with the increase in the liquid flow rate for a given flow rate of air, consistent with the increase in droplet size with decreasing air-liquid ratio for twin –fluid atomizers. But, for a given liquid flow rate, the droplet SMD decreased and the droplet velocity increased with increasing airflow rate, which can be attributed to the increase in overall kinetic energy due to the increase in air flow rate. The flow rate of liquid was seen to be independent of air flow rate unlike conventional twin-fluid atomizers. The droplet size was found to be a function of ALR and the local volume flux of the droplets was found to be a function of the liquid flow rate.

Keywords: fluidized bed catalytic cracking, twin fluid atomizer, internal impactor, PDPA measurement, drop size

* Corresponding Author: email: akushari@iitk.ac.in, Ph. +91-5122597126.

Spray cross-sectional pattern analysis of a SIDI multi-hole injector under various flash boiling conditions

Shangze Yang¹, Min Xu^{1*}, David L. S. Hung^{1,2}, Shengqi Wu¹

1. School of Mechanical Engineering, National Engineering Laboratory of Electronic Control Technology

2. University of Michigan–SJTU Joint Institute
Shanghai Jiao Tong University
Shanghai, 200240 China

Abstract

Flash boiling spray generated by the injection of superheated fuel can greatly improve the fuel atomization, evaporation and combustion processes, which results in improved fuel economy and reduced exhaust emissions. In this study, the spray pattern of a six-hole injector at various planes below the injector tip was recorded using the planar laser Mie-scattering imaging method. N-hexane was chosen as the test fuel. Fuel temperature ranged from 25 °C to 85 °C, and ambient pressure varied from 20 kPa to 200 kPa, which producing a wide range of superheated conditions. The fuel injection pressures were 15MPa and 25MPa. The results showed that both increasing fuel temperature and decreasing ambient pressure led to expanded fuel plumes and more compact fuel plumes distribution, of which the distance between the centroid of each plume and injector tip decreased. The spray collapse length increased with decreasing Pa/Ps ratio, and also increased with injection time elapse under superheated conditions. Higher fuel injection pressure suppressed the formation of spray collapse

Keywords: SIDI injector, spray collapse, superheated condition, planar laser Mie-scattering imaging

*Corresponding author: mxu@sjtu.edu.cn

Experimental Investigations on Spray Characteristics of Potential Fuels for Advanced Combustion Engines

M. Murugesu Pandian, K. Anand*

Department of Mechanical Engineering
Indian Institute of Technology Madras
Chennai 600036, India

Abstract

Advanced low temperature combustion engines show higher sensitivity to changes in fuel properties influencing fuel reactivity and physical delay which in turn depend upon fuel spray characteristics. Spray characteristics of potential fuels for advanced combustion engines are sparsely available. The present work intends to study the effects of changes in fuel properties and fuel injection pressure on the macroscopic fuel spray behaviour. The experiments were carried out in a constant volume spray chamber with different potential fuels for advanced combustion engines, viz. Diesel with 10% gasoline (DG10), Diesel with 20% gasoline (DG20), Diesel with 20% kerosene (DK20), and Biodiesel with 30% kerosene (BK30). Fuel injection pressure and the injection strategies have been chosen as representative of direct injection diesel engine working conditions. The injection pressure was varied by using a flexible common rail test rig and an engine controller. The test fuels are injected in a constant volume spray chamber that was maintained at 3 MPa pressure and ambient temperature conditions. The spray characteristics such as spray tip penetration distance, spray cone angle were obtained from a spray visualization system for the different fuel blends. The obtained results show that the changes in physical properties of the fuel blends significantly affect the macroscopic spray behaviour. Further, a comparison of obtained results with that of conventional diesel fuel show significant differences in the spray characteristics of these fuel blends.

Keywords: Spray, atomization, Bio fuel, Spray penetration length, Spray Cone Angle

*corresponding author: anand_k@iitm.ac.in

Experimental and numerical characterization of fatty acid and methyl ester sprays

R. D. Lanjekar^{1*}, D. Deshmukh²

Spray and Combustion Laboratory, Discipline of Mechanical Engineering,
Indian Institute of Technology Indore, Indore 453552, India.

Abstract

The spray characteristics of Straight Vegetable Oil (SVO) can be better understood by studying sprays of its individual components which are pure fatty acids. Oleic fatty acid is one of the major component of SVOs commonly used as fuel for diesel engine or for producing biodiesel. This research paper presents experimental and numerical study on spray characteristics of pure oleic fatty acid and methyl oleate. The simulation results are validated with experimental spray tip penetration for n-heptane. The validated model is used to predict spray tip penetration for methyl oleate and oleic fatty acid. It is found that spray tip penetration predictions are closely matching measurements for methyl oleate, whereas predicted spray tip penetration for oleic fatty acid is 7-10% lower than that of the experimental measurements.

Keywords: oleic fatty acid, methyl oleate, n-heptane, spray tip penetration, openfoam

*Corresponding author: phd12120301@iiti.ac.in

Effect of biodiesel-ethanol blends on spray atomization

V. D. Chaudhari^{1*}, A. Kulkarni², D. L. Deshmukh³

Spray and Combustion Laboratory, Discipline of Mechanical Engineering
Indian Institute of Technology Indore.
Indore 453552, India

Abstract

In the present research work, biodiesel-ethanol (BDE) blend sprays are studied using high-speed shadowgraphy technique. Ethanol is blended with biodiesel in 10%, 20% and 30% proportion by volume. Pure biodiesel (BD100) is used as a reference fuel. The experimental results show a decrease in spray tip penetration and increased cone angle with increasing ethanol percentage in the BDE blend. The BDE30 blend shows a maximum spray cone angle and reduced spray tip penetration among all the blends for all fuel injection pressures. Finely dispersed droplet cloud is observed in front of the spray tip for BDE30 spray. The reduced kinematic viscosity, surface tension and density of BDE blends may be a reason for improved atomization.

Keywords: Biodiesel-ethanol (BDE) blends, spray cone angle, spray tip penetration

* Corresponding author: phd1401203003@iiti.ac.in

Impinging Sprays

Like-on-Like Impingement and Atomization of Highly Viscous Fluids

I. C. Ruiz-Rodriguez^{*}, M. Avulapati and L. C. Ganippa

Department of Mechanical Engineering

Brunel University London

London, UB8 2PH, United Kingdom

Abstract

Atomization of high viscosity fluid finds its applications in the fuel injection systems for space propulsion, agricultural sprays, chemical processes and for pharmaceutical industries. Despite its wide application, the exact mechanisms involved with the breakup of these high viscous fluids have not been completely understood. In this work the break-up characteristics of a highly viscous liquid sheet was investigated through the like-on-like injection configuration. The influence of nozzle exit flow velocity profile before impingement was varied to study its effect on sheet topology and droplet break-up for water-glycerol fluid mixtures at different fluid injection pressures. The nozzle exit profile was varied by using nozzles of different length to diameter ratios.

Using shorter injectors, it was found that atomization resembled that of lower viscosity fluids. The liquid sheet destabilized into a wide range of drops promptly, even at low injection pressures. In contrast, for similar injection pressures, using the long injectors the sheet stretched and did not break down to significant droplets. The former effect was attributed to the incomplete development of the boundary layer in the nozzle, thus upon the jet exit, instabilities developed promptly. The findings are significant; as they demonstrate that under the correct injection conditions, atomization of high viscosity fluids becomes a reality even at low injection pressures.

Keywords: Atomization, high-viscosity fluids, sheet break-up, perforations, like-on-like impingement

^{*}Corresponding author: 1115854@my.brunel.ac.uk

Experimental Investigation of Behavior and Drop Size of Impinging Spray at Ultra-Low Temperature Conditions

¹Huijia Lv, ^{1,2}David L.S. Hung*, ²Min Xu, ²Hujie Pan, ²Xue Dong

¹University of Michigan-Shanghai Jiao Tong University Joint Institute

²School of Mechanical Engineering

Shanghai Jiao Tong University

Shanghai, China

Abstract

Spray impingement on cylinder wall and piston affects the fuel-air mixing and the subsequent combustion as well as emission in spark-ignition direct-injection engines. It reduces the engine cold start capability especially in cold regions. To characterize the behavior and droplet size of the impinging spray at ultra-low temperature conditions, Phase Doppler Interferometry (PDI) was implemented to obtain the spatial and temporal distributions of the velocity and droplet size. Temperature of as low as -30°C was achieved for both fuel and impingement plate in this experimental investigation. The formation and development of the impinging vortex of the spray structure is described qualitatively in both time domain and spatial domain by correlating the distribution of validated droplet counts and velocity component normal to the impingement plate. Statistical quantities of droplet size are calculated in the downstream region where only post-impingement droplets are present. Results show that the behavior of the vortex is similar at ultra-low temperature conditions. In addition, plate temperature has a weaker impact on the post-impingement droplet size when compared to the fuel temperature.

Keywords: Spray impingement, behavior, drop size, PDI

*Corresponding author: dhung@sjtu.edu.cn

Drop Size Distribution Effects In Intersecting Sprays

K. Dhivyaraja* , Sri Vallabha Deevi and Mahesh V. Panchagnula

Department of Applied Mechanics
Indian Institute of Technology Madras
Chennai 600036, India

Abstract

Intersecting sprays occur in a wide range of engineering applications. It is commonly assumed that the drop size distributions in such sprays can be modeled by the principle of superposition. We herein test that proposition experimentally using Phase Doppler Particle Anemometry (PDPA). Two spray nozzles are placed close to each other and PDPA measurements of droplet properties at several locations in the intersecting region are obtained. Measurements from individual sprays, at the same location, are obtained by turning on one spray at a time. Intersecting spray effect is tested by comparing the drop size probability density function (*pdf*) from the intersecting spray with the *pdf* calculated from superposing the individual sprays at the same location. Measurements are obtained for two different flows – a) dilute low Weber number spray from a nebulizer, b) dense high Weber number flow from a simplex atomizer. These two experiments offer different conditions in the intersection region. In the simplex atomizer spray, large number density leads to significant collision rates, which could alter the droplet size distribution.

Keywords: Intersecting Sprays, PDPA, Probability Density Function, Superposition

*Corresponding author: dhivyan26@gmail.com

Mixing field and penetration of impinging twin liquid jets injected obliquely against Supersonic air cross flow

Venu G*, Venkat S Iyengar, Srinivas J., Sathiyamoorthy K.
Propulsion Division, CSIR-National Aerospace Laboratories
Bengaluru, KA 560037 INDIA

and

T.M. Muruganandam, and S. R. Chakravarthy
Department of Aerospace Engineering
Indian Institute of Technology Madras
Chennai, TN 600036 INDIA

Abstract

The spray characteristics of a novel twin-jet liquid injector configuration with various combinations of forward (FA) and toe-in angles (TA) in a Mach 2 cross flow are studied. Four different injector configurations are considered with two orifice sizes of 0.8 mm diameter spaced 3 mm apart and varying (TA) and (FA) 0°- 30° from vertical in both the upstream and lateral directions. These injectors are assessed for their performance in terms of their mixing and penetration characteristics for different momentum flux ratios in the range 3-12. The Mie scattering technique is adopted to quantify the mixing field and penetration of the different configurations in a rectangular test section with full optical access. The degree of unmixedness (DU) is deduced from the liquid volume fraction fields obtained by Mie scattering at different cross-sections at and downstream of the injection plane. Together the variation of the DU and penetration along the flow up to 50 mm downstream from the injector indicate that the injector with FA and TA = 30° performs the best at lower momentum flux ratios (3, 6) whereas the one with FA = TA = 15 is superior at higher ratios (12).

