PSFVIP13

The 13th Pacific Symposium on Flow Visualization and Image Processing
August 7 - 10, 2022
Kogakuin University, Shinjuku, Tokyo, Japan

Program Book

Organized by Pacific Center of Thermal Fluids Engineering (PCTFE),
Visualization Society of Japan (VSJ) and Kogakuin University

Cooperated by JSME, HTSJ, GMSI

Contents

Welcome Address

Welcome to the PSFVIP13, The 13th Pacific Symposium on Flow Visualization and Image Processing. PSFVIP13 is organized by Pacific Center of Thermal Fluids Engineering (PCTFE), Visualization Society of Japan (VSJ) and Kogakuin University, cooperating by The Japan Society for Mechanical Engineers (JSME), The Heat Transfer Society of Japan (HTSJ) and GMSI, The University of Tokyo.

The objective of this symposium is to provide an opportunity for world-wide researches and engineers in all areas of flow visualization, image processing and related techniques to exchange knowledge and new experience in the spirit of collaboration and friendship. Papers dealing with current research ranging from fundamental principles and techniques to applications on the above topics are appropriate. Topics include, but are not limited to, flow field visualization, surface flow visualization, computer assisted techniques, graphical display of data, digital image processing and CFD studies.

PSFVIP13 is held at the same time and venue as the 50th Anniversary Symposium of Visualization Society of Japan, and some sessions is held jointly. PSFVIP13 offers a hybrid type of symposium that combines face-to-face and online modes. The "online type" will be able to respond to the situation of Covid-19, and people from overseas and other distant places will be able to participate easily. In the “face-to-face format”, participants will be able to have in-depth discussions with each other. We hope the symposium will be a wonderful experience for all participants.

Chair of PSFVIP13

Yasuyuki Yokono
Senior Researcher
School of Engineering
The University of Tokyo

Satoshi Someya
Duputy Director
Research Institute for Energy Conservation
National Institute of Advanced
Industrial Science and Technology

PSFVIP13 Committees

Symposium Chair

Yokono, Yasuyuki	The University of Tokyo	Japan
Someya, Satoshi	AIST	Japan

Secretary General

Funatani, Shumpei	University of Yamanashi	Japan

PCTFE President

Ishizuka, Masaru	Toyama Prefectural University	Japan

Advisor

Mochizuki, Sadanari	Tokyo University of Agriculture and Technology	Japan

Executive Committee

Baba, Kazutaka	JP-DOT.COM Co. Ltd.	Japan
Fukue, Takashi	Kanazawa Institute of Technology	Japan
Fushinobu, Kazuyoshi	Tokyo Institute of Technology	Japan
Hatakeyama, Tomoyuki	Toyama Prefectural University	Japan
Hiratsuka, Masaki	Kogakuin University	Japan
Ito, Shinichiro	Kogakuin University	Japan
Kawaguchi, Tatsuya	Tokyo Institute of Technology	Japan
Kibushi, Risako	Toyama Prefectural University	Japan
Koizumi, Katsuhiro	Nabtesco R&D Center	Japan
Komiya, Atsuki	Tohoku University	Japan
Konda, Kazuko	The University of Tokyo	Japan
Kubota, Yoshihiro	Toyo University	Japan
Li, Yanrong	Ibaraki University	Japan
Matsuda, Yu	Waseda University	Japan
Nakai, Kumi	AIST	Japan
Nishi, Koji	Ashikaga University	Japan
Nonomura, Taku	Tohoku University	Japan
Saitoh, Hironori	Sojo University	Japan
Torii, Shuichi	Kumamoto University	Japan
Tsubokura, Makoto	Kobe University	Japan
Yamagata, Takayuki	Niigata University	Japan
Yuki, Kazuhisa	Tokyo University of Science at Yamaguchi	Japan

Symposium Secretariat

GMSI Office	The University of Tokyo	Japan

Honorary Committee

Ay, H	National Kauohsiung Unversity of Applied Sciences	Taiwan
Brynjolfsson, S.	University of Iceland	Iceland
Cardone, G	University of Naples	Italy
Carlomagno, G.M.	Univ. of Napoles	Italy
Cervantes de Gortari,J.	National Univ. of Mexico	Mexico
Chan, S. H.	Yuan Ze University	Taiwan
Chinda Charoenphonphanich	KMITL	Thailand
De Vahl Davis, G.	Univ. of New South Wales	Australia
Dincer, I.	University of Ontario Institute of Technology	Canada
Djilali, N.	University of Victoria	Canada
Dost, S.	University of Victoria	Canada
Hanamura, K.	Tokyo Institute of Technology	Japan
Iguchi, M.	Hokkaido University	Japan
Inoue, M.	Komatsu	Japan
Jezek, J.	Czech Technical University in Prague	Czech
Kobayashi, T.	JARI	Japan

Kompenhans, J.	DLR	Germany
Lee, J. S.	Seoul National University	Korea
Lee, S. Y.	KAIST	Korea
Liou, T.M.	National Tsing Hua University	Taiwan
Mallinson, G.	University of Auckland	New Zealand
Merzkirch, W.	Univ. Essen	Germany
Mochizuki, S	Tokyo University of A & M	Japan
Milton, B.	University of New South Wales	Australia
Prenel, J. P.	Franche-Comte University	France
Raupenstrauch, H.	Montan University	Austria
Sang Joon Lee	POSTECH	Korea
Settles, G.S.	Penn State University	USA
Spitas, C	Delft University of Technology	Netherlands
Suzuki, K	Tokyo University of Science at Yamaguchi	Japan
Takeishi, K. T	okushima Bunri Univesity	Japan
Wang, B. X.	Tsinghua University	China
Winoto, S. H.	National University of Singapore	Singapore
Yanuar	University of Indonesia	Indonesia
Yoo, J.Y.	Seoul National University	Korea
Znamenskaya Irina	Lomonosov Moscow State University	Russia
Zuna, P.	Czech Technical University in Prague	Czech

International Scientific Committee

Akoshima, Megumi	AIST	Japan
Asano, Hitoshi	Kobe University	Japan
Cardone, Gennaro	University of Napoles	Italy
Carlomagno, G.M.	Univ. of Napoles	Italy
Cater, J.	University of Auckland	New Zealand
Choi, Chang Kyoung	Michigan Technology University	USA
Fujisawa, Nobuyuki	Niigata University	Japan
Fukue, Takashi	Kanazawa Institute of Technology	Japan
Fumoto, Koji	Hirosaki University	Japan
Hirasawa, Shigeki	Kobe University	Japan
Hirata, Katsuya	Doshisha University	Japan
Hirose, Koichi	Iwate University	Japan
Hirota, Masafumi	Mie University	Japan
Hishida, Koichi	Keio University	Japan
Hong, C. W.	National Tsing Hua University	Taiwan
Hu, Hui	Iowa State University	USA
Ichimiya, Koichi	University of Yamanashi	Japan
Ipposhi, Shigetoshi	Mitsubishi Electric Co.Ltd.	Japan
Itaya, Yoshinori	Gifu University	Japan
Iwai, Hiroshi	Kyoto University	Japan
Joshi, Y.	Georgia Institute of Technology	USA
Kaewpradap, A.	King Mongkut's University of Technology Thonburi	Thailand
Kano, Ichiro	Yamagata University	Japan
Katoh, Yasuo	Yamaguchi University	Japan
Kawanami, Tsuyoshi	Kobe University	Japan
Kedzierski, M.	National Institute of Standards and Technology	USA
Kuzma-Kichta, Y. A.	Moscow Power Eng. Institute	Russia
Kihm, Kenneth D.	University of Tennessee	USA
Kim, H. D.	Andong National University	Korea
Kim, K. C.	Pusan National University	Korea
Komiya, Atsuki	Tohoku University	Japan
Komiyama, Masaharu	Gifu University	Japan
Kunugi, Tomoaki	Kyoto University	Japan
Lee, S. J.	Pohang University of Science and Technology	Korea
Li, H.Y.	Huafan University	Taiwan
Liou, Tong-Miin	National Thing Hua University	Taiwan
LIU, Yingzheng	Shanghai Jiaotong University	China
Maekawa, Hiroshi	The University of Electro-Communications	Japan
Markovich, Dmitry	Kutateladze Institute of Thermophysics	Russia

Maruyama, Shigenao	Tohoku University	Japan
Matsumoto, Mitsuhiro	Kyoto University	Japan
Matsumoto, Ryosuke	Kansai University	Japan
Matsushima, Hitoshi	Nihon University	Japan
Mosharov, Vladimir	Moscow University Physics and Technology	Russia
Motosuke, Masahiro	Tokyo University of Science	Japan
Munakata, Tetsuo	AIST	Japan
Murata, Akira	Tokyo University of Agriculture and Technology	Japan
Nagai, Niro	University of Fukui	Japan
Nagasaki, Takao	Tokyo Institute of Technology	Japan
Nakabe, Kazuyoshi	Kyoto University	Japan
Nakamura, Hajime	National Defense Academy	Japan
Nakamura, Yuji	Toyohashi University of Technology	Japan
Ng, Eddie	Nanyang Technical University	Singapore
Nishi, Koji	NIDEC CORPORATION Japan	Japan
Nishida, Keiya	Hiroshima University	Japan
Oda, Yutaka	Kansai University	Japan
Ohmi, Kazuo	Osaka Sangyo University	Japan
Ohara, Taku	Tohoku University	Japan
Okajima, Junnosuke	Tohoku University	Japan
Okuyama, Masaaki	Yamagata University	Japan
Oomura, Takahiro	National Institute of Technology, Wakayama College	Japan
Ozawa, Mamoru	Kansai University	Japan
Paeng, D. G.	Jeju Water Industry Center	Korea
Riznic, J.	Canadian Nuclear Safety Commission	Canada
Rosengarten, G.	University of New South Wales	Australia
Roy, R.P.	Arizona State University	USA
Saito, Takushi	Tokyo Institute of Technology	Japan
Senaha, Izuru	Ryukyu University	Japan
Shibahara, Masahiko	Osaka University	Japan
Shiomi, Junichiro	The University of Tokyo	Japan
Someya, Satoshi	AIST	Japan
Suzuki, Hiroshi	Kobe University	Japan
Tabe, Yutaka	Hokkaido University	Japan
Tagawa, Masato	Nagoya Institute of Technology	Japan
Taguchi, Yoshihiro	Keio University	Japan
Takei, Masahiro	Chiba University	Japan
Tong, Albert Y.	University of Texas at Arlington	USA
Tsushima, Shoji	Osaka University	Japan
Yamada, Shunsuke	National Defense Academy	Japan
Yamashiro, Hikaru	National Institute of Technology, Okinawa College	Japan
Yasuhara, Caoru	Yamagata University	Japan
Yang, C. Y.	National Central University	Taiwan
Yan, W.M.	National Taipei University of Technology	Taiwan
Yazawa, Kazuaki	Purdue University	USA
Yeh, L-T	Huawei Technologies	USA
Yokono, Yasuyuki	The University of Tokyo	Japan
Yuuki, Kazuhisa	Tokyo University of Science, Yamaguchi	Japan
Zhu, Ning	Shizuoka Institute of Science and Technology	Japan

Timetable

Aug. 7		Aug. 8		Aug. 9		Aug. 10	
		9:00-10:00	Registration	9:00-9:40	Plenary Lecture II (Room A) Professor Han Seo Ko	9:00-9:40	Plenary Lecture III (Room A) Professor Nobuyuki Oshima
		10:20-11:50	Registration	9:50-10:30	VSJ Memorial Plenary Lecture 2	10:00-12:00	Technical Sessions 6 (6) Room F,G,H,I F(GS7) 3, 25, 53, 57, 76, 78 G(GS6) 86, 32, 55, 58, 75 H(OS3(VSJ OS6)) J63, J68, J85, J90 I(GS2) 8, 10, 111, 18, 90, 99
				10:40-11:40	VSJ Memorial Panel Session		
		13:00-13:50	Opening (Room A) Plenary Lecture I (Room A) Professor Jan A.Stasiek	13:00-14:40	Technical Sessions 3 (5) Room F,G,H,I,E F(OS6) 33, 48, 54, 100, 61 G(OS2) 44, 46, 56, 74, 84 H(OS1(VSJ OS3)) J4, J9, J17, J29 I(GS13) 14, 41, 50, 69, 80 E(OS9) 12, 21, 22, 26, 30	13:00-15:00	Technical Sessions 7 (6) Room F,G,H F(GS7) 59, 93, 94, 96, 5 G(GS4) 43, 67, 72, 85, 91 H(OS3) 124, 119, 120, 121, 122, 123
		14:00-14:40	VSJ Memorial Plenary Lecture 1				
		15:00-16:20	Technical Sessions 1 (4) Room F,G,H,I,E F(OS5) 6, 28, 68 G(OS4) 24, 47, 97, 98 H(OS1) 88, 89, 105, 106 I(GS15) 110, 64, 65, 95 E(OS8) 4, 13, 16, 19	15:00-16:20	Technical Sessions 4 (4) Room F,G,H,I,E F(OS7) 29, 17, 34, 7 G(OS2 (VSJ OS2)) J33, J36, J69, J88 H(OS1) 51, 77, 109, 108 I(GS2) 2, 23, 37, 101 E(OS8) 83, 60, 114, 39	RED: joint session with VSJ	
16:00-17:00	Registration	16:40-18:00	Technical Sessions 2 (4) Room G,H,I,E G(OS4) 35, 102, 107 H(OS1) 62, 79, 87, 113 I(GS15) 45, 66, 70, 73 E(OS8) 9, 20, 31, 11	16:40-18:00	Technical Sessions 5 (4) Room F,G,H,E F(OS7) 52, 63, 104, 49 G(OS2 (VSJ OS2)) J34, J57, J70 H(OS1(VSJ OS3)) J28, J32, J50 E(OS8 (VSJ OS7)) J39, J73, J74, J75		
17:00-18:00	Welcome Reception	18:30-20:00	PSFVIP13 Symposium Dinner	18:30-20:00	VSJ Bnquet		

PSFVIP13 offers a hybrid type of symposium that combines face-to-face and online modes.

Online will be by "Zoom."

Room URL and the proceedings password are informed to the registered participants by e-mail.

Eduroam is available in the symposium site.

Plenary Lectures: 35minutes talk, 5minutes Q/A

Technical sessions: 15minutes talk, 3minutes Q/A

Plenary Lectures

Professor Jan A. Stasiek
Professor, Gdansk University of Technology, Poland

Liquid crystal thermography as a modern and unique tool for technical and scientific research

The beginning of the 21st century is marked by the years of intensive research and development with micro- and nanotechnology, highly effective energy conversion technology and methods of reducing its impact on environmental degradation. Scientific researchers dealing with heat and mass transfer must face new challenges in the design and construction of modern and highly efficient processes, devices and apparatus. Designers of thermal devices expect, for example, more detailed information on the local values of the heat transfer coefficient, without limiting to its averaged values. These expectations are met by automated and fully computerized liquid crystal thermography (LCT) based on physico-chemical properties, among others: cholesterol esters (with a helicoids structure) as well as colour digital images processing. It allows not only to study complex geometries or the influence of inhomogeneous boundary conditions, but also pulsatile and turbulent flows of large scales and low frequencies. Phenomena characterized by such a complex heat transfer can be found in most devices where energy conversion processes take place. Furthermore, liquid crystal thermography (LCT) assisted by particle image velocimetry (PIV) and digital image processing (DIP) is an excellent and recognized measurement method used in the study of advanced thermal-flow processes and in major world-wide research centres. Theoretically, LCT enables the visualization of two-dimensional temperature and velocity fields with an accuracy of one pixel and identifying the influence of many thermo-physical and flow factors on the formation of a laminar and turbulent boundary layer. Therefore ,in the plenary lecture, the history of LCT techniques will be reviewed and the principal methods are described, using examples from current literature and own research.

Professor Han Seo Ko
Professor School of Mechanical Engineering,
Sungkyunkwan University, Korea.

Practice Your Scales! Nanomaterials for Fast Energy Processes

The interaction between electrostatic and hydrodynamic phenomena was analyzed by adopting advanced techniques for numerical simulation and flow visualization. Rising bubbles inside a conductive medium can affect formation of electric field. The current varies depending on the occupation of non-conductive volume, and local electric signals can be interpreted as dispersed phase. Interesting influence between different physics can be applied to the study on a wire-mesh sensor (WMS) which is used to measure the characteristics of the multiphase flow. Electrohydrodynamic (EHD) flow is a promising field, especially for gas pumping applications. The EHD flow can be produced by the formation of non-thermal plasma (NTP) on a surface with high curvature. In this study, an analytical model was employed, and the experiment was carried out with the configuration of a wire and two cylinders in an open space. The Townsend relation between electric signals was checked for the stable generation of corona discharge, and the empirical equation was employed to verify the relation between parameters, such as current, voltage, and momentum. The cooling performance of the EHD flow was evaluated by comparison with the heat transfer coefficient of natural convection. The heat transfer coefficient and Nusselt number were calculated, and their correlation could be obtained.

Professor Nobuyuki Oshima
Professor, Division of Mechanical and Aerospace Engineering
Faculty of Engineering, Hokkaido university

Be Ambitious ! to use Open-source software for CFD and flow visualization

Recently, useful open-source software are published for CFD and related simulations. They are widely applied to varieties of physical and engineering problems, which are operated by difference scale computers, from notebook PC to supercomputer “FUGAKU”. As an entrance of “Open-source CFD”, a few results of major software as OpenFOAM, SU2, FrontFlow and their coupling software are introduced for the applications to bubble flow in turbulent boundary layer, flow-structure interaction of artery flow, and spray/combustion flow in gas-turbine, etc. These examples will make researchers, scientists and engineers “Be Ambitious !” to use Open-source CFD and flow visualization.

On the other hand, open-source CFD still has merits and demerits to develop, to distribute and to use, as compared to other ways by commercial software and/or unpublished special-maid ones, though the recent development of computers and networks brings much easier access to required data and information. Shall we discuss how to support such open-source world in CFD and visualization?

Program Details

joint session with VSJ

Day1	JST(UTC+9) Aug.8, 09:00-20:00
JST(UTC+9)09:00-10:00	**Room A**
	Registration
JST(UTC+9)10:20-11:50	VSJ Anuual Meeting
JST(UTC+9) Aug.8 13:00-13:50	**Opening remarks** *Session Chair: Satoshi Someya* (AIST) *Yasuyuki Yokono* (The University of Tokyo) *Masaru Ishizuka* (PCTFE Japan) **Plenary Lecture I** Session Chair: *Sumpei Funatani* (Yamanashi University) Liquid crystal thermography as a modern and unique tool for technical and scientific research *Jan A.Stasiek* (Gdansk University of Technology)
JST(UTC+9)14:00-14:40	**VSJ Memorial Plenary Lecture 1 (*in Japanese*)**
JST(UTC+9) Aug.8 15:00-16:20	**Room E**
	Technical Sessions E1 OS8*: Physiological/Biological fluid dynamics Session Chair: Kenichi Funamoto(Tohoku University)
	4 INVESTIGATION OF THE INFLUENCE OF VASCULAR ACCESS SHAPE ON HEMODYNAMIC PARAMETERS *Hitomi Anzai, Kazuki Takeda, Ai Kajiyama and Makoto Ohta*
	13 FUNDAMENTAL STUDY OF MR-MEASUREMENT-INTEGRATED SIMULATION OF HEART-AORTA-SYSTEM: APPLICATION TO AN AORTIC ARCH *Suguru Miyauchi*
	16 SIMULATION OF A PACKED SUSPENSION OF MICROSWIMMERS *Yu Kogure, Toshihiro Omori and Takuji Ishikawa*
	19 VISUALIZATION OF FLOW FIELD INDUCED BY HYDRODYNAMIC INTERACTIONS OF SWIMMING MICROORGANISM *Kiyoto Kubo, Toshihiro Omori and Takuji Ishikawa*
	Room F
	Technical Sessions F1 OS5: Thermal management of electronics Session Chair: Katsuya Hirata(Doshisha University)
	6 START-UP CHARACTERISTICS OF LOOP HEAT PIPE WITH PLATE-TYPE EVAPORATOR *Tomohiro Nakazawa, Atsushi Tsujimori, Tsubasa Usui, Koji Ono, Reo Miwa, Norifumi Miyanaga, Yui Sato and Haruka Suzuki*
	28 Visualization of Thermal Networks in Layered Structures by Means of Pump-Probe and Deconvolution Technique *Joao Vitor Thomsen Silveira, Daiki Higuma and Kazuyoshi Fushinobu*
	68 INFLUENCE ON THERMAL IMPEDANCE ALONG HEAT TRANSFER PATH OF POWER SEMICONDUCTOR PACKAGE BY COPPER PAD ON PRINTED CIRCUIT BOARD *Koji Nishi*
	Room G
	Technical Sessions G1 OS4: Combustion and Reacting Flow Session Chair: Chihiro Kondo(Okayama University of Science)
	24 NEAR-LIMITING BEHAVIOR OF POOL FIRES WITH STRONG SWIRLING FLOW *Keita Tsukamoto, Akira Odawara, Takuya Yamazaki, Tsuneyoshi Matsuoka and Yuji Nakamura*
	47 INFLUENCE OF THE THIRD STREAM TO A THREE-FEED NON-PREMIXED COMBUSTION SYSTEM *Sujeet Yadav, Panlong Yu and Hiroaki Watanabe*
	97 NUMERICAL ANALYSIS ON THE TEMPERATURE OF AN ETHANOL SPRAY IN ITS MIXTURE FORMATION PROCESS UNDER DIESEL ENGINE OPERATING CONDITIONS *Hironori Saitoh and Hikaru Tsunoda*
	98 MICROSCOPIC IMAGE DIAGNOSTICS WITH VISIBLE EMISSION LIGHT FOR SELF-BURNING SURFACE TEMPERATURE DISTRIBUTION OF CFRP (CARBON FIBER REINFORCED PLASTIC) IN OXYGEN *Yojiro Ishino and Rion Ito*
	Room H
	Technical Sessions H1 OS1*: Luminescence-based Flow Measurement for Thermofluid Analysis Session Chair: Hideo Mori(Kyushu University)
	88 Unsteady pressure distribution measurement on cylinder surface using anodized aluminum pressure-sensitive paint during shock wave passage *Shintaro Tamakuma, Kazuma Yomo, Toshihiro Ogawa and Hiroki Nagai*
	89 Development of pressure distribution measurement technology on free-flight object surface at transonic speed *Yota Hosono, Kazuma Yomo, Koichi Takahashi, Toshihiro Ogawa, Kiyonobu Ohtani and Hiroki Nagai*
	105 Evaluation of pressure distribution around small rotor blade in low oxygen partial pressure region using PSP *Kousei Ogawa, Yanrong Li, Satoshi Someya, Terumi Inagaki, Shimpei Saito, Soumei Baba and Naoki Takada*
	106 NOISE SUPPRESSION METHOD FOR PSP DATA BASED ON REDUCED-ORDER MODELING *Tomoki Inoue, Yu Matsuda, Tsubasa Ikami, Taku Nonomura, Yasuhiro Egami and Hiroki Nagai*
	Room I
	Technical Sessions I1 GS15: Classical and advanced measurement Session Chair: Hajime Nakamura(National Defense Academy)
	110 IMPROVEMENT OF SPATIAL RESOLUTION OF HIGH-SPEED-BACKGROUND-ORIENTED SCHLIEEN THROUGH ATOMOSPHERIC TURBULENCE *Toshiharu Mizukaki, Miyuki Kondo and Takeo Minezaki*
	64 THREE-DIMENTIONAL DENSYTY MEASUREMENT USING SIMULTANEOUS MULTI-ANGLE BOS MEASUREMENT SYSTEM IN A SUPERSONIC WIND TUNNEL *Masato Yamagishi, Yusuke Hirose, Shinsuke Udagawa, Tatsuro Inage and Masanori Ota*
	65 HIGH ACCURATE DENSITY MEASUREMENT OF TRANSONIC FLOW FIELD AROUND THE REENTRY CAPSULE MODEL *Yutaro Katagiri, Nao Kosaka, Masato Yamagishi, Yusuke Hirose, Masanori Ota, Masayuki Nomura, Koji Hujita, Kiyonobu Kiyota and Hiroki Nagai*
	95 IMPROVEMENT IN ACCURACY OF THE BACKGROUND ORIENTED SCHLIEREN TECHNIQUE WITH PARALLEL PROJECTION *Nao Kosaka, Yuto Igari, Yutaro Katagiri, Masato Yamagishi and Masanori Ota*

Day1	joint session with VSJ
JST(UTC+9) Aug.8 16:40-18:00	**Room E**
	Technical Sessions E2 OS8*: Physiological/Biological fluid dynamics Session Chair: Hitomi Anzai (Tohoku University)
	9 COMPARISON OF HEMODYNAMICS IN LEFT VENTRICLES WITH TRICUSPID AORTIC VALVE AND BICUSPID AORTIC VALVES *Shingo Tsuda, Suguru Miyauchi and Kenichi Funamoto*
	20 MODELING SPERM SUSPENSION IN THREE DIMENSIONS *Nanami Taketoshi, Toshihiro Omori and Takuji Ishikawa*
	31 VISUALIZATION OF A FRESHWATER-SPONGE SNEEZING *Kei Kawashima, Kenji Kikuchi and Takuji Ishikawa*
	11 COMPUTATIONAL MODELING OF THE SPONGE'S CHOANOCYTE CHAMBER *Takumi Ogawa, Toshihiro Omori, Kenji Kikuchi and Takuji Ishikawa*
	Room G
	Technical Sessions G2 OS4: Combustion and Reacting Flow Session Chair:Hironori Saitoh(Sojo University)
	35 INERT GAS FLOW RELEASED FROM BURSTING SAUSAGE-SHAPED RUBBER BALLOON AND ITS EXTINGUISHING CHARACTARISTICS *Hinako Mikami and Hiroyuki Torikai*
	102 Simultaneous measurement of gaseous flow and spark channel using inexpensive inorganic fluorescent tracer *Chihiro Kondo and Masanobu Yoshioka*
	107 SCHLIEREN VISUALIZATION OF EXTINGUISHING PROCESS OF DIFFUSION FLAMES IN AIRBURST BLAST EXTINGUISHMENT WITH MICRO EXPLOSIVES *Yuma Yoshida and Hiroyuki Torikai*
	Room H
	Technical Sessions H2 OS1*: Luminescence-based Flow Measurement for Thermofluid Analysis Session Chair: K.C.Kim(Pusan National University)
	62 Dependence of measurement accuracy and time delay of lifetime-based dual-layer PSP/TSP on film thickness *Hideo Mori, Takamitsu Kuroki and Haruka Nishiyama*
	79 Frequency Response of Carbon-nanotube Temperature-sensitive Paint *Tsubasa Ikami, Koichi Takahashi, Yasufumi Konishi and Hiroki Nagai*
	87 Development of visualization technique for hydrodynamic stress field utilizing fluorescent force probe *Reiko Kuriyama, Ryohei Okamoto, Waka Yamamoto, Hidetsugu Kitakado, Shohei Saito, Kazuya Tatsumi and Kazuyoshi Nakabe*
	113 DEVELOPMENT OF TEMPERATURE MEASUREMENT SYSTEM OF AIRFLOW USING ULTRA-FINE THERMOSENSITIVE LUMISIS PHOSPHOR WIRE *Yu Morito and Shumpei Funatani*
	Room I
	Technical Sessions I2 GS15: Classical and advanced measurement Session Chair: Masanori Ota(Chiba University)
	45 EVALUATION OF THE INFLUENCE OF READING PIXEL IN THE DRAG COEFFICIENT CALUCULATED FROM HIGH-SPEED VIDEO CAMERA IMAGES *Tomohiro Miyazaki, Takamasa Kikuchi and Akinori Muramatsu*
	66 Ultrasound Vector Projectile Imaging of Urinary Flow Dynamics *Takuro Ishii*
	70 STUDY ON DRAG COEFFICIENT FOR CIRCULAR PLATE WITH HOLES *Kazuki Namba, Yoshihiro Kubota and Osamu Mochizuki*
	73 Visualization of Shock waves above water driven by water entry using shadow graph method *Yusuke Ebii, Takamasa Kikuchi and Akinori Muramatsu*
JST(UTC+9)18:30-21:00	**PSFVIP13 Symposium Dinner**