Keywords: Injector, Scramjet, Combustor, Degree of Unmixedness, Penetration, Mie scattering, Supersonic cross flow

Numerical Simulation of Atomization by the Oblique Collision of Two Newtonian Jets

Navaneeth Krishnan ^{*1}, Rajagopal P S ², Madhukar M Rao ² and Akshai K Runchal ²

¹ACRi infotech Pvt.Ltd, AECS Layout, Kundalahalli, Bengaluru-560037, India

²CFDVRI, Mount View Complex, McLeod Ganj, Himachal Pradesh-176219, India

Abstract

In this study, unsteady incompressible two-phase flow is simulated using OpenFOAM's InterFOAM 3D VOF solver. The interFOAM solver is applied to study the atomization phenomena by the oblique collision of two Newtonian jets for different sets of Reynolds numbers and Weber numbers. At very low Reynolds number and Weber number, the closed rim is formed due to low velocity and high surface tension. Thin sheets and droplets are formed for higher Reynolds number flows. Three sets of Weber numbers and Reynolds numbers are used for the simulation. Oblique collision of two Newtonian jets at high Reynolds number flows resulting the wavy nature followed by the ligament formation and droplet formation. The rate of atomization is directly proportional to impinging angle. The solver is validated with the classical 3D dam breaking problem and results are compared with the experimental results of Martin and Moyce (1952). Here two graphs are plotted for the validation and grid independency test. Non-dimensional height of remaining water column and non-dimensional surge position are plotted against non-dimensional time for validation and grid independency test. The details of the validation problem and application are given in the paper.

Keywords: OpenFOAM, Multiphase flow, VOF, Dam Breaking, Atomization

*Corresponding author: navaneeth.acricfd@gmail.com

Numerical Simulation of Impinging Jets with Newtonian and Non-Newtonian Fluid using 3-D unsteady Finite Element VOF solver

Atish Ezhuthasan* and Jayakumar J S
Department of Mechanical Engineering
Amrita School of Engineering
Amrita University
Amritapuri Campus
Kollam 690525, India

T Jayachandran and Ajith M
VSSC/ISRO
Thiruvananthapuram 695022, India

Abstract

In this study, moving interface boundary in a three- dimensional unsteady incompressible two phase flow is simulated using finite element method. The governing incompressible Navier-Stokes equations are discretized using Taylor-Galerkin method which provides a natural up winding term. An Eulerian velocity correction method is used to achieve mass conservation and the resulting Pressure-Poisson equation is solved using a conjugate gradient method in an element by element manner. A modified volume of fluid method is used to track the moving interface and the Continuum Surface Force model is used to incorporate the surface tension. A transformation function of volume fraction is solved to minimize the diffusion at the interface. The code is validated with dam breaking problem and excellent results for non-dimensional height position and non-dimensional time are achieved. The 3D solver is applied to the simulation of atomization by two impinging jets. Numerical simulation is carried out on two impinging jets with three set of Weber and Reynolds number for Newtonian as well as non-Newtonian fluids. In non-Newtonian fluids viscosity varies due to stress and viscosity is incorporated using power-law constitutive equation. Gels are used as non-Newtonian fluids. In this paper aluminized kerosene gel is used. Breakup characteristics and sheet formation of Newtonian and non-Newtonian fluid are discussed.

Keywords: multi-phase flow, non-Newtonian fluids, Newtonian fluids, atomization, breakup characteristics, surface tension.

* Corresponding author: adi25291@gmail.com

Air-Assisted Atomization, Supersonic Flows

A Spray Structure Model of Liquid Jet being injected into Supersonic Gaseous Cross-flow

Qinglian Li^{*1,2}, Liyin Wu³, Chun Li^{1,2}, Yeming Zeng⁴

- (1. Science and Technology on Scramjet Laboratory, National University of Defense Technology, Changsha 410073, China
2. College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, China
3. Hyper velocity Aerodynamic Institute, China Aerodynamics Research and Development Center, Mianyang, China
4. Shanghai Institute of Space Propulsion, Shanghai 201112, China)

Abstract

Phase Doppler Anemometry (PDA) and Pulsed Laser Background Imaging methodology were applied in the study of the atomization process of round liquid jet of 1mm diameter being injected into supersonic gaseous crossflow with Mach number of 2.1. Plentiful results were gained and it showed that surface waves appeared clearly leading to the breakup of the jet and atomization process developed quickly and intensively. Based on the experimental results, the paper presented a structure model to describe the physical process of atomization and the spray distribution. The whole spray was distinguished into three adjacent regions with noticeable features along the gaseous streamwise: continuous jet region, quick atomization region and uniform spray region. The jet breakup position was decided quantitatively from a series of high-resolution spray images, which indicated the finish of 1st region. Droplets size data being gained by PDA were applied to infer the position where the secondary atomization was accomplished, and it is the end point of the 2nd region. In the 3rd region, the Sauter Mean Diameters vary slightly streamwise although there are still the processes of a few large droplets' breakup and small droplets' coalescence. The spray in the quick atomization region could be divided into three layers along the injection direction: thin spray layer with large droplets, thin spray layer with small droplets, dense spray layer with large droplets.

Keywords: supersonic crossflow, liquid jet, surface wave, spray proportion, boundary band, PDA, pulsed laser background imaging, spray structure.

*Corresponding author: peakdreamer@163.com

Investigations on Droplet Sizes of Liquid Jet Atomization in Supersonic Crossflow

Yeming Zeng¹, Qinglian Li², Liyin Wu², Chun Li²

¹Shanghai Institute of Space Propulsion, Shanghai 201112, China

²College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, China

Abstract

Particle Dynamics Analysis (PDA) technique was used to investigate the droplet sizes of liquid fuel atomization in Ma2.1 supersonic crossflow. The distribution characteristics of droplet diameters in different spray locations and the change of droplet sizes along gas flow direction were studied. Influence of large droplets to Sauter Mean Diameter (SMD) was analyzed under the condition of limited PDA sampling. The probability threshold method was introduced based on cumulative probability of droplets. A deep analysis was carried out on the distributions of SMD along gas flow and longitudinal direction of the spray.

Keywords: PDA, supersonic crossflow, probability threshold method

Experimental study on cavity influence on liquid jet-penetration in supersonic crossflow

Chenyang LI 1,2, Qinglian LI 1,2 *, Chun LI 1,2 and Liyin WU 1,2

1. College of Aerospace Science and Engineering

National University of Defense Technology

Changsha 410073 , China;

2. Science and Technology on Scramjet Laboratory

National University of Defense Technology

Changsha 410073, China

Abstract

The incorporation of cavities in supersonic combustion chambers is an effective mean of slowing down the flow for fuel injection and hold stable combustion. To understand the spray characteristics in supersonic crossflow, especially for the liquid fuel injection, is essential to the optimum design of supersonic propulsion mechanisms. An experimental investigation on the penetration of cavity upstream liquid injection was conducted with high-speed photography. Injection parameters contained momentum flux ratios and jet positions. The injection driven pressure was adjusted to change the jet-to-freestream momentum flux ratio. The effect of jet positions, classified as 5 distances of the orifice to the cavity front edge, was studied experimentally. For comparing and analyzing the influence of the cavity, flat-plate injection was also investigated by replacing the cavity module. Results indicate that for the liquid injection with a cavity, the penetration was consistent with plate jet penetration in the near-field. The cavity jet penetration decreased at the front edge because of the mass exchange between the cavity and the main flow field. Jet penetration increased at the outlet of the cavity due to the shear layer impinging the cavity trailing edge, producing a high pressure field. The penetration decreased to the lowest level at the cavity front edge when the jet position is $3D$ (where D is the cavity depth).

Keywords: cavity ; supersonic crossflow ; liquid jet ; penetration

* Corresponding author: peakerdreamer@163.com

Investigation on forward injection of impinging liquid jets in supersonic cross flow

Sathiyamoorthy K^a and Venkat S Iyengar^a

^aScientist, Propulsion Division, CSIR-National Aerospace Laboratories
Bangalore-560017, India

Panchabudhe Lakhan M^c, Muruganandam T M^b and S R Chakravarthy^b

^b Professor, Department of Aerospace Engineering, IIT-Madras

^c MTech Student, Department of Aerospace Engineering, IIT-Madras
Chennai-600036, India

Abstract

The effects of forward and toe-in angles of twin impinging jets in Mach 2 air cross flow on vertical and upstream penetration and the shock structure are studied. The injector has twin orifices of 0.7 mm diameter with lateral spacing of 3 mm. The forward angle is varied as 30°, 45° and 60° with respect to vertical. Two toe-in angles of 45° and 60° are investigated. Using water as injectant, the jet to air momentum flux ratio (J) is varied as 1.8-6.4. Schlieren imaging at 1000 fps is performed to examine the flow field. The images are processed to obtain the plume trajectory and penetration. Higher J increases the maximum vertical and upstream penetration for a particular injector. Increase in the forward angle increases the upstream penetration with penalty of reduction in the vertical penetration. Considerable reduction in vertical penetration is observed when the toe-in angle is increased from 45° to 60° for the same forward angle. The toe-in angle has minimal effect on the upstream penetration of the plume but has considerable effect on the shape of the upstream plume trajectory.

Keywords: Supersonic combustion, impinging jet, jet in supersonic cross flow, forward angle, Toe-in angle.

Engine Sprays - 3

Analysis in Unsteady Evaporating Gasoline Spray with LIEF Method

Masahiro Matsuura^{*}, Vikas Semwal, Shingo Murakami,
Eriko Matsumura, and Jiro Senda
Department of Mechanical Engineering
Doshisha University
Kyoto, 610-0321, Japan

Abstract

These days, direct injection system is adopted in not only diesel engine but also gasoline engine to improve its thermal efficiency. Direct-injection (DI) gasoline engine has higher thermal efficiency than conventional port-fuel injection (PFI) gasoline engine. However, it is more difficult to make the fuel-air mixtures in a short time and it affect combustion characteristics strongly. Therefore, for further improvement in DI gasoline engine, it is necessary to understand the spray, the mixture formation process and the evaporation process. The purpose of this paper is to characterize gasoline direct injection spray. The Laser-Induced Exciplex Fluorescence (LIEF) method is diagnostic method which can clarify the liquid and vapor phase simultaneously separating fluorescence signals from liquid and vapor phases of spray. When the fuel doped two tracers is injected from nozzle, monomer and Exciplex emit different wave length fluorescence. In this research iso-octane is used as fuel, and Fluorobenzene and Triethylamine (TEA) are chosen as dopant. In order not to change the property of the fuel, an attention is paid to the proportion of the dopant. Our research use gasoline direct injection multi hole nozzle in high pressure and high temperature field. Using injection pressure and photograph timing as the parameters, spray atomization was observed and analyzed. As injection pressure increase, atomization is improved and spray tip penetration is decreased. It is observed that vapor luminance increases instead of liquid luminance decrease spatiotemporally as photograph timing is late.

Keywords: Laser-Induced exciplex fluorescence, D.I.S.I. engine, gasoline, Spray

^{*}Corresponding author: duq0540@mail4.doshisha.ac.jp

Verification of Breakup Model for Direct Injection Gasoline Spray

Tomoyuki Mukayama*, Hirotaka Tabata, Eriko Matsumura, and Jiro Senda
Department of Science and Engineering
Doshisha University
1-3 Tataramiyakodani, Kyotanabe, Kyoto, 610-0394, Japan

Abstract

In direct injection gasoline engines, the spray structure has dominant effect on the ignition and combustion processes. In this study, the authors conducted analysis of non-evaporative spray features injected from multi-hole nozzle for direct injection gasoline engine by experiment and numerical simulation using the commercial CFD code. Here, in the experiment the spray features were grasped with Super High Spatial Resolution Photography (SHSRP), which was originally developed by the author's group. Further, the authors used the Kelvin-Helmholtz and Rayleigh-Taylor (KH-RT) model, which is a breakup model to be used in the analysis of diesel spray and the Modified Taylor Analogy Breakup (MTAB) model that describes the breakup at low fuel injection pressure proposed in this study in numerical calculation. And, it was verified for the applicable limit of the breakup model to the direct injection gasoline spray by comparing the results of experiment and numerical calculation. As a result, spray tip penetration and droplet diameter are in good agreement with experimental results in more than $\Delta P_{inj}=8\text{MPa}$. But it is difficult to apply the MTAB model in $\Delta P_{inj}=4\text{MPa}$.