joint session with VSJ

Day2	JST(UTC+9) Aug.9, 09:00-20:00
JST(UTC+9) Aug.9 09:00-09:40	**Room A** **Plenary Lecture Ⅱ** Session Chair: *Satoshi Someya* (AIST) Analysis of multi-phase and electrohydrodynamic (EHD) flow for convective heat transfer using visualization technique Han Seo Ko (Sungkyunkwan University)
JST(UTC+9)09:50-10:30	**VSJ Memorial Plenary Lecture 2 (*in Japanese*)**
JST(UTC+9)10:40-11:40	**VSJ Memorial Panel Session (*in Japanese*)**
JST(UTC+9) Aug.9 13:00-14:40	**Room E**
	Technical Sessions E3 OS9*: Wavelets and Flow Control Session Chair: Yoshiki Niizeki(Tokushima-Bunri University)
	12 Multiscale flow structures in a cube wake influenced by the front inclined hole *Jiawei Li, Xiaolei Han, Hiroka Rinoshika and Akira Rinoshika*
	21 Experimental study of moving-TRPIV on drag reduction control of coherent structures in turbulent boundary layer over superhydrophobic surface *Yue Jinhui, Wang Xinwei and Jiang Nan*
	22 Tomographic PIV measurements in the wake of a wall-mounted barchan dune *Xiaolei Han and Akira Rinoshika*
	26 3D WAVELET TRANSFORM ON FLOW STRUCTURES BEHIND A WALL-MOUNTED SHORT CYLINDER WITH A FRONT INCLINED HOLE *Hiroka Rinoshika*
	30 EXPERIMENTAL INVESTIGATION ON FLOW CHARACTERISTICS AND HEAT TRANSFER OF NONCIRCULAR SYNTHETIC JETS *Lei Wang, Lihao Feng, Yang Xu and Jinjun Wang*
	Room F
	Technical Sessions F3 OS6: Visualization-based Advanced Flow Measurement and Control Session Chair: Kumi Nakai(National Institute of Advanced Industrial Science and Technology)
	33 STUDY OF UNDEREXPANDED SUPERSONIC JETS FROM AXISYMMETRIC MICRO-NOZZLES IN LOW REYNOLDS NUMBER REGIONS *Tenta Tashiro, Sinichiro Nakao and Yoshiaki Miyazato*
	48 SPATIAL SUPERRESOLUTION BASED ON PROPER ORTHORGONAL DECOMPOSITION AND BAYSIEN ESTIMATION OF SUBSONIC JET FLOW MEASURED BY TWO MAGNIFICATION PIV SIMULTANEOUS MEASUREMENT *Honda Harutaka, Yuta Ozawa and Taku Nonomura*
	54 A TIME-RESOLVED THREE-DIMENSIONAL DENSITY MEASUREMENT FOR AN ASYMMETRICAL UNSTEADY SUPERSONIC IMPINGING JET USING A FIBRE BOS TECHNIQUE *Takuma Kuroda, Masahiro Kobayashi and Takahiro Ukai*
	100 RESOLUTION ENHANCEMENT BY THE EDSR METHOD FOR BOS ANALYSES *Katsunari Ota and Takahiro Ukai*
	61 Improvement of Robustness on Real-time Flow Field Measurement using Sparse Processing PIV *Chihaya Abe, Naoki Kanda, Sayumi Kaneko, Kumi Nakai and Taku Nonomura*
	Room G
	Technical Sessions G3 OS2*: Measurement Technique　Noninvasive Measurement Technique Session Chair: Jun Sakakibara(Meiji University)
	44 COLOR PIV FOR MEASURING THREE-DIMENSIONAL WAKE STRUCTURES OF A DELTA WING FROM 0 TO 90 DEGREE IN ANGLE OF ATTACK *Kokoro Ochi, Yasufumi Horimoto, Hyun Jin Park, Yuji Tasaka and Yuichi Murai*
	46 EXPERIMENTAL INVESTIGATION OF SHEAR-INDUCED BIREFRINGENCE PROPERTIES OF RHEOSCOPIC FLUIDS FOR VISUALIZATION OF STRESS FIELD OF FLUID FLOW *Kosei Hayashi, Shuntaro Tanaka and Katsuaki Shirai*
	56 Improvement of particle tracking velocimetry with deep learning *Taishi Yano and Yuji Nakanishi*
	74 DEVELOPMENT OF HIGH-RESOLUTION PSV USING CODED ILLUMINATION AND FEATURE MATCHING ALGORITHM *Yusaku Tsukamoto and Shumpei Funatani*
	84 Time-resolved PIV Measurements of Flow Field around a Badminton Shuttlecock during Turnover Process *Yuki Sakurai, Kenichi Nakagawa and Hiroaki Hasegawa*
	Room H
	Technical Sessions H3 (***in Japanese***) OS1*(VSJ OS3): Luminescence-based Flow Measurement for Thermofluid Analysis Session Chair: Kazunori Mitsuo(JAXA)
	J4 Development of a white screen layer using a water-soluble polymer suitable for fast-responding pressure-sensitive paint *Hiromu Horie, Miku Yamazaki, Yushi Matsumura and Yasuhiro Egami*
	J9 Quantitative study of property changes due to interference between dyes in mixed-type two-color PSPs *Yushi Matsumura, Yuki Ishida, Takumi Inayoshi, Hiromu Horie and Yasuhiro Egami*
	J17 Progress of Simultaneous Measurement of Pressure and Temperature Fields by DL-PTSP Applying Lifetime-Based Calibration Method *Takamitsu Kuroki, Haruka Nishiyama and Hideo Mori*
	J29 Unsteady Measurement of Back Flow Induced by Surging Caused by Transonic Centrifugal Compressor Using DL-PTSP *Masashi Yoshikawa, Kodai Tsuji, Hideo Mori and Masato Furukawa*
	Room I
	Technical Sessions I3 GS13: Heat transfer and combustion Session Chair: Atsuki Komiya(Tohoku University)
	14 TIME-RESOLVED VISUALIZATION OF HEAT FLUXES FROM NEAR-WALL TURBULENT FLOWS *Irina Znamenskaya, Ekaterina Koroteeva, and Vladimir Chirich*
	41 Flow Boiling Heat Transfer in a Plate Heat Exchanger with Mixed Chevron Angle Plates *Mu-Ting Hsieh, Chien-Yuh Yang and Fu-Chen Lin*
	50 VISUALIZATION OF TRANSIENT HEAT TRANSFER BY A DROPLET BOUNCING ON A HEATED SURFACE VIA THERMOGRAPHY *Masaki Yoshida, Shunsuke Yamada, Yuki Funami and Hajime Nakamura*
	69 EXPERIMENTAL STUDY OF THERMAL CONVECTION BY HEATING AND COOLING THE SIDE OR TOP SURFACE OF A ROTATING CYLINDRICAL VESSEL *Kohei Shimoura, Tatsunori Ihara, Takashi Noguchi and Katsuya Hirata*
	80 LOCAL AND TRANSIENT HEAT TRANSFER MEASUREMENT FOR DROPWISE CONDENSATION USING TEMPERATURE SENSITIVE PAINTS *Kohei Matsushima, Tomoya Uchimura, and Yutaku Kita*

Day2	joint session with VSJ
JST(UTC+9) Aug.9 15:00-16:20	**Room E**
	Technical Sessions E4 OS8*: Physiological/Biological fluid dynamics Session Chair: Yoshihiro Kubota (Toyo University)
	83 VIDEO ANALYSIS ABOUT THE GLIMMER SYNCHRONIZATION OF LUCIOLA PARVULA FIREFLIES *Shiro Kobayashi, Nao Ninomiya, Masayuki Iigo and Rion Tada*
	60 Visualization of Bubble Dynamics inside the Proofing Bread Dough using X-ray Microtomography *Kyosuke Kimura, Kenji Kikuchi, Keiko Numayama, Takuji Ishikawa and Taimei Miyagawa*
	114 Visualization of microbial transport in the gut of a zebrafish larva *Yuta Kikunaga, Kenji Kikuchi, Keiko Numayama-Tsuruta and Takuji Ishikawa*
	39 INFLUENCE OF THE PERIODIC FLOW ON THE FLOW STRUCTURE AROUND THE ROBOTIC FISH *Maoto Kawano, Yohsuke Tanaka and Yoshitaka Isoda*
	Room F
	Technical Sessions F4 OS7: Pipe flows, channel flows and internal flows Session Chair: Shumpei Funatani (Yamanashi University)
	29 VISUALIZATION OF MIXING AND FLOW STRUCTURE IN BLADE-FREE PLANETARY MIXER WITH FREE SURFACE *Takayuki Yamagata, Kota Sato and Tsubasa Igarashi*
	17 MASS TRANSFER AND VELOCITY MEASUREMENTS IN T-JUNCTION PIPING SYSTEM *Kazuki Munemura, Kazuki Kawaguchi, Takayuki Yamagata and Ryo Morita*
	34 NUMERICAL SIMULATION OF MASS TRANSFER AND VELOCITY IN T-JUNCTION PIPING SYSTEM *Masaaki Satake, Shun Watanabe, Ryo Morita and Takayuki Yamagata*
	7 VISUALIZATION OF INTERMITTENT TURBULENT STRUCTURE IN ECCENTRIC ANNULAR PIPE FLOW *Kohei Yatsushiro, Yasufumi Horimoto and Takahiro Tsukahara*
	Room G
	Technical Sessions G4 (***in Japanese***) OS2*(VSJ OS2): Measurement Technique Noninvasive Measurement Technique Session Chair: Shigeo Hosokawa (Kansai University)
	J33 Development of local vorticity instrumentation method using stretched film tracer *Yuki Kudo, Shigeru Muraka and Yohsuke Tanaka*
	J36 Vortex Structure Formed by a Main flow and a Sweeping Jet *Shoma Tanaka and Masaki Fuchiwaki*
	J69 Time Series image processing for measuring particle size of single droplet atomization *Tokiha Yada, Kyohei Mizuno, Koudai Iwasaki, Kouta Nakata, Makoto Asahara and Takechi Miyasaka*
	J88 Characteristics of unsteady surface tension convection driven by spot heating of a thin liquid layer *Tiwari Ratnanjali, Shuma Ogawa and Koichi Nishino*
	Room H
	Technical Sessions H4 OS1*: Luminescence-based Flow Measurement for Thermofluid Analysis Session Chair: Hiroki Nagai(Tohoku University)
	51 VISUALIZATION OF TEMPERATURE FIELD IN A CHANNEL BY LASER INDUCED FLUORESCENCE AND TEMPERATURE DEPENDENCE OF PARTICLE BEHAVIOR IN THE CHANNEL *Kazuki Hirai, Masahiro Yajima, Hideaki Kato and Katsuaki Shirai*
	77 ADAPTIVE WINDOW TECHNIQUE FOR LIFETIME-BASED TEMPERATURE AND VELOCITY SIMULTANEOUS MEASUREMENT USING THERMOGRAPHIC PARTICLE TRACKING VELOCIMETRY WITH A SINGLE CAMERA *Tao Cai, Jeongmin Han, Mirae Kim, Juyong Jung, Hyungmin Shin and Kyung Chun Kim*
	109 A CRYOGENIC TEMPERATURE SENSITIVE PAINT MEASUREMENT USING METAL COMPLEX MATERIALS AND PHOSPHORS ~TEMPERATURE SENSITIVITY OF OPTICAL PROPERTIES ~ *Yanrong Li, Sou Yasuzawa, Satoshi Someya, Shimpei Saito, Soumei Baba and Naoki Takada*
	108 A CRYOGENIC TEMPERATURE SENSITIVE PAINT MEASUREMENT USING METAL COMPLEX MATERIALS AND PHOSPHORS ~TEMPERATURE SENSITIVITY OF OPTICAL PROPERTIES ~ *Sou Yasuzawa, Satoshi Someya, Yanrong Li, Shimpei Saito, Soumei Baba and Naoki Takada*
	Room I
	Technical Sessions I4 GS2: Multiphase flow (other) Session Chair: Eiji ISHII(Hitachi Corporation)
	2 GAS HYDRATE FORMATION AND DISSOCIATION UNDER STATIC AND DYNAMIC CONDITIONS *Yasuharu Nakajima, Joji Yamamoto, Satoru Takano, Marcio Yamamoto, Masao Ono, Shigeo Kanada and Kazuhisa Otsubo*
	23 Reversal Motion of the Solid Particles with High Thermal Conductivity in the Solid-dispersed Rayleigh-Bénard Convection *Shi Dai*
	37 THE FIBER-ORIENTATION OF CELLULOSE NANOFIBER SUSPENSIONS IN A PLANAR EXPANSION FLOW *Akiyoshi Kusano, Akiomi Ushida and Taisuke Sato*
	101 Free surface flow of concentrated particle suspension in Hele-Shaw cell *Takashi Koshiba and Takehiro Yamamoto*

Day2	joint session with VSJ
JST(UTC+9) Aug.9 16:40-18:00	**Room E**
	Technical Sessions E5 (***in Japanese***) OS8* (VSJ OS7): Physiological/Biological fluid dynamics Session Chair: Kenji Kikuchi (Tohoku University)
	J39 Behaviors of a Pair of Vortex Ring Formed by a Flapping Butterfly *Sei Haishi and Masaki Fuchiwaki*
	J73 Blood flow visualizations with particle image velocimetry and computational fluid dynamics analysis for therapy of cardiovascular diseases *Shuya Shida, Toru Masuzawa, Masahiro Osa and Makoto Ohta*
	J74 Perception of Tone Burst Generated by Animals *Kazuki Sugiyama, Yoshihiro Kubota and Osamu Mochizuki*
	J75 Shape of Water Splash by Object Impinging to Water Surface *Takashi Yasui and Yoshihiro Kubota*
	Room F
	Technical Sessions F5 OS7: Pipe flows, channel flows and internal flows Session Chair: Takayuki Yamagata(Niigata University)
	52 VISUALIZATION-BASED MEASUREMENT OF FLOW VELOCITY IN A MOLDED PIPE WITH MATCHED REFRACTIVE INDEX *Shunya Ishii, Katsuaki Shirai and Sakura Fukushima*
	63 IMAGE PROCESSING OF FLOW VISUALISATION PICTURES TO DETERMINE THE STRUCTURE OF TRANSITIONAL CHANNEL FLOW OF AQUEOUS POLYMER SOLUTIONS *Takato Okuda, Xin Song, Sattaya Yimprasert, Per Henrik Alfredsson and Masaharu Matsubara*
	104 EFFECT OF CONTTAINER SIZE ON FLOW STRUCTURE IN A RECTANGULAR CONTAINER INJECTED AIR FROM A SLIT NOZZLE *Takuto Ichihara, Takahiro Kiwata, Kuniaki Toyoda, Hiroaki Uchida, Yasuharu Kawamura and Masamichi Tsuji*
	49 NUMERICAL ANALYSIS OF INTERNAL FLOW IN A SEPARATE HERRINGBONE PLATE HEAT EXCHANGER *Daiki Fujiwara and Shuichi Torii*
	Room G
	Technical Sessions G5 (***in Japanese***) OS2*(VSJ OS2): Measurement Technique Noninvasive Measurement Technique Session Chair: Koichi Nishino(Yokohama National University)
	J34 Visualization of die-cast fluid behavior using PIV analysis *Kousuke Ouchi, Yuki Kasiwabara and Yuta Sunami*
	J57 Visualization of Hydrogen Partial Pressure Distribution by Raman Imaging *Junpei Yamamoto, Chiaki Mizutani, Koji Ogawa, Hironori Ohira, Masashi Maeda and Tsutomu Hosoi*
	J70 Near-infrared visualization of water vapor distributions in air: flow analysis using a block matching method *Naoto Kakuta and Shintaro Ozawa*
	Room H
	Technical Sessions H5 (***in Japanese***) OS1*(VSJ OS3): Luminescence-based Flow Measurement for Thermofluid Analysis Session Chair: Yasushiro Egami(AITECH)
	J24 Effects of Antioxidants on Photodegradation of the Dual-Luminophore Pressure-Sensitive Paint *Kazuki Uchida, Yuta Ozawa, Keisuke Asai and Taku Nonomura*
	J32 Development for simultaneous mesurement of pressure and temperature using luminescent paint *Maito Kanehiro, Koshiro Tsutsumi, Takuma Moriya, Mizue Munekata and Hiroyuki Yoshikawa*
	J50 Mathematical optimization processing for pressure-sensitive paint data *Koyo Kubotam Tomoki Inoue, Tsubasa Ikami, Yasuhiro Egami Hiroki Nagai and Yu Matsuda*
JST(UTC+9)18:30-20:00	**VSJ Banquet**

joint session with VSJ

Day3	JST(UTC+9) Aug.10, 09:00-15:00
	Room A
JST(UTC+9) Aug.10 09:00-9:40	**Plenary Lecture Ⅲ** Session Chair: Satoshi Someya (AIST) Be Ambitious ! to use Open-source software for CFD and flow visualization *Nobuyuki Oshima* (Hokkaido university)
JST(UTC+9) Aug.10 10:00-12:00	**Room F**
	Technical Sessions F6 GS7: Vortex Session Chair: Aref Afsharfard(Pusan National University, Ferdowsi University of Mashhad)
	3 EXPERIMENTAL STUDY ON VORTEX WIDTH AND HYSTERESIS EFFECT OF THE FLOW BETWEEN TWO ROTATING CYLINDERS WITH HIGH ASPECT RATIO *Yuki Sugimoto, Tomoki Marumiya, Haruki Inatani, Katsuya Hirata and Takashi Noguchi*
	25 Unsteady Flow Instability between Two Parallel Circular Plates in Power Generator *Naho Kishizawa, Itsuro Honda, Naohisa Takagaki and Osamu Kawanami*
	53 Differences in Flow Around Golf Balls with Different Dimple Occupancy, Volume Ratio and Depth *Kohei Moriyama and Hiroo Okanaga*
	57 Experiments on flow field control using an acoustic flow generator device *Hideki Kawashima, Shigeyuki Miyazaki and Takamichi Hiroi*
	76 HEAD-ON VORTEX RING COLLISIONS UPON SMALL LATTICE POSTS *Daniel T. H. New, Chuanhua Liu, Darius Koi Yik Tham, Bowen Xu and Haotian Li*
	78 EXPERIMENTAL INVESTIGATION ON THE FLOW STRUCTURE OF BUOYANCY VORTICES USING PARTICLE IMAGE VELOCIMETRY *Ziyang Wang, Neil A. Hawkes, Michael MacDonald, John E. Cater and Richard G.J. Flay*
	Room G
	Technical Sessions G6 GS6: Jet Session Chair: Masaki Fuchiwaki(Kyushu Institute of Technology)
	86 Visualization and measurement of internal flow in a thin vortex chuck research on improvement of silicon wafer adsorption performance *Fukaya Naoyuki and Funatani Shumpei*
	32 Influence of synthetic jet on multiscale features in wall-bounded turbulence *Biaohui Li and Nan Jiang*
	55 PRELIMINARY STUDY OF MICROJETS FROM RECTANTULAR CONVEREGNT NOZZLES WITH HIGH-ASPECT RATIOS *Shota Yoshimi, Sinichiro Nakao and Yoshiaki Miyazato*
	58 PRELIMINARY STUDY OF UNDEREXPANDED SONIC JETS FROM ELLIPTIC CONVERGENT NOZZLES *Tatsuya Nagata, Muhammad M.Islam, Takeshi Miyaguni, Shinitiro Nakao and Yoshiaki Miyazato*
	75 DETECTION OF LARGE-SCALE STRUCTURES IN A TEMPORAL ROUND JET USING MULTIDIMENSIONAL DYNAMIC MODE DECOMPOSITION *Mamoru Takahashi, Ren Fukui, Koichi Tsujimoto, Toshitake Ando and Toshihiko Shakouchi*
	Room H
	Technical Sessions H6 (***in Japanese***) OS3*(VSJ OS6): Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering Session Chair: Hiroshige Kikura(Tokyo Institute of Technology)
	J63 Fundamental Study on 3D Shape Information Acquisition using Monocular Camera and Ultrasonic Sensor *Kazuya Yasui, Takeshi Moriya, Hideharu Takahashi and Hiroshige Kikura*
	J68 UVP measurement of near-wall in Taylor-Couette flow with small aspect ratio *Yoshihiko Oishi, Hideki Kawai and Hiroshige Kikura*
	J85 Bubble sizing in bubbly pipe flow using ultrasound echography *Yuichi Murai, Hyun Jin Park, Yuji Tasaka and Shintaro Akasaka*
	J90 Visualization of continuously rising bubbles in gallium alloy *Sana Maeda, Hideki Murakawa, Fu Xin and Kazushi Adachi*
	Room I
	Technical Sessions I6 GS2: Multiphase flow (Droplet) Session Chair: Nao Ninomiya (Utsunomiya University)
	8 In-Situ Real-Time Measurement of Droplets Dynamics Using a High-Speed Streaming Camera *Tenshiro Ichimura, Chihiro Inoue and George Kuwabara*
	10 The Effect of Curvature Distortion on Internal Flow Measurement of a Levitated Droplet using PIV *Eugene Gatete, Kaneko Akiko and Shen Biao*
	111 VISUALIZATION STUDY ON DROPLET ELECTROHYDRODYNAMIC DEFORMATION IN COMBINED DC ELECTRIC FIELD AND SHEAR FLOW *Guanqiu Hao, Xiangdong Liu and Yongping Chen*
	18 Phase-shifting ellipsometer based on pixelated polarization camera to measure nano-thick liquid film and nanoparticle deposits *Eita Shoji, Taiga Saito, Akira Hoshino and Tetsushi Biwa*
	90 Visualization study of nano particle behavior within pores *Yusaku Abe and Yu Matsuda*
	99 NANOPARTICLES ACCUMULATION IN A PREDETERMINED POSITION UNDER AC ELECTRIC FIELD *Ahmed Abdelghany, Yuko Okui, Yoshiyasu Ichikawa and Masahiro Motosuke*

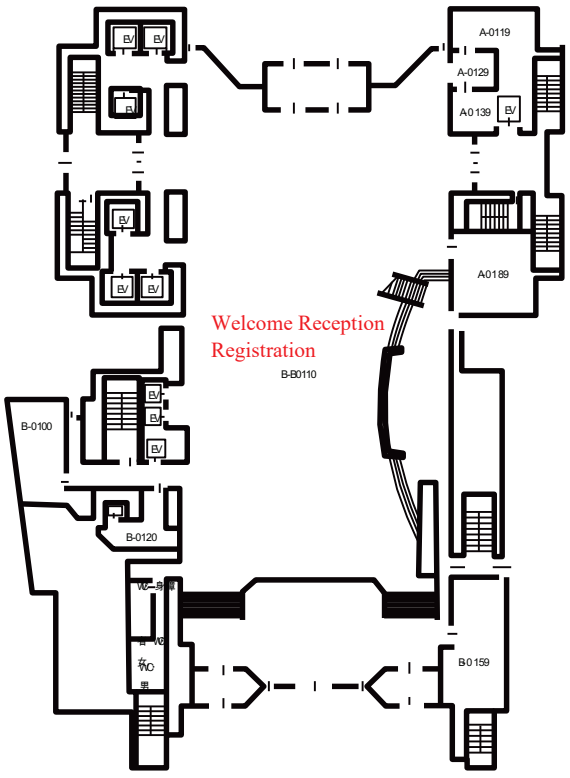
Day3	joint session with VSJ
JST(UTC+9) Aug.10 13:00-15:00	**Room F**
	Technical Sessions F7 GS7: Vortex Session Chair: Yasufumi Horimoto(Hokkaido University)
	59 OSCILLATORY CHARACTERISTICS OF SHOCK TRAINS IN SQUARE DUCTS *Takato Inadomi, Ryota Fukunaga, Taishi Takeshita, Shinichiro Nakao and Yoshiaki Miyazato*
	93 STUDY THE EFFECT OF DOWNSTREAM BLUFF BODY TO ENHANCE FIV-BASED ENERGY HARVESTING, A VISUALIZATION STUDY *Sajjad Hosseini, Aref Afsharfard and Kyung Chun Kim*
	94 FSI-BASED DESIGN OF A DVA FOR DECREASING UNDESIRED VIBRATIONS OF A ROTATING BEAM, A VISUALIZATION STUDY *Amirreza Shahsavari, Aref Afsharfard and Kyung Chun Kim*
	96 STEREO-PIV MEASUREMENT OF A BI-STABLE WAKE BEHIND A SIMPLIFIED SEDAN-TYPE VEHICLE MODEL IN A CROSSWIND *Chiharu Hasegawa, Takuji Nakashima, Takenori Hiraoka, Taiga Kanehira, Hidemi Mutsuda, Keigo Shimizu and Yusuke Nakamura*
	5 ACTIVE CONTROL FOR DRAG REDUCTION OF A CIRCULAR CYLINDER BASED ON REINFORCEMENT LEARNING: PIV MEASUREMENTS OF THE CYLINDER WAKE *Alessandro Scala, Gerardo Paolillo, Carlo Salvatore Greco, Tommaso Astarita and Gennaro Cardone*
	Room G
	Technical Sessions G7 GS4: Numerical visualization and image processing Session Chair: Takahiro Tsukahara(Tokyo University of Science)
	43 CFD ANALYSIS AND VISUALIZATION OF CAVITATING BUBBLY FLOW STRUCTURE IN A CENTRIFUGAL PUMP USING MULTI-PROCESS CAVITATION MODEL *Akihisa Yamada, Taiki Takamine, Shin-ichi Tsuda and Satoshi Watanabe*
	67 COMPARISON OF STRAIGHT-SWIMMING PERFORMANCE OF PITCHING WING BETWEEN STEADY AND PERIODIC FLOWS *Yoshitaka Isoda and Yohsuke Tanaka*
	72 Effect of Surface Wettability for Disk Friction on Rotating Disk in Centrifugal Impeller *Kento Sakai, Yumeno Inaba and Kazuyoshi Miyagawa*
	85 UNSUPERVISED DEEP LEARNING OF THE STRUCTURAL CHARACTERISTICS IN A TURBULENT CHANNEL FLOW AT A LOW REYNOLDS NUMBER *Mohammad Javad Sayyari, Jinyul Hwang and Kyung Chun Kim*
	91 Development of Paddlewheel Aerator Performance Predictions Model *Priyambodo Nur Ardi Nugroho, Muh Anis Mustaghfirin, Dwi Sasmita Aji Pambudi, Eky Novianarenti, Dyah Arum Wulandari and Shuichi Torii*
	Room H
	Technical Sessions H7 OS3*: Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering Session Chair: Hideharu Takahashi(Tokyo Institute of Technology)
	124 (***Session Keynote***) EVOLVING ULTRASONIC VELOCITY PROFILER and eUVP *Yasushi Takeda, Erich Windhab and Hiroshige Kikura*
	119 FUNDAMENTAL RESEARCH ON COMBINATION OF SOLID-LIQUID LEVEL MEASUREMENT AND ELEMENTAL COMPOSITION ANALYSIS USING ULTRASOUND AND LASER *Yuan Chen, Naruki Shoji, Zeliang Zhang, Tran Tri Vien, Hideharu Takahashi and Hiroshige Kikura*
	120 Remote Leakage Flow Visualization with Arm-Type Robot and Three-Dimensional Vector UVP *Naruki Shoji, Takeshi Moriya, Yuan Chen, Hideharu Takahashi and Hiroshige Kikura*
	121 Visualization of Fuel Debris Shape and Measurement of Water Level using Parametric Ultrasound *Tri Vien Tran, Hiroto Tanabe, Hideharu Takahashi and Hiroshige Kikura*
	122 RANGE EXTENSION ON ULTRASONIC MEASUREMENT OF BUBBLY FLOW VELOCITY PROFILE *Wongsakorn Wongsaroj, Hideharu Takahashi, Natee Thong-Un, Weerachon Treenuson and Hiroshige Kikura*
	123 Measurement of Multiphase Flow in Rotating Field by Ultrasonic Velocity Profiler *Takumi Konno, Yoshihiko Oishi, Hideki Kawai and Sho Osanai*

Floor Map

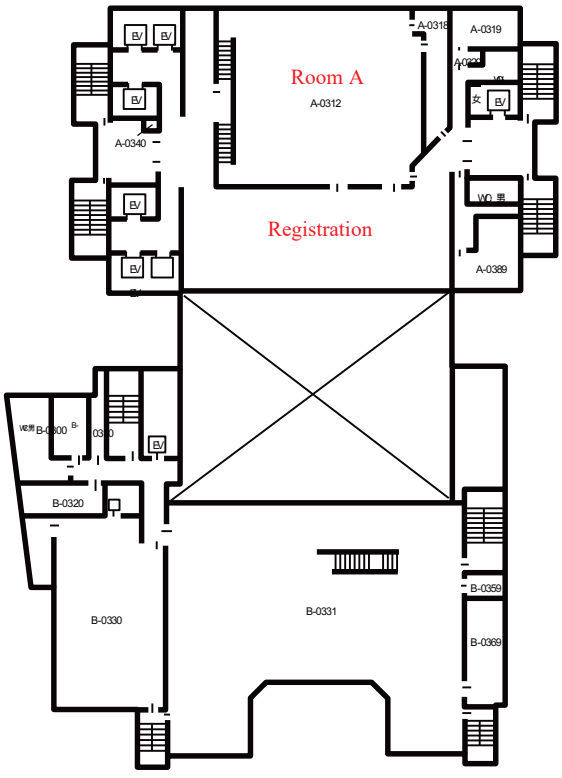
B1 Floor

- Please enter Kogakuin University from the entrance of the underground passage on B1 floor
  All other entrances are closed
- Registration
  1st Floor on Aug. 7
  3rd Floor on Aug. 8, 9, 10
- For preventing COVID-19 outbreaks
- ✓ Wearing a mask
- ✓ Disinfect hands with alcohol
- ✓ No entry to the premises if body temperature is 37.5 C or higher
- ✓ Eating and drinking are not allowed except in designated areas
- ✓ When eating or drinking, eat silently
- ✓ Avoid the “Three Cs”!
  1.Closed spaces
  2.Crowded places
  3. Close contact settings