Keywords: Gasoline Spray, Numerical calculation, Breakup model

*Corresponding author: euq1503@mail4.doshisha.ac.jp

Spark-Ignited Combustion Characteristics of Gasoline Spray Injected by Hole-Type Nozzle for Direct Injection Engine

Keiichi Katayama^a, Gnanasekaran KishorKumar^a, Hongliang Luo^a, Keiya Nishida^{a,*}, Yoichi Ogata^a

^a Department of Mechanical System Engineering, University of Hiroshima, Higashi-Hiroshima,
739-8527, Japan

Abstract

An experimental study was carried out to investigate the effects of ignition and impinging strategy on spray and combustion characteristics in a spray guided direct injection spark ignition (DISI) engine under stratified operations. Different impinging and ignition strategies, which included impinging angle, ignition position and ignition timing were applied. Combustion in a stratified DISI engine was found to have both lean premixed and mixing controlled flame characteristics. The stratified mixture characteristic for the ignition timing was verified as a dominant factor of the flame characteristics. High speed shadowgraph images were used to identify the ignition robustness and the inter-action of spray with spark discharge. Two-color Pyrometry method was implemented to examine the combustion characteristic on soot concentration by varying the impinging and ignition parameters. The results show that the spray tip penetration after the spray has impinged on the flat wall surface become significantly better at acute impinging angle than at perpendicular impinging angle. The impingement spray flame duration lasted much longer when compared to the free spray. In addition, effects of ignition conditions on combustion characteristics against the wall were also investigated. These investigations are favorable in further understanding the impinging spray and combustion characteristics of DISI engines

Keywords: Free spray, Wall-impingement-spray, Combustion, Direct injection gasoline engine

*Corresponding author: nishida@mec.hiroshima-u.ac.jp

Air Entrainment of High Pressure Methane Gas and Effects of Nozzle Diameter and Injection Pressure

Hiroshi Tajima*, Daisuke Tsuru and Yusuke Shintani
Interdisciplinary Graduate School of Engineering Sciences
Kyushu University
Kasuga, Fukuoka 816-8580, Japan

Abstract

The air entrainment process of a high-density gas jet was investigated through PIV measurements of the ambient air motion around a gas jet in a constant volume chamber for visualization. The gas jet penetration process was simultaneously photographed by a shadowgram technique. Pure methane was selected as reference fuel gas and its injection pressure was controlled around 30 MPa as a typical value in gas direct injection (GI) engines, but the ambient air pressure was set at 3.0 MPa much lower than the GI engines' compression pressure usually exceeding 10 MPa. Moreover, the ambient air was kept cold at room temperature. Additional measurements of a gas oil spray were done to capture essential difference between the gas jet and the liquid spray under the same ambient conditions and with the same injection momentum. At first, PIV measurements revealed that the gas jets induced more furious air motion than the liquid spray especially around the jet tip, where the jet seemed to have clear spherical distribution of velocity vectors. Resulting backward air flow behind the jet tip collided with forward air flow surrounding the jet body and seemed to cause strong air entrainment. Second, in place of the so-called breakup length in liquid sprays, the gas jet started with a shock train structure as an underexpansion flow, and no entrainment occurred into the structure. After the shock train, the jet tip penetration showed similar behaviors with the spray tip of the reference gas oil. At last, the jet tip penetration was found to have reasonable agreement with prediction results on a classic momentum theory. So, the differential pressure instead of the pressure ration between the gas injection pressure and the ambient pressure would be better to characterize the high density gas jet.

Keywords: Gas jet, Air entrainment, PIV measurement, momentum theory

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

Schlieren, Shadowgraph, Mie-scattering visualization of diesel and gasoline sprays in high pressure/high temperature chamber under GDCI engine low load condition

D.H. Kim^{1*}, S. S. Park², and C. S. Bae²

1.Department of Aerospace Engineering

2.Department of Mechanical Engineering

Korea Advanced Science and Technology

Republic of Korea

Abstract

Three visualization methods, Schlieren, Shadowgraph, and Mie-scattering, were applied to compare diesel and gasoline spray structures. Fuels were injected into a high pressure/high temperature chamber under the same in-cylinder pressure and temperature condition of low load in a gasoline direct injection compression ignition (GDCI) engine. Two injection pressures (40 and 80 MPa), two ambient pressures (4.2 and 1.7 MPa), and two ambient temperatures (908 and 677 K) were used. The images from the different methods were overlapped to show liquid and vapor phases more clearly. Vapor developments of the two fuels were similar; however, different liquid developments were seen. At the same injection pressure and ambient temperature, gasoline liquid propagated quickly and evaporated rapidly than diesel liquid phase. At the low ambient temperature and pressure condition, gasoline and diesel sprays with higher injection pressures showed longer liquid lengths due to higher spray momentum. At the higher ambient temperature condition, the gasoline liquid length was shorter for the higher injection pressure. Higher volatility of gasoline is the main reason for this shorter liquid length under higher injection pressure and higher ambient temperature conditions. In contrast to diesel spray under higher injection pressure for lower particulate matter (PM) emissions, gasoline spray under higher injection pressure leads to unstable combustion due to excessively lean mixture formation.

Keywords: Spray, gasoline, diesel, Mie-scattering, Shadowgraph, Schlieren

*Corresponding author: proftwixt@thescientist.edu

Swirl Spray

**Experimental study on the effect of swirl chamber geometry
on the dynamic characteristics of closed-type swirl injector**

Y. J. Chung, S.K. Jeong, and Y. Yoon *

Department of Mechanical and Aerospace Engineering
Seoul National University
Seoul, 151-472, Korea

Abstract

Injector is one of the key component in liquid rocket engines especially when suppressing the combustion instability. Dynamic characteristics of the injector should be investigated in order to establish criteria for injector modification. The liquid film thickness was selected as the primary output parameter since it is closely related with the mass flow rate of the swirl injector. closed-type swirl injectors have been widely used in liquid rocket engines. In this study, dynamic characteristics of the closed-type swirl injector with varying geometries were investigated through experiments. Influences of diameter of the swirl chamber, length of the swirl chamber, converging angle between the swirl chamber were analyzed. As a result, peak frequencies were found which varies by the swirl chamber geometry of the closed-type swirl injector.

Keywords: Closed-type Swirl Injector, Liquid Film Thickness, Dynamic Characteristics, Swirl Chamber, Orifice

*Corresponding author: ybyoon@snu.ac.kr

Breakup dynamics of liquid sheet in coaxial swirling gas flow field

Kuppuraj Rajamanickam, Vishnu Goodwill, and Saptarshi Basu*

Department of Mechanical Engineering

Indian Institute of Science

Bangalore 560012, India

Abstract

In this paper, the near field breakup and atomization of the liquid sheet in the co-annular swirling gas flow field is presented. High-speed shadowgraphy is employed to characterize the global features and breakup dynamics of liquid sheet across a wide range of airflow rates. The parameter momentum ratio (MR) is used to illustrate the results. Breakup regimes are constructed based on the instantaneous visualization of the liquid sheet. At high airflow rates, the breakup process is shown to be a function of gas phase dominant. Theoretical expressions have been proposed for breakup length scales, the results show good agreement with experimental values (beyond critical MR values).

Keywords: coaxial atomization, swirl flows, high-speed imaging, near field breakup.

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

A study on the flow characteristics of bi-swirl injectors for a thrust chamber

W. Yoon, B. Lee, and K. Ahn^{*}
School of Mechanical Engineering
Chungbuk National University
Chungbuk 28644, Korea

Abstract

The bi-swirl injector, which is combined by two single swirl injectors, is a special injector for a liquid rocket engine. The spray characteristics of a swirl injector is very important due to the combustion efficiency and stable operation of a rocket engine. In this study, the bi-swirl coaxial injector, which has different recess lengths and swirl direction, was designed and manufactured. The discharge coefficients of the bi-swirl coaxial injectors were investigated according to the injection mode (single injection and bi-injection), different recess length, and swirl direction. From the experimental data, it was found that the discharge coefficient was almost constant independent of the pressure drop in single-injection. However, the discharge coefficient of the outer swirl injector in bi-injection decreased up to 10% as the mixture ratio increased under tip mixing and internal mixing. Due to the collision of the inner flow on the outer flow in bi-injection, a portion of the momentum from the inner flow was thought to block the outer flow to discharge through the outer injector. The swirl flow direction did not significantly affect the discharge coefficient in both single-injection and bi-injection.

Keywords: Discharge coefficient, thrust chamber, swirl coaxial injector, recess length, co-swirl, counter-swirl

^{*}Corresponding author: kbahn@cbnu.ac.kr

Spray Dynamics in a Swirling Jet Air-blast Atomizer

Srihari Dinesh Kumar Juvva, S. Mariappan, and A. Kushari*
Department of Aerospace Engineering
Indian Institute of Technology Kanpur
Kanpur 208016, India

Abstract

In this study, some features of the spray generated by a swirling jet air-blast atomizer, investigated in an atmospheric test rig, are reported. In this atomizer, liquid was supplied through radial holes drilled into the hub of the atomizer. The liquid then encounters the swirling airflow in an annular passage. The swirling was created by supplying the air through tangential ports into the annular passage. Liquid and air jets interact to form atomized liquid droplets. The features of the ensuing spray were studied using high-speed laser shadowgraph and laser Mie scattering. The images were correlated with the measured droplet size and velocity data, 38 mm downstream of the atomizer exit, using a PDPA system. The spray images showed a transition from a hollow cone spray to a partially solid cone spray as the ALR (air-liquid ratio) was increased. At an ALR of 1.5, the centre region of the spray had negative velocity suggesting the droplets were moving towards the centre of the spray. This can be attributed to the presence of a strong vortex core in that region. This resulted in increased shear induced atomization, which leads to the conversion from hollow to solid cone spray. Furthermore, tracking of the spray path in the high speed images show that at lower ALR's the spray follows a helical path, which becomes much steeper at increased ALR, suggesting the formation of the central vortex core.

Keywords: Spray, atomization, measurement, drop size, drop velocity, imaging diagnostics

* Corresponding author; Email: akushari@iitk.ac.in, Ph.: +91-5122597126

Image Processing

Prediction of windward trajectory of liquid jet in cross flow using image processing technique

G Muthuselvan^{1*}, P Suriyanarayanan², Jeyaseelan AR³, Rajeshwari N⁴,
Muralidhara HS⁵, L Venkatakrishnan⁶

^{1, 3, 4, 5} Combustion Laboratory, Propulsion division,

^{2, 6} Experimental Aerodynamics Division

CSIR-NAL, Bangalore

Abstract

The main objective of the present experimental work is to predict the windward trajectory of liquid jet injected into cross flow air stream. Ten different test conditions are simulated to cover wide range of Weber number (100 – 400) and momentum flux ratios (25 – 300). Air is passed through rectangular duct with cross section of 200x120 mm and Jet-A liquid fuel is injected crosswise from 0.5 mm circular orifice. The fuel pipe was located at a distance of 25mm from bottom surface of test section to avoid the influence of boundary layer. Pulsed arc light with 750ns exposure and slit dimension of 1 mm is used as light source. The shadowgraph set up with bending mirror is used at the light source and imaging end. A labview program in NI based instrumentation system is developed to monitor and maintain constant operating conditions of liquid and air medium. Closed loop feedback system is developed to maintain constant Weber number and momentum flux ratios for longer duration of experiments. To process shadowgraph images and predict windward trajectory of liquid jet, a Matlab program has been developed using image processing technique. Several instantaneous images are processed and average windward trajectory is predicted for each condition of Weber number and momentum flux ratio. With increase in momentum flux ratios, the breakup mode shift from column to mixed breakup mode. With increase in Weber number the breakup mode shift from column to shear breakup mode. At smaller momentum flux ratios, with increase in Weber number the overall penetration of windward trajectory decreases. But at higher momentum flux ratios, with increase in Weber number the overall penetration of windward trajectory does not changes. Two different correlations in the form of power law are developed for windward trajectory of liquid jet according to range of momentum flux ratios. Better matching between developed correlation and experimental data is observed.