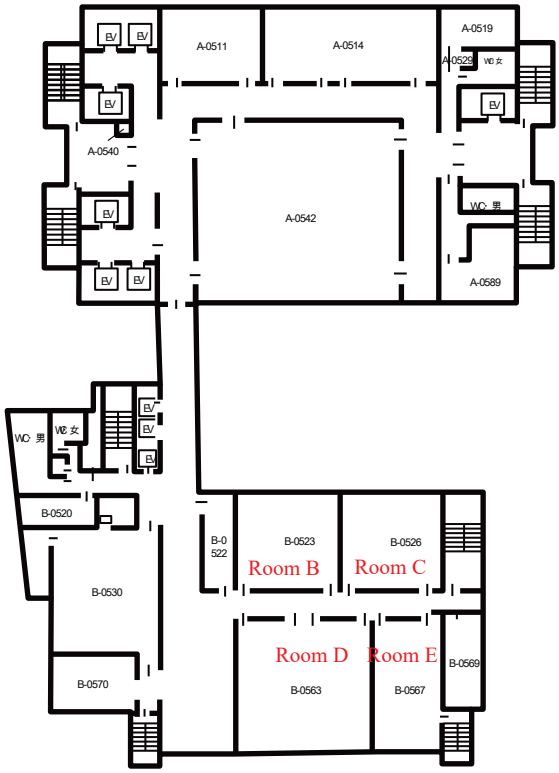
1st Floor



3rd Floor



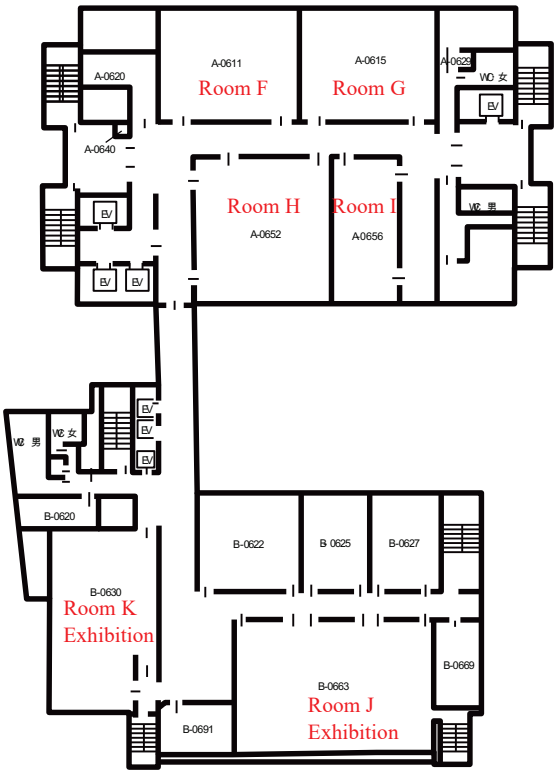
5th Floor



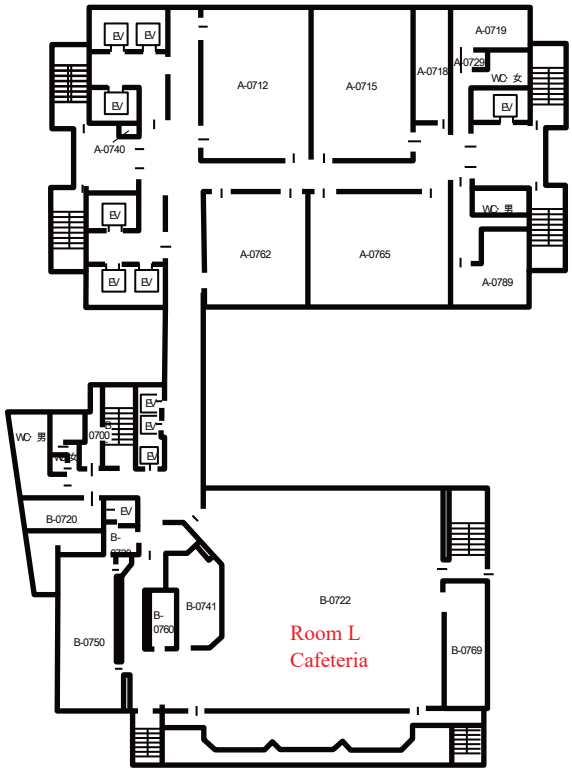
- Plenary / VSJ Memorial lecture session
  Room A
- VSJ Technical session
  Room B, C, D, E
- PSFVIP13 Technical session
  Room F, G, H, I
- Corporate Exhibition
  Room J, K
- Refreshments during the breaks
  Room J, K
- Lunch on 12:00-13:00, Aug. 9, 10
  Room L
- VSJ banquet on 18:30-20:00, Aug. 9
  (Free of charge)
  Room L

* PSFVIP13 participants able to attend all the plenary and the technical sessions including VSJ symposium

6th Floor

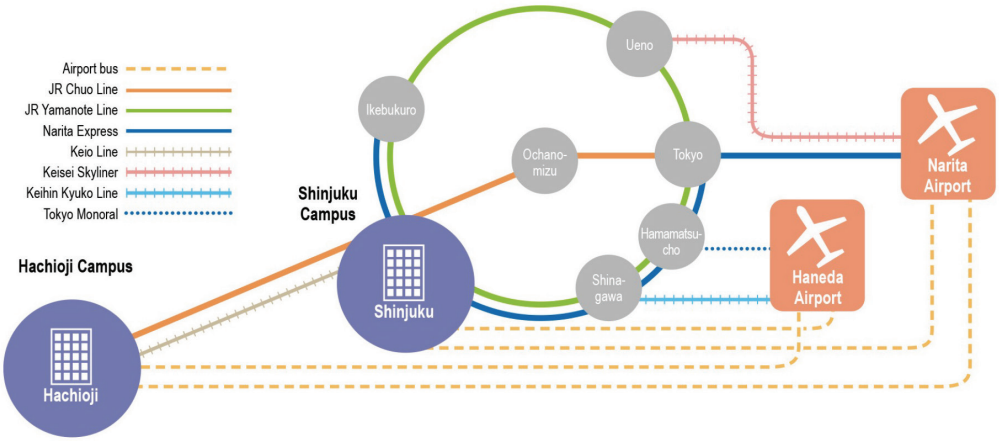


7th Floor



Kogakuin University Access

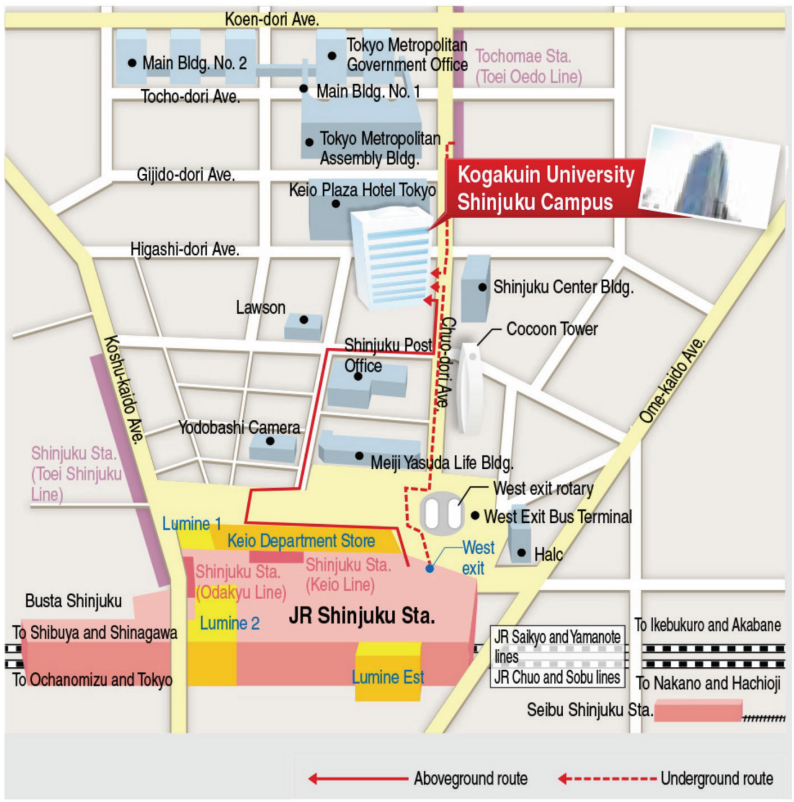
from Airport



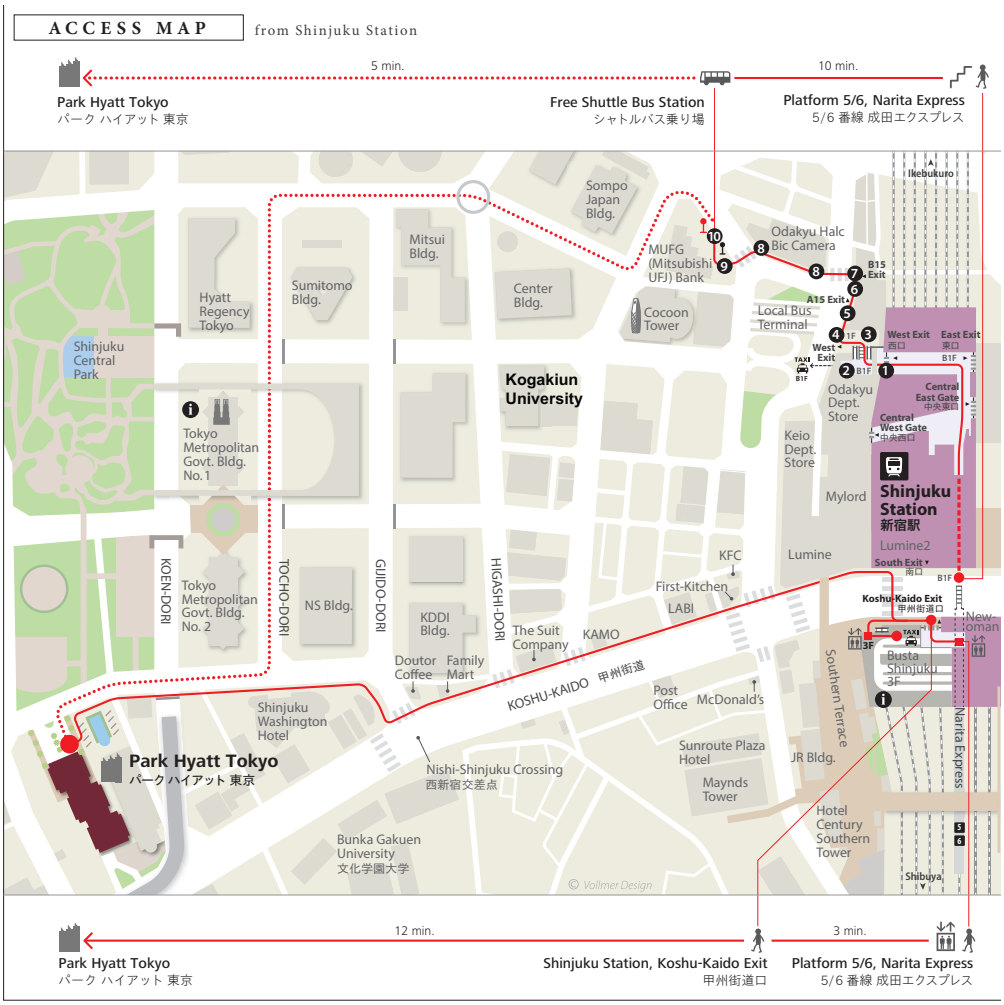
Narita Airport	Narita Express (approx. 80 min.)		Shinjuku
Narita Airport	Keisei Skyliner (approx. 60 min.)	Ueno	JR Yamanote Line (approx. 24 min.) Shinjuku
Narita Airport	Airport bus (approx. 85-115 min.)		Shinjuku
Narita Airport	Airport bus (approx. 120-160 min.)		Hachioji
Shinjuku	JR Chuo Line (approx. 46 min.)		Hachioji
Shinjuku	Keio Line (approx. 46 min.)		Hachioji
Haneda Airport	Airport bus (approx. 35-65 min.)		Shinjuku
Haneda Airport	Airport bus (approx. 75-120 min.)		Hachioji
Haneda Airport	Keihin Kyuko Line (approx. 13 min.)	Shinagawa	JR Yamanote Line (approx. 18 min.) Shinjuku
Haneda Airport	Tokyo Monoral (approx. 13 min.)	Hamamatsucho	JR Yamanote Line (approx. 21 min.) Shinjuku

from Station

Shinjuku Campus



Park Hyatt Tokyo Access



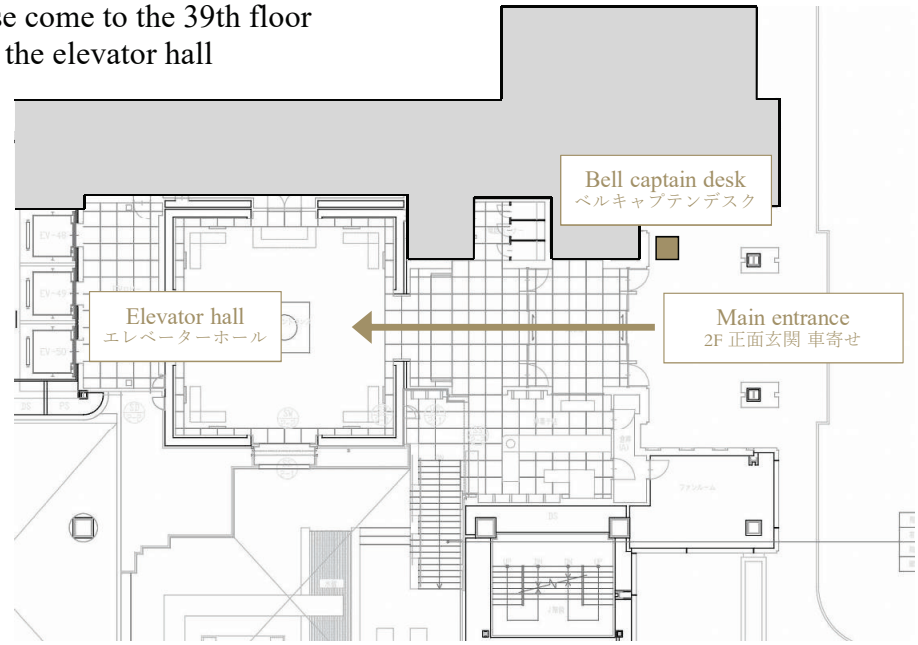
Travel times are indicative. 所要時間は目安です。

Google Map https://goo.gl/ZPpLqC

2F Entrance Map
2階 ホテルエントランスのご案内

The Ballroom

Please come to the 39th floor
from the elevator hall



GAS HYDRATE FORMATION AND DISSOCIATION UNDER STATIC AND DYNAMIC CONDITIONS

*Y. Nakajima[1], J. Yamamoto[1], S. Takano[1], M. Yamamoto[1], M. Ono[1], S. Kanada[1] and K. Otsubo[1]

[1] National Maritime Research Institute, 6-38-1 Shinkawa, Mitaka, Tokyo 181-0004 Japan
*nakajima@m.mpat.go.jp

ABSTRACT

Gas hydrates are ice-like crystalline solids consisting of water and guest molecules such as methane and carbon dioxide (CO_2), and thermodynamically stable under the high-pressure and low-temperature conditions like permafrost and deep seabed. [1] In Japan, methane hydrate has been found under the seafloor in the Exclusive Economic Zone so that it is expected as an unconventional natural gas resource. [2] The authors have investigated the characteristics of formation and dissociation of gas hydrates by visual observation. [3] In this study, the formation and dissociation processes of gas hydrates at both static and dynamic conditions were observed by using a probe-type microscope camera.

In the experiments at static condition, a single pressure chamber was half filled with water, and pressurized by injection of CO_2 gas up to the pressure of ca. 4MPa. While it was cooled to ca. 2 deg. C, the gas injection into the chamber was continued to keep the inner pressure. Gas hydrate formation was observed in two stages. In the early stage, although initial gas hydrate formation was observed on the surface of liquid films on the window, bulk water in the chamber was expected to be oversaturated by dissolved CO_2. As the gas-liquid interface was continuously disturbed, excess amount of gas dissolved in water was consumed to form massive gas hydrates in the chamber. As the pressure was lowered, gas hydrates were dissociated to form gas bubbles.

At dynamic condition, two pressure chambers were connected with a straight tube to form a tubular vessel. The vessel was settled on a seesaw-type platform so that filling water in the vessel can flow by moving the board of the platform. Two microscope cameras were mounted on both pressure chambers of the vessel. Gas hydrate formation was initialized in a chamber, then, the vessel was fluctuated on the platform to observe the behaviour of the hydrates, which was analyzed by image analysis.

REFERENCES

[1] Sloan, E.D. and Koh, C.A., Clathrate Hydrates of Natural Gases, 3rd Edition (2007), CRC Press.

[2] Matsumoto, R., Ryu, B.-J., Lee, S.-R., Lin, S., Wu, S., Sain, K., Pecher, I. and Riedel, M., “Occurrence and Exploration of Gas Hydrate in the Marginal Seas and Continental Margin of the Asia and Oceania Region”, Marine and Petroleum Geology, Vol. 28, No. 10 (2011), pp. 1751-1767.

[3] Nakajima, Y., Yamamoto, J., Takano, S., Yamamoto, M., Ono, M., Kanada, S. and Otsubo, K., “Observation of Gas Hydrate Formation at Gas-Liquid Interface”, Proc. International Conference on Power Engineering-2021 (2021).

EXPERIMENTAL STUDY ON VORTEX WIDTH AND HYSTERESIS EFFECT OF THE FLOW BETWEEN TWO ROTATING CYLINDERS WITH HIGH ASPECT RATIO

*Y. Sugimoto[1], T. Marumiya[1], H. Inatani[1], K. Hirata[1] and T. Noguchi[1]

[1] Doshisha University, 1-3 TataraMiyakodani, Kyotanabe City, Kyoto, Japan
*ctwg0572@mail4.doshisha.ac.jp

ABSTRACT

In this study, we experimentally investigate the influence of the wavenumbers of the Taylor vortex flow TVF and the wavy vortex flow WVF upon the borders between the regime of the Couette flow CF and the TVF regime and between the TVF regime and the WVF regime. In order to obtain the steady state mode with an expected wavenumber at an expected condition, we use a control method based on the hysteresis effect by the acceleration of the inner cylinder rotation.

Flow is visualized by the aluminum powder suspension method. Main dimensions of the present experimental setup are 40 mm in the inner cylinder radius, 45 mm in the outer cylinder radius, and 945 mm in the cylinder height.

At first, we established the control method of the wavenumber. More specifically, at the outer cylinder Reynolds number R_o = 200 and the inner cylinder Reynolds number R_i = 800 in the WVF regime, we can control the normalized wavenumber $2\pi/\lambda^*$ in a range of 2.71-3.41. At R_o = 500 and R_i = 700 in the TVF regime, we can control $2\pi/\lambda^*$ in a range of 3.14-3.53. In the present control, $2\pi/\lambda^*$ tends to decrease with increasing rotational acceleration of the inner cylinder. Next, we show the influence of $2\pi/\lambda^*$ upon the borders, supposing such two initial conditions as (R_o, R_i) = (200, 800) and (500, 700).

As a result, the influence upon the border between the CF regime and TVF regime is negligible. On the other hand, the influence upon the between the WVF regime and is not negligible. More specifically, the smaller the wavenumber is, the stronger the influence of it upon the border is.

REFERENCES

[1] G. I. Taylor, Philos. Trans. R. Soc., Ser. A, 223, 289 (1923).
[2] H. Furukawa, T. Watanabe and I. Nakamura - Proc. 9th Int. Symp. Flow Visual, 715 (2000).
[3] C. D. Andereck, S. S. Liu and H. L. Swinney, J. Fluid Mech. Vol.164, 155-183 (1986).

INVESTIGATION OF THE INFLUENCE OF VASCULAR ACCESS SHAPE ON HEMODYNAMIC PARAMETERS

*H. Anzai[1], K. Takeda[1,2], A. Kajiyama[2], and M. Ohta[1]

[1] Institute of Fluid Science, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi 980-8577 Japan
*hitomi.anzai.b5@tohoku.ac.jp
[2]Graduate School of Biomedical Engineering, Tohoku University, 6-6 Aramaki-Aza-Aoba, Aoba-ku, Sendai, Miyagi, 980-8579, Japan

ABSTRACT

Arteriovenous fistulas (AVFs) have been recognized as the gold standard of vascular access for hemodialysis. However, the development of postoperative stenosis is a serious problem. Although it is widely known that hemodynamic stress affects vascular lesions, there are few reports on hemodynamic changes due to AVF geometry, except for the anastomotic angle between the artery and vein.

Recently, devices that support blood vessels from the outside have been developed [1]. This has made it possible to maintain a more arbitrary vessel shape after AVF construction. The purpose of this study is to clarify the effect of vessel shape on hemodynamics when wearing a support device through parametric study using computational fluid dynamics (CFD).

An AVF model with an ideal shape with straight and curved anastomoses was constructed. The anastomotic angle α, shunt vessel height H_F, x-axis angle β, anastomotic curvature R_A, and anastomotic height H_A were varied, respectively. Time-averaged wall shear stress (TAWSS), osciratolly shear index (OSI), and time-averaged vorticity (TAV) were calculated from the transient analysis using pulsatile waveforms.

The results of CFD analysis for AVF placement show the idealized models with larger α or large R_A had smaller blood flow disturbance. Furthermore, uniform time-averaged WSS distribution was generated on these models. A uniform WSS distribution is considered to preserve the shape of the vascular lumen and reduce the risk of the vascular lesion such as stenosis. The operation of vascular shape may improve the hemodynamics which may affect the onset of vascular lesions such as stenosis.

REFERENCES

[1] Chemla, E., Velazquez, C., D′Abate, F., Ramachandran, V., and Maytham, G., "Arteriovenous fistula construction with the VasQ™ external support device: a pilot study", The Journal of Vascular Access, Vol., 17, No., 3(2016), pp. 243-248.

ACTIVE CONTROL FOR DRAG REDUCTION OF A CIRCULAR CYLINDER BASED ON REINFORCEMENT LEARNING: PIV MEASUREMENTS OF THE CYLINDER WAKE

A. Scala, G. Paolillo, C.S. Greco, T. Astarita and G. Cardone

University of Naples "Federico II", Via Claudio 21, Naples, 80125, Italy
*gennaro.cardone@unina.it

ABSTRACT

The aim of the present work is to investigate, experimentally via Particle Image Velocimetry, the fluid dynamic characteristics of a circular cylinder wake, which is controlled by a synthetic jet in order to reduce the related aerodynamic drag.
The circular cylinder, placed in an open-circuit wind tunnel, presents a slot in its rear position, through which the synthetic jet is issued [1]. The latter is obtained by coupling the hollow interior of the cylinder with a loudspeaker driven by an electrical signal, which is generated using a wave generator coupled with a power amplifier. The aerodynamic drag is measured by two load cells, located at the basis of the cylinder, the output signals of which are amplified and then acquired through an Arduino card.
Reinforcement Learning [2-3] is applied to find the electrical signal waveshape, supplied to the loudspeaker, that offers the largest reduction of the cylinder aerodynamic drag. In particular, two different shapes are tested: those obtained as Fourier series at a chosen fundamental frequency and those obtained by varying the duty cycle of a sinusoidal signal. The latter are found to reduce more the cylinder aerodynamic drag.
Starting from the identified optimal waveshape, the use of Particle Image Velocimetry allows to obtain the instantaneous two-dimensional velocity fields in a plane containing the synthetic jet slot; these measurements are used to investigate the mean flow quantities and turbulent statistics of the phenomenon.

REFERENCES

[1] Greco, C.S., Paolillo, G., Astarita, T., and Cardone, G., "The von Kármán street behind a circular cylinder: Flow control through synthetic jet placed at the rear stagnation point", Journal of Fluid Mechanics, 901 (2020).

[2] Fan, D., Yang, L., Wang, Z., Triantafyllou, M.S., and Karniadakis, G.E., "Reinforcement learning for bluff body active flow control in experiments and simulations", Proceedings of the National Academy of Sciences, 117(42), 26091-26098, (2020).

[3] Brunton, S.L., Noack, B.R., Koumoutsakos, P., "Machine learning for fluid mechanics", Annual Review of Fluid Mechanics, 52, 477-508, (2020).

START-UP CHARACTERISTICS OF LOOP HEAT PIPE WITH PLATE-TYPE EVAPORATOR

*Tomohiro Nakazawa[1], Atsushi. Tsujimori[2], Tsubasa Usui[1], Koji Ono[1],
Reo Miwa[1], Norifumi Miyanaga[2], Yui SATO[2] and Haruka Suzuki[2]

[1] Graduate School, Kanto Gakuin University, 1-50-1 Mutsuurahigashi, Kanazawa, Yokohama, Kanagawa 236-8501, Japan
tujimori@kanto-gakuin.ac.jp
[2] Kanto Gakuin University, 1-50-1 Mutsuurahigashi, Kanazawa, Yokohama, Kanagawa 236-8501, Japan

ABSTRACT

In this study, experimental equipment with a plate-type evaporator was manufactured and its start-up characteristics were investigated. The evaporator consisted of an upper part, a porous carbon wick, and a lower part. The upper part was used as the reservoir for the liquid refrigerant. In the lower part, grooves for the vapor paths were machined and were lined with porous carbon with a pore diameter of 5 μm. To visualize the flow of the working fluid in the vapor paths of the evaporator during start-up, the lower part of the evaporator with grooves was made of a polycarbonate resin plate. In the experiment, a heat load was applied to the evaporator from the bottom side and the refrigerant in the vapor paths was observed. The experimental results show that it took several minutes for the liquid refrigerant in the vapor paths to start evaporating and about 30 s for the vapor paths to be filled with refrigerant vapor. Moreover, the vaporization started from the fluid-merging part and then expanded to the branches of the vapor paths. We also constructed a calculation model for only the vapor paths, and the calculation results show that the two-phase flow flows out to the vapor line as in the experimental results, but the vaporization starts from the edge of the branches of the vapor paths. It is considered that in the experiment, the capillary wick was set on the vapor paths in the evaporator and the liquid refrigerant in the vapor paths was pushed by the generated refrigerant vapor and flowed out not only to the vapor line but also to the reservoir through the capillary wick, which caused the difference between the calculation and experimental results.

REFERENCE

[1] K. Takenouchi, A. Tsujimori, M. Maruoka and K. Togashi, “Start-up Characteristics of Loop Heat Pipe”, 5th International Conference on Design Engineering and Science, (2020), pp. 117–122.

VISUALIZATION OF INTERMITTENT TURBULENT STRUCTURE IN ECCENTRIC ANNULAR PIPE FLOW

*K. Yatsushiro[1], Y. Horimoto[2] and T. Tsukahara[1]

[1] Tokyo University of Science, 2641 Yamazaki, Noda, Chiba 278-8510 Japan
*7521555@ed.tus.ac.jp
[2] Hokkaido University, N13 W8, Sapporo, Hokkaido 060-8628 Japan

ABSTRACT

Annular pipe flow is common in industrial applications and has attracted researchers' interest because of the variety of its intermittent turbulent states, i.e., localized turbulence, for the concentric annular flow [1]. The eccentric annular pipe flow and its heat and mass transfers are also practically important, since the actual equipment is more or less eccentric. A recent study [2] revealed that the critical Reynolds number of a linear instability and the quasi-periodic coherent structures were significantly dependent on the eccentricity. The purpose of this study is to experimentally investigate the dependences of the eccentricity and the radius ratio on the localized turbulence and coherent structures in a transitional Reynolds-number regime.

We used an apparatus consisting of one fixed outer pipe and radially-movable inner pipe, whose diameters are respectively D_o = 14 mm and D_i = 2–6 mm corresponding to the radius ratio of $\gamma \equiv D_i/D_o$ = 0.14–0.43. The eccentricity is defined as $\varepsilon = 2d/D_h$, where d is the distance between the pipe axes and D_h (= $D_o - D_i$) is the hydraulic equivalent diameter. We examined four ε of 0 (concentric) and 0.2–0.8 (eccentric). The pipe length is 2600 mm (> $210D_h$). For visualization with light-reflective mica flakes, we used two mirrors that allow us to observe the entire flow simultaneously from 3 azimuthal directions to grasp the stereoscopic structures. We determined the transition processes of flow states from developed turbulence to laminar, with detecting localized turbulent regions.

In the concentric annular pipe flow, we confirmed the presence of helical pattern of localized turbulence, as Ishida et al. [1] demonstrated only in their numerical simulations. The helical shape was successfully demonstrated by our multi-directional observation. Our experiments revealed that the increase in the radius ratio decreased the Reynolds number of relaminarization. Interestingly, intermittent turbulent structures appeared at very low Reynolds numbers for the eccentric cases, regardless of the radius ratio. The localized turbulence for ε = 0.2–0.4 survived at much lower Reynolds numbers that the critical value for ε = 0.

REFERENCES

[1] Ishida, T., Yohann, D. and Tsukahara, T., "Transitional structures in annular Poiseuille flow depending on radius ratio", Journal of Fluid Mechanics, Vol. 794 (2016), R2.

[2] Lamarche-Gagnon, M.-É and Tavoularis, S., "Further experiments and analysis on flow instability in eccentric annular channels", Journal of Fluid Mechanics, Vol. 915 (2021), A34.