Keywords: Liquid jet in cross flow, shadowgraph technique, image processing technique

-
- Corresponding Author
mechmuthu1@nal.res.in
Paper number 78

Development of Algorithm for Determining Particle Size in PDIA

Tryambak Gangopadhyay¹, R. V. Ravikrishna² and Achintya Mukhopadhyay^{1*}

¹Department of Mechanical Engineering, Jadavpur University, Kolkata, India

²Department of Mechanical Engineering, Indian Institute of Science Bangalore, Karnataka 560012, India

Abstract

Droplet size measurement finds important applications in engine industry and agriculture studies among others. An algorithm is being developed to determine the particle size in Particle/Droplet Image Analysis. Binary image is first formed using a threshold value to detect the particles and to trace the boundary of each of the particles. Detection of optimum number is important to increase the processing speed and decrease unnecessary selection, without neglecting any particles. An appropriate method is incorporated to achieve this by calculating optimum threshold level for initial detection. For each droplet, image intensity profile along the diameter is used to calculate two relative luminance values (considering luminance of perfect white as 1) at different percentages of the difference between the maximum and minimum intensities. The dependence of halo area at the edge of the droplet is studied for different defocus distances and different thresholds. The algorithm calculates two values of equivalent diameters at two threshold levels. The average of these two values is considered the final equivalent diameter. The use of two threshold levels minimizes the error, mainly for objects with more defocus. The accuracy of this technique is verified using calibration plate images comprising of particles ranging from 10 to 200 microns. The defocus distance range studied in this work is in the range of -320 to 320 microns. The results when compared with actual sizes show high accuracy, around 90% for 10 and 20 micron droplet. The accuracy is more for bigger particles, around 98% for 100 and 200 micron particles. Suitable user dependent factor is introduced for rejection or acceptance of defocused droplets. The algorithm also performs well for very dense sprays and can detect majority of the particles present. The droplet distribution obtained for a dense spray image confirms the robustness and accuracy of the algorithm.

Keywords: Droplet size, threshold level, defocus, optimum ratio, intensity profile, equivalent diameter

POD and DMD of circular liquid sheet originated from a simplified pendent type fire sprinkler

Chetankumar. S. Vegad*, Ramgopal Sampath, S. R. Chakravarthy, Amit Kumar
National Centre for Combustion Research and Development,
and Department of Aerospace Engineering,
Indian Institute of Technology Madras, Chennai 600 036 India

Abstract

The present work concerns the unstable behaviour of a circular liquid sheet produced by impingement of a water jet against a cone-disc deflector. The objective is to deduce the radial growth rate of the liquid sheet and identify critical conditions of the instability wave for sheet breakup. The liquid sheet is imaged by time-resolved planar laser induced fluorescence. Proper orthogonal decomposition (POD) of the spatial fields indicates ~75% of the energy content to be in the first few modes. Temporal POD indicates the prevalence of 13, 33, and 60 Hz recurring dominantly in these modes. Time windows of image frames are chosen to confirm these frequencies and the growth rate of different features of the waves by dynamic mode decomposition (DMD). The results reveal that the upward lift of the entire liquid sheet occurs at 13 Hz. The convective velocity of the liquid sheet increases radially outward due to sheet thinning, which causes faster spatial and temporal growth of the radial wave at a frequency of 33 Hz, as observed from instantaneous images. Finally, the sheet is observed to break up at a frequency of 60 Hz, as revealed by the DMD of the chosen time window of frames.

Keywords: Liquid sheet, POD, DMD, PLIF

*Corresponding author: vegad.cst@smail.iitm.ac.in

Development and Validation of an Open-Source Program for Particle / Droplet Image Analysis

Babu, Aravindh R¹ and T.N.C., Anand ^{2*}

¹ Department of Mechanical Engineering
National Institute of Technology
Tiruchirappalli 620015, India

² Department of Mechanical Engineering
Indian Institute of Technology Madras
Chennai 600036, India

Abstract

The Particle/Droplet Image Analysis (PDIA) method is often used to find droplet sizes in sprays using backlit images of droplets acquired at high magnification provided by long distance microscopes. Since the field of view at high magnifications is small, several hundred or thousand images are often processed to obtain statistically significant results. Hence, the time for processing each image is an important parameter. In the present work, an open source code was developed for processing droplet images. The dual and multiple thresholding methodologies of PDIA were implemented in a C++ program and optimized using the open-source routines in the OpenCV Libraries. The program is provided with a graphical user interface using the open source platform Qt, and supports image processing capabilities such as noise filtering and reference image calculation, and file processing capabilities such as directory management and batch processing. Using images of filled circles with known radii and focal distance deviation, calibration curves were plotted to determine the degree of defocus of an image. The accuracy of the program was evaluated by sizing droplets from a mono-disperse droplet generator, and the program was observed to be both fast and accurate.

Keywords: Droplet sizing, depth-of-field, degree-of-defocus, threshold, particle/droplet image analysis, atomization, imaging diagnostics

*Corresponding author: anand@iitm.ac.in

Numerical Sprays - 2

Large Eddy Simulation of Particle Laden Flows

Sri Vallabha Deevi ^{1*}, Joseph Mathew ²

Laboratory for Turbulence Computations,
Department of Aerospace Engineering,
Indian Institute of Science,
Bangalore 560012, India.

Abstract

Large Eddy Simulations of particle laden flows at moderate Stokes number are carried out. Gas phase is computed using Explicit Filtering Large Eddy Simulation, which employs filtering of conserved quantities using a higher order compact filter as the sub-grid-scale (SGS) model. Particles are treated as point masses and are tracked using a Lagrangian approach. Two way coupling is used to handle interactions between the continuous phase and dispersed phase. Particle modeling is intentionally kept simple without accounting for the effect of carrier phase SGS fluctuations or inter particle collisions, to assess the ability of LES method, and quantify the effect of SGS modeling on particle behavior. Simulation of a recent experiment of particle laden jets at high mass loading of 40% showed close quantitative agreement of self similar behavior for both phases. Centerline decay of continuous phase velocity as well as particle concentration showed excellent match with experimental measurements. Further studies revealed that particle behavior is strongly affected by particle size, and SGS modeling for lateral migration becomes important as particle size decreases.

Keywords: particle laden flow, Large Eddy Simulation, Stokes number, SGS model, explicit filtering method

*Corresponding author: srivallabha.deevi@gmail.com

Computational study of liquid jet breakup in swirling cross flow

Surya Prakash R¹, Suhas Jain S², Gaurav Tomar^{2*}, R V Ravikrishna^{2†}, B N Raghunandan^{1‡}

¹ Department of Aerospace Engineering

² Department of Mechanical Engineering

Indian Institute of Science

Bangalore 560012, India

25 June 2016

Abstract

Liquid jets in cross flow find wide applications in engineering fields, especially in the aerospace domain – combustors of gas turbine engines. The current study involves Volume-Of-Fluid based numerical simulations of a liquid jet injected into gas-crossflow in swirling motion. The liquid is injected radially outwards from a central tube to a confined annular space with gas crossflow. Simulations are conducted at high liquid-to-gas density ratio (Dr) of 180:1 and different liquid-to-gas momentum ratios (Q) upto 25. The Swirl Number (SN) of the gas crossflow is varied between 0 and 0.84. The liquid jet undergoes column and shear modes of breakup as observed in experimental studies. Spray characteristics such as drop size distribution, velocity of the drops, shape factor and column breakup length are analysed. Simulations effectively capture the influence of swirl flow on the spray characteristics.

Keywords: Atomization and Sprays, Volume of Fluid, Liquid Jet in Cross Flow, Shape factor, Drop size & velocity, Column breakup length

*gtom@mecheng.iisc.ernet.in

†ravikris@mecheng.iisc.ernet.in

‡raghubn@aero.iisc.ernet.in

A glance at Omega-Y and VOF-DPM hybrid spray models using studies to demonstrate their industrial applicability

V. Kumar^{1*}, M. Braun², and J. Schuetze²

1 ANSYS India, Plot No. 34/1, Rajiv Gandhi InfoTech park, Pune, India, 411057

2 ANSYS Germany GmbH, Birkenweg 14a, D-64295 Darmstadt, Germany

Abstract

Spray atomization, involving a complex gas-liquid flow near the nozzle as well as droplet breakup, fuel evaporation and fuel vapor-air mixing, is of utmost importance in determining the combustion efficiency and emission production in internal combustion engines and gas turbine combustors. Being a very important part of this, primary atomization is also analyzed in studies of propulsion, power and many other process applications. Computational Fluid Dynamics (CFD) has become one of the most important tools for both understanding spray development and improving spray characteristics. However, complex interactions between the physical phenomena involved still make it a challenge to be modeled.

This paper presents an overview of some of the common approaches for modeling sprays, namely the full Lagrangian approach, the phenomenological approach of the Ω -Y model (a variant of the Σ -Y model) and the more expensive hybrid approach using VOF-Lagrangian coupling. Simulation results obtained using ANSYS Fluent version R17.0 are presented pertaining to the Ω -Y model and the VOF-Lagrangian hybrid model, and their pros and cons are discussed in detail. While none of the models has a clear advantage over other models in all respects, the approach using the Σ -Y model seems to strike some balance between complexity and accuracy for industrial applications. On the other hand, with overhead on computational requirement, VOF-Lagrangian approach can be used to get a more detailed simulation of the phase interface dynamics enabling accurate visualization of spray structure and spray characteristics.

Keywords: Spray, non-evaporating, primary atomization, drop size, CFD, Σ -Y, Ω -Y, VOF, interface tracking, DPM, Lagrangian tracking

*Corresponding author: vivek.kumar@ansys.com

Simulating mechanical fuel injection system to improve its capabilities

N. Balasubramanian^{1*}, U. Vivekkumar², and S. Jayabalan³

Stanadyne India Private Ltd
Thiruvallur 602025, India

Abstract

This paper presents the simulation work carried out, on a mechanical fuel injection system, applicable for a single cylinder diesel engine. Developing countries are catching up with stringent emission norms, and the automobile industry is handling the challenges using more electronics. In this process, the conventional fuel injection systems are being phased out, owing to their limitations in the control of injection quantity, pressure and timing. Indian automobile industry is generally cost driven, and the three wheeler segment is more specific about the cost. Close to one million three-wheelers are produced in India annually, and they are largely powered by single cylinder diesel engines, using mechanical unit pumps and injectors. To cater this cost driven market, it is important to explore the possibilities of extending the usage of mechanical fuel injection systems, to meet stringent emissions, at increased injection pressures. Therefore a 1-D hydraulic simulation is carried out in this research work, to optimize the performance of the fuel injection system, in order to meet the engine requirements of the three wheeler segment. The simulation brings out the effect of different parameters, of various components used in the fuel injection system on its delivery performance. The results of the fuel injection system model built is appropriately validated with the experimental results.

Keywords: Fuel injection, mechanical injection system, unit pump, injection simulation

*Corresponding author: nbalasubramanian@stanadyne.com

Numerical Simulation of Transport of Polydisperse Polykinetic Spray in Spatio-temporally Fluctuating Crossflow using Quadrature Method of Moments

Sudeepa Mondal^{1, 1}, Aravindhan Ramesh^{2, 2}, Achintya Mukhopadhyay^{1, 3}

¹ Department of Mechanical Engineering, Jadavpur University, Kolkata – 700 032, India

² Department of Mechanical Engineering, National Institute of Technology Karnataka, Surathkal, India

Abstract

Transport of a polydisperse polykinetic spray injected in a gaseous crossflow has been simulated taking into account the size and velocity distribution of the droplets in the spray by using Quadrature Method of Moments with a relaxation approach to address the correlation between droplet size and velocity. A spatio-temporal fluctuation has been imposed on a constant gaseous flow as a surrogate for thermoacoustic oscillations encountered in spray combustors. The results show significant size-based segregation and clustering of droplets in response to fluctuations in the gas flow. To reduce the effect of these oscillations on the variation of droplet mass density over the time period of oscillation, a case of pulsed injection has been considered, which can be used to control the variation to a certain extent.

Keywords: Polydisperse sprays, method of moments, crossflow, control, fluctuations

A Rudimentary Model for Breakup Length in Effervescent Sprays

Hrishikesh Gadgil^{*1}, B. N. Raghunandan²

¹Department of Aerospace Engineering, Indian Institute of Technology Bombay, Mumbai 400076

²Department of Aerospace Engineering, Indian Institute of Science, Bangalore 560012

Abstract

Effervescent atomization has been a topic of interest in recent years for combustor application because of its superior qualities over other twin-fluid atomization techniques. The primary breakup in effervescent jets occurs because of rapidly expanding gas bubbles particularly at low gas-to-liquid ratios. This paper presents a reduced order model for predicting the breakup length in effervescent jets which is of practical interest to designers. In this novel approach, expansion of a gas bubble trapped inside the liquid jet is modeled by suitably modifying Rayleigh's equation that was originally derived for an expanding bubble in a vast pool of quiescent liquid. Order of magnitude analysis is used to predict influence of certain terms in the equation. This highly non-linear differential equation is solved numerically and temporal growth of bubble radius is obtained. The breakup event, termed as bubble bursting, is detected when the bubble radius exceeds that of a liquid jet. Flow visualization of near orifice spray is carried out to obtain the mean bubble bursting distance from the images. The values of breakup length predicted by the model are found to agree well with the mean values of experimental data. The comparisons carried out at different flow conditions are also included.