IN-SITU REAL-TIME MEASUREMENT OF DROPLETS DYNAMICS USING A HIGH-SPEED STREAMING CAMERA

*T. Ichimura[1], C. Inoue[1], G. Kuwabara[2]

[1] Kyushu University, 744 Motoka, Nishi-ku, Fukuoka 819-0385 Japan
*ichimura.tenshiro.489@s.kyushu-u.ac.jp
[2]Photron Ltd., Kandajimbocho, Chiyoda-ku, Tokyo 101-0051 Japan

ABSTRACT

High speed imaging is a beneficial approach to clarify time-resolved multi-phase flow, such as droplet dynamics in a spray. However, it has been necessary to separate the image capturing step and the analysis step for quantitative measurement; thus, it has been recognized that the high-speed imaging is distinct from a real-time measurement technique. Real-time high-speed measurement for a spray is suitable for inspection of products and for feedback control of flow system. Recently, a high-speed streaming camera is developed, which can take images over 1kHz and translate them to a PC in a simultaneous manner, potentially allowing for the high-speed real-time measurement of multi-phase flow.

We employ a high-speed streaming camera (Photron INFINICAM UC-1), controlled by in-house C++ program implementing the image processing library of OpenCV, achieving the non-tracking measurement of drop size distribution and the Particle Tracking Velocimetry (PTV) for drop size and velocity. A laptop PC (MSI Delta15) is used for the in-situ measurement.

We successfully quantify diameters for the number of 10,000 droplets and deduce the size distribution within approximately 1 second, achieving a processing speed of 2.5 kHz at the imaging framerate of 1kHz. Then, we measure diameter and velocity of a single free-falling droplet. Time-step for the PTV analysis considers the sequence number tagged to each image to avoid calculation of missing images. The processing speed is greatly improved by using the velocity information of the droplet to restrict the search are. The PTV is successfully conducted in real-time at a processing speed of over 1kHz. The measurement accuracy is calculated by comparing the measured droplet velocity with the theoretical free-fall velocity, convincing that the measurement accuracy is more than 90% for a droplet without deformation.

REFERENCES

[1] Fischer, A., “Fundamental flow measurement capabilities of optical Doppler and time-of-flight principles”, Experiments in Fluids, Vol. 62, No. 2 (2021).

[2] Shaffer F., Ibarra E., and Savaş Ö., “Visualization of submerged turbulent jets using particle tracking velocimetry”, Journal of Visualization, Vol. 24, No.4 (2021), pp.699-710.

COMPARISON OF HEMODYNAMICS IN LEFT VENTRICLES WITH TRICUSPID AORTIC VALVE AND BICUSPID AORTIC VALVES

*Shingo Tsuda[1,2], Suguru Miyauchi[3], Kenichi Funamoto[1,2,4]

[1] Graduate School of Engineering, Tohoku University, 6-6-01 Aramaki-aza Aoba, Aoba-ku, Sendai, Miyagi 980-8579, Japan
*tsuda.shingo.t1@dc.tohoku.ac.jp
[2] Institute of Fluid Science, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi 980-8577, Japan
[3]Faculty of Engineering, University of Miyazaki, 1-1 Gakuen Kibanadai-nishi, Miyazaki, Miyazaki 889-2192, Japan
[4]Graduate School of Biomedical Engineering, Tohoku University, 6-6-12 Aramaki-aza Aoba, Aoba-ku, Sendai, Miyagi 980-8579, Japan

ABSTRACT

The aortic valve between the left ventricle (LV) and the aorta is generally a tricuspid valve (TV) with three cusps, but it is a congenital bicuspid valve (BV) with only two cusps for 0.5%-1.0% in the whole population. The BV reportedly increases a risk of aortic diseases, such as aortic dissection and aneurysm, by causing a jet flow of blood [1, 2]. However, the detailed changes by aortic valve morphology on hemodynamics remain to be understood. The purpose of this study was to clarify the effect of BV on hemodynamics in the LV located at the upstream of the aorta. We constructed a LV model from magnetic resonance imaging of the LV in a healthy volunteer, adding simplified internal structures of trabeculae carneae and papillary muscles and assuming a deformation during a pulsation [3]. A TV or two types of BV with different opening shapes and areas were installed at the aortic valve position [2, 4]. Numerical simulation of blood flow was then performed with the three LV models to compare the velocity field, vortex structure, and hemodynamic parameters such as wall shear stress (WSS). LV models with BVs showed a larger velocity of blood flow that formed a jet flow from the LV to the aorta and a larger time-averaged WSS near the aortic valve than that with TV. These findings support that the BV might contribute to the above-mentioned aortic diseases. As for vortex structures in the LV, no clear difference was observed between the models. But, between the BV models, there was a difference in the flow velocity, resulting from the valve positions even if their opening areas were equivalent. Thus, the aortic valve morphology changes blood flow ejected from the LV, and it is important to employ a model integrated the LV and the aorta for hemodynamic analysis in the aorta.

REFERENCES

[1] Levack, M.M., et al., “Quantitative three-dimensional echocardiographic analysis of the bicuspid aortic valve and aortic root: A single modality approach”, Journal of Cardiac Surgery, Vol. 35, No. 2 (2020), pp. 375-382.

[2] Chandran, K.B., et al., “Patient-specific bicuspid valve dyna-mics: Overview of methods and challenges”, Journal of Biomechanics, Vol. 46, No. 2 (2013), pp. 208-216.

[3] Hosoi, K., et al., “Numerical Analysis of the Effect of the Inner Structure of a Left Ventricle on the Blood Flow Field: Effect of Papillary Muscles and Trabeculae Carneaes”, Proceedings of the 15th International Conference on Flow Dynamics (ICFD2018), (2018), pp. 622-623

[4] Haimerl, J., et al., “Quantification of aortic valve area and left ventricular muscle mass in healthy subjects and patients with symptomatic aortic valve stenosis by MRI”, Zeitschrift fur Kardiologie, Vol. 94, No. 3 (2005), pp. 173-181.

THE EFFECT OF CURVATURE DISTORTION ON INTERNAL FLOW MEASUREMENT OF A LEVITATED DROPLET USING PIV

*Eugene GATETE [1], Akiko KANEKO[2] and Shen Biao[3]
[*1]Department of Engineering Mechanics and Energy, University of Tsukuba
* egateos48@gmail.com and s2130194@u.tsukuba.ac.jp

[2] Department of Engineering Mechanics and Energy, University of Tsukuba, 1-1-1Tennodai, Tsukuba, Ibaraki, Japan
[3] Department of Engineering Mechanics and Energy, University of Tsukuba, 1-1-1Tennodai, Tsukuba, Ibaraki, Japan

ABSTRACT

Visualization of the curvature distorted internal flow in levitated droplets is crucially needed due to the application of droplet dynamics in different areas of engineering, medical sciences, material engineering, and so on. In the present study, an acoustically levitated droplet is focused on and particle image velocimetry (PIV) is used to visualize and measure flow fields inside the droplet as the measurement technique. Although the flow structure in the levitated droplet is required in order to understand the internal flow structure of the droplet, it is limited by the low Web number and curvature distortion when visualization approaches are applied. Researchers have studied droplet levitation technologies, however, there is still less experimental and analytical information about the effect of curvature distortion on the internal flow of a levitated droplet. To understand the flow field inside an acoustically levitated droplet with high accuracy, it is important to experimentally investigate the effect of the parameters that influence the curvature distortion development of the flow inside the droplet. The aim of this research is to visualize and investigate the effect of curvature distortion on the internal flow measurement of a levitated droplet which will lead to the curvature distortion correction. In this study, the effect of curvature distortion on internal flow elucidation was performed based on the refractive index, surface tension, sound pressure, and diameter. The internal flow fields were experimentally investigated and visualized using the PIV. The experimental results showed that the internal flow curvature distorted increases as the refractive index, sound pressure, and surface tension increase.

Keywords: Acoustic levitation, Aspect ratio, Curvature distortion, internal flow, Particle image velocimetry (PIV), and Refractive index.

COMPUTATIONAL MODELING OF THE SPONGE'S CHOANOCYTE CHAMBER

*Takumi Ogawa[1], Toshihiro Omori[1], Kenji Kikuchi[1], and Takuji Ishikawa[1]

[1] Tohoku University, 6-6-01 Aoba, Aramaki, Aoba-ku, Sendai, Miyagi 980-8579 Japan
*takumi.ogawa.t2@dc.tohoku.ac.jp

ABSTRACT

Sponges are the evolutionarily oldest animals among multicellular organisms. They are filter feeders enclosing the choanocyte chamber with tens of flagella, and they can filter a volume of water several times their body volume[1]. The choanocyte chamber consists of uni-flagellated collar cells arranged in a spherical configuration, and the wave of flagellar beating propagates toward the center of sphere. The wave of flagella creates a flow of water within the choanocyte chamber, and nutrients in the water are filtered by the collar around the flagellum for feeding[2]. Though flagellar driven flow in choanocyte chamber is important for the filter feeding, the flow field within the choanocyte chamber has not been clarified. How can a spherically symmetrical flagellar arrangement produce a unidirectional flow? To break the symmetry, inner structures within the choanocyte chamber, such as membrane and cone cell ring[3], are thought to play a role.

In this study, we developed a three-dimensional choanocyte chamber model using a boundary element-slender-body coupling method to clarify the mechanism of the pumping function induced by the flagellar beat. We investigated the flagellar driven flow rate through the outlet of the chamber and the effect of inner structures on the outlet flow rate.

We found that the inner structures contribute to create unidirectional flow by preventing water circulation around the outlet. As a result, the outlet flow rate was enhanced by combination of the membrane and cone cell ring, and it reaches more than two times larger than that of no internal structure model.

REFERENCES

[1] Asadzadef, S.S. *et al.* "Hydrodynamics of the leucon sponge pump", Journal of The Royal Society Interface, Vol. 16, (2019).

[2] Ludeman, D.A. *et al.* "The energetic cost of filtration by demosponges and their behavioural response to ambient currents", Journal of Experimental Biology, Vol. 220, (2017), pp. 4743-4744.

[3] Langenbruch P. *et al.* "Canal systems and choanocyte chambers in freshwater sponges (Polifera, Spongillidae)", Zoomophology, Vol. 107, (1987) pp. 11-16.

MULTISCALE FLOW STRUCTURES IN A CUBE WAKE INFLUENCED BY THE FRONT INCLINED HOLE

*Jiawei Li[1], Xiaolei Han[1], Hiroka Rinoshika[2] and Akira Rinoshika[1,2]

[1] Ministry of Education Key Laboratory of Fluid Mechanics, Beijing University of Aeronautics and Astronautics, Beijing 100191, China
*lijiawei@buaa.edu.cn
[2] Department of Mechanical Systems Engineering, Yamagata University, 4-3-16 Jonan, Yonezawa-shi, Yamagata 992-8510, Japan

ABSTRACT

Time-resolved particle image velocimetry (TR-PIV) is used to experimentally study the multi-scale wake structures of a wall-mounted cube and the effects of a front inclined hole (FIH). The cube model has a side length of D = 50 mm, and the Reynolds number Re_D is 7800. The inlet of FIH is located at the height of 0.71D on the front surface of the cube, and the outlet is located at the center of the free end. The statistical analysis, the orthogonal wavelet multi-resolution analysis, and the vortex dynamics are applied to this study. The statistical results show that FIH can reduce the recirculation zone around the cube, inhibit the downwash flow, and change the development of free-end shear flow. The turbulence intensity and turbulent kinetic energy (TKE) in the wake also decreased significantly due to FIH. Multi-scales flow structures in cube wake are extracted by the wavelet multi-resolution analysis. From the shear layer around the cube to the near wake and then to the far wake, the main structures of the flow field experience the transformation process of small- to middle- and then to large-scale structures. Under the influence of FIH, the scale of the flow structure near the free end is smaller, the large-scale structure in the wake is weakened, and the transition from small- to middle-scales is no longer obvious. The vortex dynamics analysis reveals the existence and evolution of vortex structures with different scales. Because of FIH, the trend moving to the bottom wall of large- and middle-scale coherent structures is weakened, while the existent range in streamwise of small-scale structures has longer. In the large-scale component of wake, we found the transverse secondary vortex, which is formed by the combined action of the shedding transverse vortex and the downwash flow.

REFERENCES

[1] Li, J., Rinoshika, H. and Rinoshika, A., “Effect of front inclined hole on flow structure around a wall-mounted cube”, Experimental Thermal and Fluid Science, Vol. 120 (2021).

[2] Rinoshika, A. and Zhou, Y., “Orthogonal wavelet multi-resolution analysis of a turbulent cylinder wake”, Journal of Fluid Mechanics, Vol. 524 (2005), pp. 229-248.

[3] Li, J., Rinoshika, H. and Rinoshika, A., “Passive control on wake structure of a wall-mounted cube by using a horizontal hole”, Ocean Engineering, Vol. 213 (2020).

[4] Shadden, S.C., J.O. Dabiri and J.E. Marsden, “Lagrangian analysis of fluid transport in empirical vortex ring flows”, Physics of Fluids, Vol. 18, No. 4 (2006).

FUNDAMENTAL STUDY OF MR-MEASUREMENT-INTEGRATED SIMULATION OF HEART-AORTA−SYSTEM: APPLICATION TO AN AORTIC ARCH

*Suguru Miyauchi[1], Madoka Kon[2,3], Toshiyuki Hayase[4]

[1]Faculty of Engineering, University of Miyazaki, 1-1, Gakuen Kibanadai-nishi, Miyazaki, Miyazaki 889-2192 Japan
*suguru.miyauchi@cc.miyazaki-u.ac.jp
[2]Insitute of Fluid Science, Tohoku University, 2-1-1, Katahira, Aoba-ku, Sendai, Miyagi 980-8577 Japan
[3]School of Engieering, Tohoku University, 6-6, Aramaki-aza Aoba, Aoba-ku, Sendai, Miyagi 980-8579 Japan
[4]Frontier Research Institute for Interdisciplinary Sciences, Tohoku University, 6-3, Aramaki-aza Aoba, Aoba-ku, Sendai, Miyagi 980-8578 Japan

ABSTRACT

Since the initiation and progression of cardiovascular diseases are related to hemodynamics, it is important to obtain accurate hemodynamic parameters. Clinical measurements including magnetic resonance (MR) imaging and computational fluid dynamics (CFD) simulation are mainly used to investigate the parameters. MR imaging can obtain information on vascular geometry and blood flow non-invasively, but the accuracy of MR data is relatively low. On the other hand, CFD simulation can obtain detailed information on the blood flow field, but it is difficult to give accurate boundary conditions. To compensate the above-mentioned drawbacks of the MR imaging and CFD simulation, MR-measurement-integrated (MR-MI) simulation has been developed [1]. In the MR-MI simulation, MR data are fed back to numerical simulations to reproduce actual blood flow fields. The effectiveness of the MR-MI simulation was demonstrated for blood flows in descending and ascending aortas [2, 3], which are known as the common sites of the aneurysm. The purpose of this study was to obtain accurate blood flow field in an aortic arch using the MR-MI simulation. In this study, three kinds of simulations were compared: the ordinary CFD simulation in which the flow rate obtained by MRI was given to the inlet boundary condition, the ordinary CFD simulation in which the inlet flow rate was corrected to minimize the difference between the measured and computed mean flow velocities in the direction of the blood vessel axis in the region of interest (ROI) set at the arch, and the MR-MI simulation. As the result, the flow velocity error in the ROI was reduced by the correction of the input flow rate and the feedback of the MR data. In addition, the more complicated temporal-spatial variation in the wall shear stress was reproduced by the MR-MI simulation compared to those of ordinary simulations. These results show the effectiveness of the MR-MI simulation to the blood flow in the aorta.