Keywords: Effervescent atomization, breakup length, GLR

^{*}Corresponding author: gadgil@aero.iitb.ac.in

Miscellaneous Topics 1

Microscopic Observation of Fuel Spray Injected into Cross-Flow and Impinged on Flat Wall

Min Guo^{a,b}, Zhanbo Si^a, Nagisa Shimasaki^a, Keiya Nishida^{a,*}, Youichi Ogata^a, Julien Manin^c, and Chenglong Tang^d

^a Department of Mechanical System Engineering, University of Hiroshima, Higashi-Hiroshima 739-8527, Japan

^b School of Mechanical and Electronic Engineering, Wuhan University of Technology, Wuhan 430070, PR China

^c Sandia National Laboratories, 7011 East Ave, Livermore 94550, CA, United States

^d State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, PR China

Abstract

Spray-wall impingement characteristics, as well as the interaction between spray and a cross-flow have fundamental effects on the combustion process and exhaust emissions in a direct-injection (DI) gasoline engine. In order to investigate the cross-flow effect on characteristics of the impinged spray and near-field spray, an experimental study was conducted to analyze the microscopic behaviors of fuel spray under various cross-flow velocities and ambient pressures. The spray is generated by a valve covered orifice (VCO) injector. Macroscopic spray images were obtained by a high speed video camera. Using a high speed video camera combined with a long-distance microscope, the impingement region spray and near-field spray were measured. Results show that the spray tip penetration increases significantly when the cross-flow velocity is further increased. However, ambient pressure has a slight inhibition effect on the penetration. A moment of droplets velocity reversion is defined under cross-flow conditions. With the increase of cross-flow velocity and ambient pressure, the time of droplets velocity reversion is decreased. For near-field spray observation, droplets cloud stripping from main spray body have been observed at the start of injection (SOI) under cross-flow condition. Additionally, at the end of injection (EOI), increasing cross-flow velocity tend to form tinier droplets. This reveals that the cross-flow is beneficial to the fuel spray breakup.

Keywords: Fuel spray, cross-flow, wall-impingement, internal combustion engine

* Corresponding author: nishida@mec.hiroshima-u.ac.jp

Experimental investigation on flow characteristics of diesel spray impinging on flat wall

Y. Zama^{1*}, Y. Odawara², and T. Furuhashi³
Faculty of Mechanical Science and Technology
Gunma University
1-5-1, Tenjin-cho, Kiryu, Gunma, 376-8515 Japan

Abstract

Recently, a diesel engine has been downsized due to improvement of fuel economy. Injection pressure of the diesel engine tends to increase due to promotion of fuel atomization. Therefore, the diesel spray impinges to a cavity wall in a combustion chamber of recent diesel engines. Moreover, heat loss from the engine occurred due to the heat transfer between the impingement spray and the cavity wall. Therefore, it is necessary to understand the impingement phenomenon of the diesel spray for improvement of the thermal efficiency. In order to understand the impingement mechanism between the spray and the wall, velocity measurement of an impingement diesel spray was needed. Moreover, the heat transfer strongly depends on the flow characteristics of impingement spray. Velocity information of the impingement spray might provide heat transfer mechanism between the spray and the wall. In this study, velocity field of an impingement diesel spray was measured with time-resolved PIV. Effect of ambient gas density, injection pressure and impingement angle on flow characteristics of the impingement diesel spray was investigated. Moreover, velocity of the diesel spray just before impingement to the wall was measured in order to investigate dependence of the impinging spray velocity on the post-impingement spray velocity. As the result, averaged radial peak velocities of the post-impingement diesel spray decreased with an increase of ambient gas density. Moreover, dependence of the impinging spray velocity on the averaged peak velocities of the post-impingement spray was clarified.

Keywords: Impingement diesel spray, Ambient gas density, Injection pressure, Impingement angle, PIV

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

Intensification of hydrogen production from methanol partial oxidation using sprays

Wei-Hsin Chen ^{1,*}, Chun-Ting Shen ²

1. Department of Aeronautics and Astronautics, National Cheng Kung University, Tainan 701, Taiwan
2. Department of Greenergy, National University of Tainan, Tainan 700, Taiwan

Abstract

Reaction characteristics of partial oxidation of methanol (POM) over a Pt/Al₂O₃ catalyst are investigated, with emphasis on the POM performance enhanced by sprays. The influences of the O₂-to-methanol molar ratio (the O₂/C ratio), the catalyst bed with/without preheating, and the preheating temperature on POM are also examined. Compared to liquid methanol dropped onto the catalyst bed, atomized methanol can be uniformly dispersed on the catalyst bed when sprays are applied. Therefore, the POM performance is enhanced drastically by sprays. With increasing the O₂/C ratio, POM progressively evolves into methanol combustion, and methane formation inherently changes from kinetically dominating mechanism to thermodynamically governing one. When sprays are practiced, the performance of POM under preheating is close to that under cold start, and an increase in preheating temperature intensifies the performance to a small extent. With the operation of sprays, the relative methanol conversion can be improved over 30%. Therefore, using sprays and cold start for POM over the Pt/Al₂O₃ catalyst is a feasible operation to produce syngas.

Keywords: Partial oxidation of methanol (POM); Sprays; Pt/Al₂O₃ catalyst; Cold start and preheating; Hydrogen and syngas production.

*Corresponding author: weihsinchen@gmail.com

Production of Medical Microspheres with Hybrid Atomization Processes for Embolization Applications

M. R. Wang^{*}, P. Y. Chen, P. Y. Ho

Department of Aeronautics and Astronautics, National Cheng Kung University, Taiwan

Abstract

In the fabrication of microspheres for medical application, monodispersed particle sizes are usually preferred as they give a more precisely controlled operation. Polycaprolactone and poly(lactic-co-glycolic acid) have obvious advantages over other polymers employed for biomedical applications due to their exceptional biocompatibility and biodegradability. This research program aims at the development of the medical device for the production of embolization microsphere and various potential medical applications. Hence a new process called hybrid atomization by combining the techniques of acoustic excitation, electrostatic atomization scheme was developed to produce the mono-dispersed microsphere for medical applications. The goal of this research is to produce the mono-dispersed microsphere with diameters ranging from 50 μm to 400 μm to meet the required specifications of the medical applications. Results show that the particle size can be controlled by the applied voltage, flow rate, excitation frequency, and material conductivity. Results also show that the new developed system can be used to produce the mono-dispersed microspheres in the size range between 60.82 μm to 488.9 μm of medical materials with high viscosity. These microspheres can be used to fulfill the requirements in various medical treatments.

Keywords: embolization microsphere, monodispersed particle, hybrid atomization, acoustic excitation, electrostatic atomization, medical materials

A Study on Atomization Process for Carbon Black Reactor

Madhavi Gupta¹, Raj Gopal Tripathi¹, Manoj Kandakure^{1*}, and Ranjan Ghosal²

¹Aditya Birla Science and Technology Co. Pvt. Ltd., Taloja, Navi Mumbai-410208, India

²Birla Carbon R&D, Taloja, Navi Mumbai-410208, India

Abstract

Carbon black (CB), a carbon nanoparticle, is industrially produced via furnace black process in a highly controlled environment to obtain CB particles of varied structure and surface chemistry. It is used as an additive in diverse applications (tire, rubber, plastics, printing inks, paints, etc.) to enhance their mechanical, electrical or optical properties. In this process, carbon black feedstock (CBFS) is injected through atomizers to achieve optimum atomization and distribution of the droplets into highly turbulent gases at high temperature. The quality of atomization and droplet distribution significantly affect vaporization of droplets, subsequent gaseous pyrolysis and ultimately carbon black particle formation. The present work aims to study the atomization process through experimental characterization and numerical simulations of atomizers. Atomizers have been characterized for droplet size and velocities using high speed imaging technique. The measurements have been found to be in good agreement with the calculated values from correlations published in the literature. This data has been used as an input to CFD model which captures detailed gas phase pyrolysis chemistry with droplet breakup and evaporation. CFD simulations have been performed to study the effect of various atomizer parameters and process conditions on droplet penetration and residence time. The present study will help in developing a deeper understanding and subsequent optimization of the atomization process.

Keywords: High speed imaging, Computational Fluid Dynamics (CFD), atomization, droplet breakup, evaporation, penetration, carbon black process

*Corresponding author: manoj.kandakure@adityabirla.com

Jet characteristics of high pressure gas under argon atmosphere

Yinchun GONG¹, Xuehai GONG¹, Jun DENG¹, Liguang LI^{1,2*}

(1.School of Automotive Studies, Tongji University, Shanghai 200092, China

2. CDHK, Tongji University, Shanghai 200092, China)

Abstract

In this paper, the effects of argon atmosphere on the jet characteristics of high pressure gas with different densities were presented. The experiment was carried out in a constant volume vessel using high speed photograph with the schlieren photograph method. The jet development process and jet characteristics of the methane and the oxygen under different injection conditions in the argon atmosphere were analyzed. The results show that the jet penetration increases with the increase of injection pressure and decreases with the increase of the ambient pressure in the argon atmosphere. Under the same injection pressure and ambient pressure, the jet penetration of oxygen is greater than that of methane jet penetration, and the jet dispersion angle of oxygen is less than that of methane. The penetration of methane in the argon atmosphere is less than that of methane in nitrogen and the jet dispersion of methane in argon atmosphere is greater than that of methane in nitrogen.

keywords Jet Characteristics, Argon, Methane, Oxygen, High pressure injection

* Corresponding author: liguang@tongji.edu.cn

Miscellaneous Topics 2

Suppression of Obstructed Fire using Water Mist Generated by a Twin-Fluid Nozzle

Suresh Lal

Center for Fire Explosive & Environment safety
DRDO, New Delhi, India

Abhijit Kushari*

Indian Institute of Technology Kanpur, Kanpur, India.

Abstract

The two protocols on environment, i.e., Montreal Protocol and Kyoto Protocol have restricted the use of halons for fire suppression applications due to their adverse impact on the ozone layer. Water mist based systems have emerged as a viable alternative to replace halons for firefighting. This paper reports the use of water mist for the suppression of obstructed fires that cannot be extinguished by a conventional sprinkle system. Due to their small size, the droplets in the mist could be convected to the fire as they get entrained in the air stream around the fire and successfully suppressed the fire, without direct impact, at a reasonably short period of time. A twin-fluid mist generator was developed and used for this purpose. The effect of the atomizing gas on the efficacy of the fire suppression system was experimentally established that showed that the use of an inert gas like nitrogen for atomizing water is more efficient in the suppression of fire.

Keywords: Fire suppression, obstructed fire, water mist, effect of atomizing gas

*Corresponding author: akushari@iitk.ac.in, Ph. - +91-512-2597126

Effect of Side-Tube on the Agglomeration of Carbon Dioxide Snow

T.C. Lin^{1*}, S.H. Liu², Y.H. Li², M.R. Wang²

¹ Research and Services Headquarters, National Cheng Kung University, Tainan, Taiwan

² Department of Aeronautics and Astronautics, National Cheng Kung University, Tainan, Taiwan

Abstract

Carbon dioxide (CO₂) snow jets are widely used in medical and industrial applications, such as cryogenic therapy, food processing, pharmaceutical granulation, particle removal and surface cleaning. In semiconductor fabrication, CO₂ snow cleaning has unique organic and particle removal abilities with eco-friendly that are better than those of traditional surface cleaning methods. This paper experimentally investigates the side-tube effect on agglomeration of CO₂ snow inside a tube. By using the additive manufacturing method by 3D printer, complicated design with converging-diverging configuration and curved surface can easily be realized. Results show that a complicated particle motion in the upper portion of the tube is responsible for the formation of large snow. With the side-tube, a second recirculation occurs. It may improve or decrease the effect of agglomeration according to the setting position of side-tube. In this research, the snow size can be increase by 176%, and a maximum snow transformation rate of 31.9% has been achieved in this paper, which is 47% improvement compare with the case without side-tube (21.8%).

Keywords: Spray, particle size, CO₂ snow, agglomeration, recirculation, side-tube

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

Characteristics of zirconia-ethanol suspension spray with thin coaxial twin-fluid atomizer for suspension plasma spraying

Norihiko IKI *, Masato SUZUKI, Mohammed SHAHIEN and Takashi OKADA
Energy Conservation Research Institute,
National Institute of Advanced Industrial Science and Technology (AIST)
Ibaraki 3058564, Japan

Abstract

Thermal barrier coating (TBC) is necessary to increase turbine inlet temperature of a high efficient gas turbine for a power station. Suspension plasma spraying is expected to have higher flexibility in particle size of processed material than conventional plasma spraying. Therefore microstructural feature of coating can be varied by the parameters of suspension plasma spraying, such as molten droplets of ceramic material. AIST is trying to develop an axial feeding type of suspension plasma gun with thin coaxial twin-fluid atomizer. Atomization of ethanol suspension of solid particles is one of the important factors controlling the coating quality. Therefore the spray of zirconia-ethanol suspension from the thin coaxial twin fluid atomizer with new p-torch was investigated comparing with ethanol spray, zirconia-water spray and water spray. Sauter mean diameter of the zirconia aggregates derived from zirconia-ethanol suspension are smaller and more stable than those of zirconia-water suspension. Sauter mean diameter of zirconia aggregates are mainly affected by the mass flow rate of the atomizing air and the mass flow rate of the suspension than zirconia concentration. Sauter mean diameter of the zirconia aggregates derived from zirconia-ethanol spray is smaller than those of zirconia-water spray. Sauter mean diameter of the zirconia aggregates derived from zirconia-ethanol spray can be controlled in the range of 1 to 3 microns by changing the mass flow rate of the suspension, the zirconia concentration and the mass flow rate of the atomizing air

Keywords: Twin Fluid Atomization, Suspension Plasma Spraying, Liquid Jet, Sauter Mean Diameter, Slurry, Ethanol, Water

*Corresponding author: n-iki@aist.go.jp

Effect of Acoustic excitation on the Spray Characteristics of a Hollow Cone Spray

I.B. Aravind*, S. R. Chakravarthy
Department of Aerospace Engineering
Indian Institute of Technology Madras
Chennai 600036, India

Abstract

The paper reports an experimental investigation on the effect of external excitation on hollow cone spray from a pressure-swirl nozzle. The hollow cone spray is subjected to external excitation at different frequencies with amplitude kept constant. High speed images acquired are processed to extract macroscopic spray parameters like spray cone angle, spray breakup length and spray sheet width. The investigation reveals that acoustic excitation alters the spray structure by reducing the cone angle. The phase averaged spray cone angle fluctuates for different phase angle within an acoustic cycle. The cone angle fluctuation decreases with increasing excitation frequencies. The spray boundary responds to the acoustic excitation by oscillating at the frequency of excitation. The acoustic excitation at different frequencies reduces the breakup length. The sheet oscillation amplitude decreases for higher excitation frequencies. It is shown that the spray responds more readily to excitation at low frequencies.

Keywords: Hollow cone Spray, high speed shadowgraphy, acoustic excitation, spray characteristics

*

*Corresponding author: aravindib1992@gmail.com

Electrostatic Application to Ethanol Based Fuel Injected through Multiple Nozzles

J. C. González Palencia, Y. Watanabe, A. Onozato, M. Araki, S. Shiga*

Division of Mechanical Science and Technology

Gunma University

Gunma 376-8515, Japan

Abstract

Ethanol spray formation using electrostatic atomization is studied. In the experimental set-up used, ethanol flows between a nozzle and a ring with flow rates between 0.5 and 3.3 ml/min; corresponding to the fuel flow rate range in a gasoline engine under starting condition. Atomization occurs due to the application of a high voltage difference between the nozzle and the ring. The effect of the number of nozzles was investigated considering two configurations: single-nozzle and multiple-nozzle. Additionally, the effect of applied voltage and flow rate were studied. Experiments were performed using denatured ethanol. Two copper ring electrodes were used, small ring, inner diameter of 19 mm and outer diameter of 27 mm; and large ring, inner diameter of 65 mm and outer diameter of 73 mm. Droplet diameter was measured using a 1D PDA (Particle Dynamics Analysis) system. Results show that cone-jet mode atomization is realized for both single-nozzle and multiple-nozzle configurations. SMD (Sauter Mean Diameter) was found to decrease with the increase in the applied voltage; though results are not conclusive due to the wide dispersion of the spray that makes difficult a reliable measurement. The smallest SMD found for the multiple-nozzle configuration was 6 μm ; comparable to commercial nebulizers that operate based on ultrasonic vibration using micro-nozzle arrays.

Keywords: Electrostatic atomization, multiple-nozzle, Sauter Mean Diameter (SMD), flow rate, ethanol

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

Experimental Investigation of Melt Delivery Tube Position on Metal Atomization

W.S.Prashanth¹, T.R. Sabin¹, S.Sarkar², T.N.C. Anand¹, S. Bakshi^{1*}

¹Dept. of Mechanical Engineering, IIT Madras, India

² Sandvik Asia Pvt. Ltd., Pune, India

Abstract

Metal atomization is the process of breaking up of a liquid metal jet into fine droplets. The molten metal is atomized by a gas stream to create droplets which eventually solidify to form metal powder. Based on the interaction between the metal and gas, there are two types of nozzle arrangements viz. free-fall and close-coupled nozzles. The relative position between the melt delivery tube and the gas nozzle determines whether aspiration (suction) or pressurization (back pressure) takes place. In this study, dynamic pressure measurements are done at the tip of melt delivery tube at gas only trials for different positions and at different gas flow rates, to determine the type of nozzle arrangement.

Keywords: Metal atomization, Nozzle, Melt delivery tube

*Corresponding Author: shamit@iitm.ac.in

Wednesday

9th November 2016

Plenary Lecture 3

Simulating Atomization in Complex Injector Geometries

Marcus Herrmann

School for Engineering of Matter, Transport and Energy
Arizona State University

Abstract

Significant progress has been made in the past decade to predict atomization using detailed numerical simulations. Much of the early work in this area focused on atomization in simple geometries, for example liquid jets issuing from straight pipes into rectangular channels with and without cross flows, e.g., the atomization of liquid jets in cross flow, and the injection of liquids jets into stagnant environments typical for Diesel injectors. However, in the past couple of years, simulations of realistic complex injectors have become viable, offering a path to study application relevant geometries not only via experiments, but also via detailed simulations.

In this presentation, different numerical techniques will be discussed to enable the predictive simulation of atomization in complex geometries. These include conservative techniques to capture the motion of the phase interface, unstructured meshing approaches to handle complex geometries, and approaches to tackle the vast range of scales present in atomization processes, ranging from centimeter scales of the injector down to the sub-micron scale of the smallest drops. In addition to validation results for simpler injector geometries, simulation results for a realistic high-shear gas turbine injector consisting of multiple liquid jets injecting into a swirling cross flow will be presented and compared to available experimental data for the configuration. The results highlight the current capabilities to predict atomization in realistic configurations, but also show the need for continued improvement of the numerical techniques to simulate atomization.

Spray Characterization - 3

Experimental Studies on Evaporating Sprays of Diesel & Jatropa Methyl Ester (JME)

Prasad Boggavarapu* and R. V. Ravikrishna

Combustion and Spray Laboratory

Department of Mechanical Engineering

Indian Institute of Science

Bangalore 560012, India

Abstract

In the present study, the spray structure of Jatropa methyl ester (JME) and diesel are compared under evaporating conditions. The evaporating spray characteristics are described in terms of liquid penetration, vapour penetration, liquid length and spread angle. A high-pressure chamber with optical access is used in the present study for spray characterization. The chamber is filled with nitrogen which is heated to reach a final pressure of 50-bar and a final temperature of 900-K. The spray characteristics are studying by injecting diesel through a custom-made single-hole nozzle into the high-pressure and high-temperature nitrogen gas. The shadowgraphy technique is used to record the liquid phase and vapour phase images which are then analyzed to obtain the spray characteristics. The results showed that the liquid length of JME is longer by 19%, 17% and 11% at injection pressures of 500, 1000 and 1500 bar, respectively. At a given operating condition, the vapour penetration of JME and diesel are similar due to the same initial momentum of both the sprays. A diesel-ethanol emulsion (60% diesel, 20% ethanol) along with 20% JME is also studied in terms of its evaporating spray structure. It is observed that the liquid penetration and the vapour penetration of the blend match closely with that of diesel. An analytical model for vapour penetration has been verified against the experimental data and found to match well with the data. A parametric variation of fuel density showed that the effect of fuel density variation is compensated by the corresponding variation in injection velocity for a given injection pressure. This resulted in a similar vapour penetration for all fuels at that condition, thus explaining the experimentally-observed trend.

Keywords: Evaporating sprays, Biodiesel, Jatropa methyl ester (JME), diesel spray, Shadowgraphy.

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

Experimental analysis on the swirl spray formation under low injection pressure

S. Sakatani*, A. Niwa, J. Kaniyu, and E. Matsumura

Department of Mechanical Engineering

Doshisha University

Miyakodani 1-3, Tatara, Kyotanabe, Kyoto, 610-0321 Japan

Abstract

Diesel Particulate Filter (DPF) is a very effective after-treatment device to limit particulate emissions from diesel engines. It is necessary to regenerate the DPF regularly. Injecting fuels into the exhaust line upstream of the Diesel Oxidation Catalyst (DOC), hydrocarbons are oxidized on the DOC, which increases the exhaust gas temperature at the DPF inlet, this results in regeneration of DPF. In burning the particulate matter efficiently, it is necessary that the fuel is completely vaporized and uniformly mixed with exhaust gas. However, the distance between the injector and the DOC is short. There is another problem that evaporation and dispersion of the spray deteriorates. Therefore, it is important to understand spray behavior in the tail pipe. This report focuses on free spray to understand its physical phenomena using sheet scattered light method and Super High Spatial Resolution Photography with swirl nozzle and n-dodecane. As a result, swirl spray is injected in a state with swirling force, liquid sheet grows to radial direction by centrifugal force, become a hollow cone shape. When liquid sheet spreads, inside of the spray became negative pressure. And flow field from the outside to the inside of the spray is formed, follow up spray is formed. As injection pressure increases, this follow up spray becomes bigger. Also liquid and ligament region in the condition that injection pressure is 1 MPa is bigger than 4 MPa. Moreover the ligament and hole occurs irregularly in the liquid sheet, and the liquid sheet breakup of the swirl spray has occurred in the center of the liquid sheet as well as the tip. Therefore, the breakup mechanism of swirl spray under low injection pressure is different from Fraser's theories and break up is irregular due to the instability of the liquid sheet.

Keywords: Swirl nozzle, Atomization, Experimental study

*dup0556@mail4.doshisha.ac.jp

Spray Characteristics of High Rotating Speed Slinger of Vertical Distribution

M. A. Khan^{1*}, Y.T.Chen² W. H. Lai³, C.H. Hong⁴

^{1, 2, 3} Department of Aeronautics and Astronautics, National Cheng Kung University
Tainan, Taiwan

⁴Research Center for Energy Technology and Strategy, National Cheng Kung University, Taiwan

⁴ National Chung Shan Institute of Science and Technology, Taichung, Taiwan.

Abstract

The development of smaller gas turbine engine preferred injection method of slinger system has been explored intensively. In the previous research mostly focus is given to investigate the characteristics of spray and the droplet size distribution. The research is mainly focused on one of the essential element which is inertia force the spray distribution influences spatially by that. The parameters like rotational speed, flow rate, orifice diameter and the vertical distance of droplet distribution are used to analyze our research phenomena. The research will try to get the result how the vertical distance and gravity force will responsible for change of droplet distribution. The results of this research will be compared by the previous work which is dependent on horizontal distance of droplet distribution in which no gravity force is acting. It will try to optimize the result with different flow rate and rotational speed. For analyzing the result some non-dimensional number like weber number, ohnesorge Number will be used to show the result.

Keywords: Spray, Slinger, atomization, measurement, drop size, imaging diagnostics

*Corresponding author: aqibkhn86@gmail.com

Investigation of High Pressure Diesel Spray Characteristics with Computational Fluid Dynamics Approach.

D. Siva Krishna Reddy*
Mahindra & Mahindra Ltd,
MRV, Chengalpattu, 603004.

Shamit Bakshi and T.N.C. Anand,
Department of Mechanical Engineering,
Indian Institute of Technology Madras, Chennai, 600036.

Abstract

Combustion initiated by the high pressure fuel sprays plays a critical role in the performance of the reciprocating compression ignition engines. The diffusive nature of combustion leads to formation of emissions such as soot and NO_x. Reducing these emissions and improving the performance of the engine requires a good understanding of the spray characteristics under engine operating conditions. Computational fluid dynamics (CFD) can be effectively used to study the spray dynamics. In the present work, the effect of ambient temperature and injection pressure on the sprays characteristics are studied using CFD. Towards this objective, flow fields around high pressure non-reacting diesel sprays are computed with commercial CFD code, AVL FIRE. Wave model is used to describe liquid secondary break-up. The computations are performed for ambient pressure of 45 bar and temperature of 300 K and 450 K. The injection pressure varied from 600 to 1350 bar. The wave model constants are fine tuned to match the measured spray penetration length. The computed flow fields are analyzed to understand the effect of injection pressure, ambient temperature on spray penetration length, turbulence generation. It is observed that increasing the injection pressure results in higher penetration length. An increase in the spray chamber temperature also found to increase spray penetration length.

*Corresponding author: krishnareddy.iitg@gmail.com

Effect of liquid properties on the oscillation of elliptical liquid jets

K.R. Rajesh^{†, #, *}, R. Sakthikumar[§], M.V. Narasimha Prasad[†] and D. Sivakumar^{#, §}

[†]Liquid Propulsion Systems Centre, ISRO, Bangalore 560008, India

[#]Department of Aerospace Engineering, Indian Institute of Science, Bangalore 560012, India

[§]Interdisciplinary Centre for Energy Research, Indian Institute of Science,
Bangalore 560012, India

Abstract

Understanding the interfacial oscillation characteristics of liquid jets discharging from orifices is essential to describe their breakup mechanism and subsequent spray formation. The present work deals with an experimental study on the interfacial oscillation of elliptical liquid jets. The experiments are conducted with an elliptical orifice of equivalent diameter 4.93 mm with major and minor axis lengths 7.12 mm and 3.45 mm respectively. The results are obtained for three different liquids: pure water, water-glycerol mixture, and Jet A-1 fuel. The liquid jets are visualized using backlight shadow-graphy technique involving a high intensity strobe lamp and a digital camera. The interfacial oscillation of liquid jets is characterized by measuring the wavelength and amplitude of jet oscillation from recorded digital images. The wavelength of jet oscillation (λ) of elliptical liquid jet increases linearly with increase in liquid jet velocity (U_o). The jets of Jet A-1, owing to its significantly lower surface tension compared to the other two liquids, display higher value of wavelength for a given U_o . The experimental measurements of λ recorded at different flow conditions exhibit good agreement with the theoretical predictions obtained using mathematical model proposed by Rayleigh (1879) to describe capillary phenomena of liquid jet exhibiting undulations in its cross section. For a given liquid jet Weber number (We), the elliptical jets of Jet A-1 undergo a larger increase in λ compared to the jets of other two liquids. Further the variation of maximum and minimum amplitudes (D_{max} and D_{min}) of jet oscillation of elliptical liquid jets with We is discussed.

Keywords: Elliptical liquid jets, interfacial oscillation, jet breakup, elliptical orifice

*Corresponding author: rajeshkr@aero.iisc.ernet.in; rajesh_kr@lpscb.gov.in

Identifying non-Markov effects in sprays

Bhaumik Shah Vimal ¹, Sri Vallabha Deevi ², Arun Tangirala ³, Mahesh V Panchagnula ⁴ *

Indian Institute of Technology Madras,
Chennai 600036, India.

Abstract

Phase Doppler Particle Analyzer data has typically been used to construct joint size-velocity distribution functions in sprays. In this approach, the arrival time data is not utilized. This study proposes analyzing the time-series data of droplet diameter and velocity measured simultaneously at a given point in space towards discovering inherent non-Markov effects in droplet transport. Previous studies have assumed a spray to be a Markov process i.e., the characteristics of an incoming droplet is affected only by the immediately preceding droplet. We use two classical measures of serial correlations, namely auto-correlation function (ACF) and partial auto-correlation function (PACF) to examine this claim. Using these measures, correlation of series with its own past is modeled. Analysis reveals significant 5th order temporal correlation for velocity and 1st order correlation for diameter. Presence of significant coefficients also makes a strong case for an inherent predictable process driving the droplet generation and transport. Linear time-series models like auto-regressive (AR) model and auto-regressive moving average (ARMA) model were constructed for both velocity and diameter. On further investigation of these models and using statistical tools such as bi-coherence, we show conclusive evidence of inherent non-linearity present in the process.

Keywords: time series analysis, Markov process, spray, phase Doppler particle analyzer, AR/MA/ARMA models, bi-coherence

*Corresponding author: mvp@iitm.ac.in

Atomization and Breakup

A Consideration on Cavitation and Flash Boiling Atomization

Masataka Arai^{1*}, Jiro Senda², and Min Xu³

¹ Dept. Mechanical Engineering, Tokyo Denki University
Senju-Asahi-cho 5, Adachi-ku, Tokyo, 120-8551 Japan

² Doshisha University, Kyoto Japan

³ Shanghai Jiao Tong University, Shanghai China

Abstract

It is well known that cavitation in diesel fuel injector and flash boiling in gasoline fuel injector are affecting atomization and spray formation processes. However there is few general review on the difference of cavitation and flash boiling in a fuel injector. Both cavitation and flash boiling bubbles affect liquid breakup and spray formation process, but these bubbles show far different behaviors and different effects through their destruction processes. Cavitation bubble in which vapor pressure is less than surrounding liquid phase, is shrunk and generates intense turbulence flow in its final stage of shrinking. It causes strong promotion for atomization but less contribution on spray distribution except spray angle. Flash boiling bubble in which vapor pressure is higher than surroundings, expands due to superheated liquid fuel and greatly promotes atomization through its bursting. Spray behavior with flash boiling is far different from non-flash boiling spray, and flash boiling has a potential to improve spatial distribution and evaporation process of fuel spray. In this brief consideration, above phenomena are explained by the previous results mainly obtained by authors, adding cavitation and flash boiling atomization model tentatively obtained here.

Keywords: Atomization, Fuel Injector, Cavitation, Flash Boiling, Fuel Spray, IC Engine

*Masataka Arai: marai@mail.dendai.ac.jp / arai@gunma-u.ac.jp

Atomization Characteristics of Rotary Bell Cup Atomizer with Shaping Air Flow

M. Daikoku^{*1}, S. Nyui², T. Soma², Y. Saito², Y. Matsushita², H. Aoki²,
M. Shiota³, T. Inamura³, S. Ogasawara⁴, and J. Fukuno⁴

¹Department of Mechanical Engineering, Hachinohe Institute of Technology,
Hachinohe 031-8501, Japan

²Department of Chemical Engineering and Biomolecular Engineering, Tohoku University,
Sendai 980-8579, Japan

³Department of Mechanical Science and Engineering, Hirosaki University,
Hirosaki 036-8561, Japan

⁴Research and Development Division, Honda Engineering Co., Ltd., Haga 321-3395, Japan

Abstract

This study investigates the atomization characteristics of the rotary bell cup atomizer used in automotive spray painting equipment. Classification of the breakup pattern at the bell cup edge, which is significantly related to that of the liquid flow pattern on the cup surface, revealed that several liquid ligaments were produced at low rotational speeds and fine ligaments which cause Rayleigh breakup were generated at high rotational speeds. The effect of shaping air on breakup of liquid ligament generated from the edge of rotary bell cup was also investigated; the breakup phenomena of liquid ligaments were simulated by using liquid column issued from a single hole nozzle, and it was found that breakup patterns were classified by using Weber number and Reynolds number of liquid column. The atomization characteristics of liquid column were examined, and conditions favoring the generation of ghosts which suppresses proper atomization were investigated. Also, droplet formation from the single hole nozzle was compared with that using a rotary bell cup atomizer.

Keywords: Atomization characteristics, rotary bell cup atomizer, breakup behavior, liquid column/ligament, shaping air

*Corresponding author: daikoku@hi-tech.ac.jp

Breakup of a Liquid Jet under Rigid Body Rotation

Chun-Chia Chang, Hsin-Wei Huang, Chun-Kuei Chen, Ta-Hui Lin
Department of Mechanical Engineering
National Cheng Kung University
Tainan 704007, Taiwan

Abstract

The breakup process of a liquid jet under rigid-body rotation was experimentally examined in this study. A liquid nozzle issued the liquid vertically downward, and was connected to a DC motor by a set of pulley. In the experiment, the flow rate ($\dot{Q}$) of the jet and the rotation speed (ω) of nozzle were varied. For the cases of $\omega = 0$ (no rotation), 500 rpm, and 1200 rpm, the breakup length of the liquid jet was almost linearly increased with increasing the flow rate. The breakup length became smaller when the nozzle had a larger rotation speed. With a fixed flow rate, the breakup length was decreased with increasing the rotation speed; and its variation was shown by two declined straight lines connected at $\omega = 300$ rpm, in which the slope for $\omega < 300$ rpm was larger than that for $\omega > 300$ rpm. It was noted that if the rotation speed was relatively low, the cylindrical liquid column was broken to create stable and uniform-size drops. On the other hand, when the rotation speed was high, the liquid column became more unstable and its breakup process resulted in three distinct drops patterns, such as steady drops, two-drop coalescence, and multiple-drop coalescence.

Keywords: Jet breakup, rotating jet, Rayleigh instability.

Numerical Investigation of Molten Fuel Jet Breakup inside a Liquid Pool

Tuhin Maitra¹, Priyankan Datta^{2,*}, Aranyak Chakravarty^{1,2}, Achintya Mukhopadhyay²,
Koushik Ghosh², Swarnendu Sen²

¹School of Nuclear Studies and Application, Jadavpur University, Kolkata, India

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

Abstract

In a hypothetical severe accident scenario of a nuclear reactor, the reactor core might undergo tremendous heating in absence of adequate heat removal mechanism and may form molten state. The molten core material (or corium) progresses towards the reactor pressure vessel lower head in a form of jet and interact with the cold water which may lead to the fragmentation of jet. This molten fuel jet fragmentation phenomenon has gained attention among the researchers due to its high uncertainty which may influence the further accident progression followed by steam explosion.

The present problem aims at the exploration of the underlying physics along with the effect of different initial conditions on hydrodynamic fragmentation of a molten fuel jet in a pool of liquid using ANSYS FLUENT 14.5 with a 2D configuration. The Volume of Fluid (VOF) method is used to capture the molten wood metal jet fragmentation phenomenon in liquid water. Identification of the influence of jet velocity, inlet profile of the jet and jet diameter on the fragmentation mechanism are the salient objective of the present work.

Keywords:

Fuel-coolant interactions, melt jet instability, melt jet fragmentation, jet breakup time

*Corresponding author: priyankan.datta43@gmail.com

Effect of Density Ratio on the Secondary Breakup of Spherical Drops in a Gas Flow

Suhas Jain S¹, Neha Tyagi¹, Gaurav Tomar^{1*}, Surya Prakash R¹ and R. V. Ravikrishna^{1†}

¹ Department of Mechanical Engineering
Indian Institute of Science
Bangalore 560012, India

September 2016

Abstract

Breakup of drops is ubiquitous in nature. In the present study, fully resolved Volume of Fluid (VOF) simulations are performed, using an open source solver – Gerris, to understand the physics involved in the breakup of a single spherical droplet. Most of the previous numerical studies involve investigation of the breakup of drops at low-density ratios (drop liquid to ambient fluid ratio $\rho^* < 100$). It was observed that the backward bag breakup is the predominant mode in this density ratio range at moderate Weber numbers (20-80). In our previous study [1], a forward-bag breakup at a density ratio of 1000 was observed for moderate Weber numbers. In the current work, we study the breakup at density ratios of 10, 50, 100, 150, 200, 250, 500 and 1000 to understand this dependence of breakup mode on the density ratio. We found that there exists a critical density ratio above which the dynamics of drop breakup is significantly different from that at low density ratios. We study the role of the dynamics of the droplet rim, at different density ratios, in the bag formation process. Finally, we present a $\rho^* - We$ phase plot reflecting the drop behavior at various density ratios and Weber numbers.

Keywords: Secondary breakup, Volume of Fluid, Density ratio

*gtom@mecheng.iisc.ernet.in

†ravikris@mecheng.iisc.ernet.in

Studies on Secondary Atomization of Molten Gallium Droplets at Nearly Critical Weber Numbers

S. Joshi¹, T.N.C. Anand^{1*}

¹Department of Mechanical Engineering and National Centre for Combustion Research and Development, Indian Institute of Technology Madras, India

Abstract

There are several studies in literature on secondary atomization of Newtonian liquids such as n-heptane, water, ethyl alcohol and glycerol solutions, and ‘drop deformation and breakup regime maps’ have been plotted based on the studies. However, high surface tension liquids have not been studied in detail. The present research aims at studying secondary atomization of liquids having high surface tension values. One of the liquids that is studied here is molten gallium. A pneumatic on-demand droplet generator equipped with a heater produces mono-sized droplets of molten gallium and their breakup due to cross-flow is captured using diffused backlit shadow imaging. The continuous jet method is employed in this study. The Weber number is varied by changing the velocity of air jet and it is kept near the critical Weber number. Results show the bag and multimode breakup regimes for high surface tension liquids.

Keywords: Secondary atomization, backlit imaging, cross-flow, Weber number, mono-sized droplet

* Corresponding author: anand@iitm.ac.in

Miscellaneous Topics 3

Numerical Analysis of Pre- Detonation Chamber Properties with Varying Mass Flow Rates Using Hydrogen/Air Mixture and its Effects in Pulse Detonation Engine.

C T Dheeraj Kumar Singh*, Dr Gurunadh Velidi

*M. Tech in Computational Fluid Dynamics

Assistant Professor, Department of Aerospace Engineering

University of Petroleum and Energy Studies

Dehradun 248007, India

Abstract

Pulse detonation engines are the modern propulsive device which provides high thrust. They are unsteady propulsive devices which have multi cycle operations, every cycle involves fuel and air mixing and a shock wave generation in combustion chamber. Combustion chamber is maintained at high pressure and high temperature which leads to combustion of Hydrogen. Deflagration propagates forward by taking all unburned species and products formed after combustion. Propagation of Deflagration – Detonation Transition (DDT) shock wave studies are interesting since creation stable combustion and shock propagation in steady state is challenging. In this work, Numerical Studies are conducted with shchelkin spiral geometry of different diameters of 3cm and 2cm are considered with different mass flowrates at inlets, in which one is for fuel inlet and other for oxidizer. Hydrogen along with oxidizer mixture enters the Detonation zone with pre designed time delay. In this work we have considered different mass flow rates of fuel and oxidizer with the objective of meeting effective equivalence ratios. This study presents effects of variable mass flow rate on pre detonation zone conditions, detonation wave propagation and shockwave behavior.

Keywords: Pulse Detonation Engine (PDE), Deflagration Detonation Transition (DDT), Shchelkin spiral.

*Corresponding author: dheerajsingh14@stu.upes.ac.in

INFLUENCE OF SWIRL RATIO ON FUEL DISTRIBUTION AND CYCLIC VARIATION UNDER FLASH BOILING CONDITIONS

Jie Yang¹, Min Xu,^{1*} David L.S. Hung^{1,2}, Qiang Wu¹, Xue Dong¹

¹School of Mechanical Engineering, Shanghai Jiao Tong University, National Engineering Laboratory for Automotive Electronic Control Technology, Shanghai, 200240, China

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

Abstract

One effective way of suppressing the cycle-to-cycle variation (CCV) in engine is to design a combustion system that is robust to the root cause of engine variation over the entire engine working process. Flash boiling has been demonstrated as an ideal technique to produce stable fuel spray. But the generation of stable intake flow and fuel mixture remain challenging. In this study, to evaluate the capability of swirl flow to produce repeatable fuel mixture formation, the fuel distribution inside a single cylinder optical engine under two swirl ratios were measured using laser induced fluorescence (LIF) technique. The swirl ratio was regulated by a swirl control valve installed in one of the intake ports. A planar laser sheet of 266 nm wavelength from a frequency-quadrupled Nd:YAG laser was directed into the optical engine through the quartz liner. The laser sheet was located at 15 mm below the tip of the spark plug. The fluorescence signal from the polycyclic aromatic hydrocarbon (PAH) in gasoline was collected by applying a 320-420 nm band pass filter mounted in front of an intensified CCD camera. Test results show that the in-cylinder fuel distribution is strongly influenced by the swirl ratio. Specifically, under high swirl condition, the fuel is mainly concentrated on the left side of the combustion chamber. While under the low swirl, fuel is distributed more randomly over the measurement plane. This agrees well with the previous measurements of the flame kernel location. Additionally, the cycle-to-cycle variation of the fuel distribution were analyzed. Results show that well organized fuel mixture pattern with low cycle-to-cycle variation was achieved under high swirl condition.

Keywords: Swirl ratio, fuel distribution, laser induced fluorescence, intensity gradient, cycle-to-cycle variation

Relating In-Cylinder Crank-Angle-Resolved Flame Temperature Measurement to Air-Fuel Ratio in a PFI Spark-Ignition Engine using Two-Colour Method

V. Balaji¹, Mayank Mittal^{1*}, Pramod S. Mehta¹, R. Pradheep¹,
S.R. Chakravarthy², and V. Lakshminarasimhan³

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

²Department of Aerospace Engineering, IIT Madras, Chennai, India.

³TVS Motor Company Limited, Hosur, India.

Abstract

In-cylinder flame temperature distribution is an important characteristic parameter to evaluate the performance of the combustion process in an internal combustion engine. In the present work, a 110 cm³ optically accessible spark-ignition engine with port-fuel-injection (PFI) system was used to measure the spatio-temporal flame temperature distribution using two-colour method. The two-colour method has been used by other researchers for flame temperature measurement in a spark-ignition engine [1, 2]. The injection source was placed upstream of the intake valve, which provided the precise metering of fuel under all operating conditions in the present work. After relating fuel injection pressure effect on mixture homogeneity using planar laser-induced fluorescence (PLIF), the influence of relative air-fuel ratio (or Λ , i.e. inverse of equivalence ratio) on flame temperature distribution at injection pressure of 3 bar was found significant in this study. The engine was operated at fixed engine speed of 1400 rpm and part throttle opening. Results showed that significant cycle-to-cycle variations exist in flame temperature distribution inside the engine cylinder. Considering different relative air-fuel ratios ($\Lambda = 0.85, 0.9$ and 1), it was found that the flame temperature was maximum at Λ of 0.9 compared to 0.85 and 1.0 .

Keywords: Optical engine, in-cylinder flame temperature, two-colour method

*Corresponding author: mmittal@iitm.ac.in

Imaging of Fuel Distribution in a PFI Spark-Ignition Engine using Planar Laser-Induced Fluorescence

R. Pradheep^{1,2*}, Mayank Mittal^{1,2}, Pramod S. Mehta^{1,2}, V. Balaji^{1,2},
V.R. Rajagopalan³, V. Lakshminarasimhan³, and S.R. Chakravarthy²

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

² National Centre for Combustion R & D, IIT Madras, Chennai, India.

³TVS Motor Company Limited, Hosur, India.

Abstract

It is important to understand the in-cylinder mixture distribution for improved engine performance, combustion process and emissions. In the present work, in-cylinder mixture distribution is studied using a planar laser-induced fluorescence technique. An optically accessible version of a TVS four-stroke 110 cc port fuel injection engine with a fully transparent liner and piston is used. Nd-YAG laser at 266 nm is used to induce the fluorescence of 3-pentanone tracer mixed with gasoline. The fluorescence is captured using an ICCD camera with a gate width of 200 ns at different planes during the intake and compression strokes. The effects of fuel injection pressure, throttle position and equivalence ratio are studied to understand the spatial and temporal variations of fuel distribution inside the combustion chamber. Results showed the existence of rich mixture regions near the intake valve and diametrically opposite to the spark plug. It was found that at a given plane, spatial variations in the mixture distribution were much higher during the start of intake, but reduced at the end of compression. At part throttle and low speed engine operations, inhomogeneity was observed at all planes. Comparing different equivalence ratios, the temporal and spatial inhomogeneity was increased with the increase in equivalence ratio. The spatial inhomogeneity, however, was reduced with the increase in fuel injection pressure.

Keywords: PLIF, mixture distribution, PFI, 3-pentanone, inhomogeneity

*Corresponding author: pradheep316@gmail.com

Numerical Simulation of Aerosol Transport and Dynamics in a Mixing Chamber

Janki Shinde¹, Pranav Dave², P. S. Rajagopal¹, Manish Joshi³, B.K. Sapra³, S. Anand³, Madhukar M.Rao¹, Akshai K. Runchal¹

¹CFDVRi, Mount View Complex, McLeod Ganj, Himachal Pradesh-176219, India

²ACRi infotech Pvt.Ltd, AECS Layout, Kundalahalli, Bengaluru-560037, India

³Bhabha Atomic Research Centre, Bombay, India

Abstract

Flow and aerosol transport and dynamics in a mixing chamber, part of an aerosol generation system, is analyzed using Computational Fluid Dynamics (CFD) and Computational Aerosol Dynamics. A predictable aerosol size distribution of particles is important for different aerosol related experiments. The effect of residence time of the aerosol particles and mixing characteristics of the flow on the aerosol size distribution is analyzed using the ANSWER finite volume CFD code. The ANSWER uses the steady state turbulent flow field to solve the General Dynamics Equations (GDE) for the aerosol particles. The GDE includes mechanisms such as coagulation, gravitational settling and thermophoretic drift etc. A volume (and mass) preserving nodal method is used to model particle distribution change due to coagulation. The size distribution in mixing chamber design is seen to be robust for various flow scenarios at inlet number concentration of $1 \times 10^{12} / \text{m}^3$. This is explained by flow time scale being much smaller than coagulation time scale. At higher inlet concentration of $1 \times 10^{15} / \text{m}^3$ the average size distribution and outlet size distribution are significantly shifted from the inlet distribution, due to the much lower coagulation time scale. A noticeable difference between no or low swirl and high swirl flow is seen. A secondary ring inlet within the mixing chamber was seen to lead to identical aerosol distribution for different flow scenarios even at higher inlet concentration.

Keywords: CFD, Aerosol, Coagulation, Deposition, Settling, Swirling Flow, Size Distribution, Residence Time, Mixing Chamber, GDE.

Experimental investigation of the Atomization Behavior of Ethanol and Kerosene in Acoustic Fields

D. Ju^{*}, X. Jia, Z. Huang, X. Qiao, J. Xiao, Z. Huang
Key Laboratory of Power Machinery and Engineering, Ministry of Education,
Shanghai Jiao Tong University
No. 800, Dongchuan Road, Minhang District, 200240 Shanghai, P. R. China

Abstract

Working conditions of advanced Aero engines are developing from conventional to ultra-conventional (ultra-high pressure and supercritical) conditions, and it should enhance the basic research on the atomization and combustion of the aviation fuel. Combustion instability leads severe vibration to the engine combustion device and even the entire system, and it also causes significant damage to the system components. The fuel atomization and evaporation is the controlling process of combustion rate, and it is an important mechanism of the combustion instability. This paper is planned to experimentally measure the spatial fuel atomization characteristics in an environment of acoustic field by high-speed imaging. Using multi-threshold image processing to analyze the effect of the high frequency oscillation on the fuel break-up, mixing mechanism during each stage of the fuel atomization, to explore the physical description and thermodynamic control mechanism of fuel droplet atmospheric environment, to illustrate how the pressure oscillation influences atomization characteristics of ethanol and aviation kerosene under different conditions. It will provide a theoretical basis for the fuel atomization mechanism in the environment of pressure oscillation, so as to lay the theoretical foundation of excitation mechanism of the aviation kerosene atomization and evaporation on combustion instability.

Keywords: Spray, Atomization, Ethanol, Kerosene, Acoustic Fields

^{*}Corresponding author: d.ju@sjtu.edu.cn

ANSYS®

10x

ANNOUNCING

ANSYS 17.0

The world's leading engineering simulation
software just got an order of magnitude better

ansys.com/17

Photo Credits

Front cover image : Jet in a cross flow. Credit : Senthil Kumar et al.
Rear cover image : Shore Temple, Mahabalipuram. Credit : Mikhil Surendran
Cover Design : S. Radhakrishnan



ILASS – Asia 2016

18th Annual Conference on Liquid Atomization and
Spray Systems