REFERENCES

[1] Funamoto, K., et al., Annals of Biomedical Engineering, Vol. 37, No. 6 (2009), pp. 1105-1116.

[2] Ogitsu, M., et al., 14th International Conference on Flow Dynamics, (2017), pp. 264-265.

[3] Hori, Y., et al., 16th International Conference on Flow Dynamics, (2019), pp. 444-445.

TIME-RESOLVED VISUALIZATION OF HEAT FLUXES FROM NEAR-WALL TURBULENT FLOWS.

I. Znamenskaya, E. Koroteeva, V. Chirich, M.Muratov
znamen@phys.msu.ru
Lomonosov Moscow State University, 119991, Leninskie Gory, 1, Moscow, Russia

Infrared thermography with high spatio-temporal resolution offers promising new approach for truly noninvasive measurement of turbulent characteristics in the near-wall flow region.
If the material of walls bounding the flow is infrared opaque (metal walls of a shock tube channel), the recorded heat fluxes are emitted from the outer wall surfaces and the inverse heat transfer problem needs to solved to obtain the turbulent temperature fluctuations within the fluid [1]. Heat fluxes from the outer shock tube wall surfaces 1,5-2 mm thick were tested.
An infrared-transparent bounding wall, allows direct real-time measurements of heat fluxes from the fluid flow. If the studied fluid is liquid water, the recorded fluxes are emitted from a near-wall water layer, a fraction of a millimeter thick [2]. The approach is based on following the dynamics of the thermal spots - temperature inhomogeneities induced by turbulent mixing - that serve as passive flow tracers. In this work, thermal imaging of near-wall temperature fluctuations at frames rates up to 900 Hz is performed to gain new insights into the turbulent properties of impinging jet flows of various configurations. We considered a submerged water jet exiting a pipe with a circular nozzle, D = 2 mm or D = 3 mm in diameter. The pipe was placed in a large square glass reservoir tank filled with room temperature water. The nozzle was located at a distance H (from 2D to 4D) from the infrared-transparent window made of a 2-mm thick CaF2 slab, 50 mm in diameter. The window also served as an impingement plate. The jet exit Reynolds numbers, Re, based on jet exit velocities were between 5000 and 25000. The contrast on thermal images was achieved by keeping the jet temperature 5-10 °C above the ambient water temperature. Turbulent mixing of warm and cold water within submillimeter boundary layer induced thermal fluctuations that could be associated with flow dynamics. The distributions of calculated root mean square values of fluctuating near-wall apparent temperature were obtained. The analysis of thermal images allows detecting the regions of highest turbulent fluctuations and flow transition zones. Thus, the proposed measurement technique provides new insights into the turbulent properties of the near-wall flow generated by the interaction of an impinging jet with a ribbed surface and facilitates the prediction of heat transfer to the impingement wall. The results demonstrate how the presence of the rib on a smooth wall changes the distribution of turbulent fluctuations depending on flow geometry - the locations of near-wall regions with minimal flow perturbation (around the flow stagnation point) and regions with intensive turbulent mixing.

REFERENCES

1. Tiselj I., Horvat A., Mavko B., Pogrebnyak et al. Wall properties and heat transfer in near-wall turbulent flow. *Numerical Heat Transfer, Part A: Applications*, 46(7), 717–729, 2004
2. A. M. Shagiyanova, E. Yu Koroteeva, I. A. Znamenskaya, et al. Analysis of the visualization region in near-wall fluid layer by high-speed infrared thermography. *Moscow University Physics Bulletin*, 75(2), pp. 143–147, 2020.

SIMULATION OF A PACKED SUSPENSION OF MICROSWIMMERS

*Y. Kogure[1], T. Omori[1], and T. Ishikawa[1]

[1] Tohoku University, 6-6-01 Aoba, Aramaki, Aoba-ku, Sendai 980-8579 Japan
*yu.kogure.s1@dc.tohoku.ac.jp

ABSTRACT

There are various packed suspensions of microswimmers in nature. Among them, red tide caused by microscopic algae and biofilms generated by bacteria have a relationship to problems such as damage to fisheries and contaminations of medical equipment. Mass transport in microorganism suspension is one of the most important factors involved in cell growth and their habitat, so understanding the transport phenomena leads to our prediction and control of their expansion, and solution of the above-mentioned problems. Previous studies have mainly analyzed mass transport in dilute and semi-dilute suspensions of microorganisms, but rarely featured mass transport in a concentrated suspension. This study aims to develop the model of quantifying transport phenomena in a dense suspension of swimming microorganisms and to evaluate the flow field, mass transport of substances, and nutrient uptake.

Each swimming microorganism was modeled by the squirmer which swims generating slip velocities on its body surface. The squirmer changes motility by swimming modes and then represents various swimming of microorganisms. Squirmers were assumed to be arranged in the body-centered cubic lattice with a volume fraction of 50%, structurally stuck, and oriented in the same direction due to taxis. The flow around squirmers can be approximated as Stokes flow, so the velocity field around them was calculated by the boundary element method with the triply periodic boundary condition. To evaluate mass transport in a suspension, we calculated motions of particles with Brownian diffusion by Lagrange description. Then, we introduced the Peclet number, which indicates whether the mass transport is advection or diffusion dominated, as a parameter. The net flow in a suspension was set to be zero, with applying the triply periodic boundary condition.

We have successfully developed a numerical code for simulating transport phenomena in a packed suspension of squirmers. The flow structures and velocity fields generated by squirmers differed with swimming modes, so the motions of particles differed with the velocity fields. They were also dependent on the Peclet number. Thus, mass transport depended on the velocity fields with various swimming modes and the Peclet number.

REFERENCES

[1] Ishikawa, T., Locsei, J. T., and Pedley, T. J., “Fluid Particle Diffusion in a Semidilute Suspension of Model Micro-organisms”, Physical Review E, Vol. 82, (2010), 021408.

[2] Blake, J. R., “A Spherical Envelope Approach to Ciliary Propulsion”, Journal of Fluid Mechanics, Vol. 46, (1971), pp. 199-208.

MASS TRANSFER AND VELOCITY MEASUREMENTS IN T-JUNCTION PIPING SYSTEM

Kazuki Munemura[1], Kazuki Kawaguchi[1], *Takayuki Yamagata[1], and Ryo Morita[2]

[1] Niigata University, 8050 Ikarashi 2-Nocho, Nishi-ku, Niigata, Niigata 950-2181 Japan
*yamagata@eng.niigata-u.ac.jp

[2] Central research Institute of Electric Power Industry, 2-6-1 Nagasaka, Yokosuka-shi, Kanagawa 240-0196 Japan

ABSTRACT

Prediction and management of pipe-wall thinning in fossil and nuclear power plants are important for safety management and stable supply of electricity. One of the main causes of the pipe-wall thinning is flow accelerated corrosion [1], which is a mass transfer phenomenon from the pipe-wall surface into the fluid. T-junction pipe used for branching or merging pipes in the piping system is a representative element in which the pipe-wall thinning occurs frequently. Thermal striping phenomenon caused by the merging of fluids with temperature differences have mainly investigated in the T-junction pipe [2]. This study revealed that the flow pattern after merging changed depending on the flow velocity ratio of the main pipe and branch pipe. The mass transfer in the T-junction pipe is recently investigated by numerical simulation [3]. Therefore, there are few reports on the experimental investigations of the mass transfer in the T-junction. The purpose of this study was to investigate experimentary the influence of the flow rate ratio of the main pipe and branch pipe on the mass transfer and velocity field in T-junction pipe. The experimental evaluation of the mass transfer and veloicty field was carried out in a closed-curcuit water tunnel. The mass transfer coefficient was measured by the plaster dissoluition method and the velocity field was measured by particle image velocimetry with different flow rate ratio of the main pipe and branch pipe [4]. The mass transfer coefficient of the experiment showed two peaks after the main pipe and the branch pipe merged. The maximum value increased as the flow rate from the branch pipe increased. Flow separation was enhanced near the wall surface on the branch pipe side at that condition.

REFERENCES

[1] Sanchez-Caldera, L.E., Griffith, P., Rabinowicz, E., “The mechanism of corrosion-erosion in steam extraction lines of power stations”, ASME Journal of Engineering for Gas Turbines and Power, Vol. 110 (1988), pp. 180-184.

[2] Kamide, H., Igarashi, M., Kawashima, S., Kimura, N., Hayashi, K., “Study on mixing behaviour in a tee piping and numerical analyses for evaluation of thermal striping”, Nuclear Engineering and Design, Vol. 239 (2009), pp. 1192-1200.

[3] Watanabe, S., Morita, R., “Characteristic of pipe wall thinning by flow accelerated corrosion at drift region in junction piping”, Transactions of the JSME, Vol. 86 (2020), 20-00110 (16 pages).

[4] Taguchi, S., Ikarashi, Y., Yamagata, T., Fujisawa, N., Inada, F., “Mass and momentum transfer characteristics in 90° elbow under high Reynolds number”, International Communications in Heat and Mass Transfer, Vol. 90 (2018), pp. 103-110.

PHASE-SHIFTING ELLIPSOMETER BASED ON PIXELATED POLARIZATION CAMERA TO MEASURE NANO-THICK LIQUID FILM AND NANOPARTICLE DEPOSITS

*E. Shoji[1], T. Saito[1], A. Hoshino[1] and T. Biwa[1]

[1] Tohoku University, 6-6-01 Mechanical Systems Engineering, 980-8579 Japan
*eita.shoji@tohoku.ac.jp

ABSTRACT

Environmental-friendly processes utilizing inkjet technology, e.g., printed electronics, are being developed. The wetting phenomena in the processes have been widely studied because the products, i.e., the structure of the deposits that remain on the substrate, are closely related to the wetting behavior of the inkjet droplets. Especially in recent years, nanofluids containing nanoparticles have attracted attention as nano-ink, and since nanoparticles can exist in nano-thick liquid films that develop near the contact line under certain conditions, methods for measuring the liquid films and nanoparticle deposits will become increasingly important. Therefore, this study aims to develop a phase-shifting ellipsometer based on a pixelated polarization camera and measure the shape of the liquid films and nanoparticle deposits.

The principle of the developed ellipsometer is based on an existing phase-shifting ellipsometer developed by the authors[1]. The difference between them is the algorithm in the phase-shifting technique and the way to introduce the phase shift. The previous system required synchronization of the azimuthal angle of the rotating polarizer and camera, but this synchronization is no longer necessary with the use of the polarization camera in the present system. This provides better temporal resolution than previous studies[2].

In the experiments, alkanes, such as *n*-octane and *n*-decane, were used as solvents and decanoic acid-modified CeO_2 nanoparticles[3] with a diameter of approximately 6 nm were added to the alkanes. A droplet of the nanofluids was ink-jetted on a Si substrate. The amount of droplets on the substrate was controlled by the number of ink-jetted droplets with a volume of approximately 250 pL per droplet. In the experiment using *n*-octane solvent, a ring-shaped particle layer was formed at conditions with a maximum contact radius of more than 100 μm, and the inside of the ring was uniform thickness with 1–2 nanoparticles. As the droplet volume increased, the development of the nano-thick liquid film emanating from the contact line was observed, and nanoparticles deposition within the film was also observed.

REFERENCES

[1] Shoji, E., Komiya, A., Okajima, J., Kubo, M. and Tsukada, T., “Three-step phase-shifting imaging ellipsometry to measure nanofilm thickness profiles”, Optics and Lasers in Engineering, Vol. 112, (2019), pp. 145–150.

[2] Shoji, E., Kaneko, T., Yonemura, T., Kubo, M., Tsukada, T. and Komiya, A., “Measurement of dynamic wetting using phase-shifting imaging ellipsometer: comparison of pure solvent and nanoparticle suspension on film thickness profile, apparent contact angle, and precursor film length”, Experiments in Fluids, Vol. 62, (2021), pp. 1–13.

[3] Zhang, J., Ohara, S., Umetsu, M., Naka, T., Hatakeyama, Y. and Adschiri, T., “Colloidal Ceria Nanocrystals: A Tailor-Made Crystal Morphology in Supercritical Water”, Advanced Materials, Vol. 19, (2007), pp. 203–206.

VISUALIZATION OF FLOW FIELD INDUCED BY HYDRODYNAMIC INTERACTIONS OF SWIMMING MICROORGANISM

*K. Kubo[1], T. Omori[1] and T. Ishikawa[1]

[1] Tohoku University,6-6-01 Aoba Aramaki Aoba-ku , Sendai, Miyagi 980-8579 Japan
*kiyoto.kubo.q5@dc.tohoku.ac.jp

ABSTRACT

Aquatic swimming microorganisms are ubiquitous in nature and have a major impact on the global environment. For example, accumulation of planktonic microorganism in ocean can be related to toxic red tides that are detrimental to fishing industries. It has been reported that the interaction between the motility of microorganisms and the background flow caused by ocean currents and winds is important in the aggregation process of plankton [1], and it is necessary to elucidate the fluid motion around microorganisms to describe their population dynamics. In particular, there is a need to understand the nearby fluid motions that govern microbial aggregation and interactions, as well as cell-scale flows within microbial suspensions. In this study, we developed a computational model of dense suspensions of swimming microorganisms and quantified cell-cell interactions by visualizing the flow field in the suspension. Especially, we will discuss the effects of the cell deformability on the hydrodynamic interactions.

The microorganism is assumed to be a ciliate and is modeled as a capsule of a hyperelastic thin membrane enclosing a Newtonian fluid in order to take the membrane deformability into account [2]. The area incompressibility and shear resistance of membrane are described by the Skalak constitutive law [3]. The model has torque distribution slightly above the membrane for a propulsion [2]. A solid mechanics of the cell membrane is solved by a finite element method. Due to small Reynolds number of microorganism motion, the fluid flow is assumed to be governed by the Stokes equation, which is solved by the boundary element method. This model solves a fluid-structure interaction, which couples the two-way interaction between the fluid and the cell membrane. The motion of microorganisms is computed under triply periodic boundary. Ewald summation and multipole expansion techniques are used to decrease the calculation costs.

We have successfully simulated flow fields in very dense microbial suspensions with a volume fraction of 20% and visualized the flow field formed by interacting microorganisms. Different swimming modes are expressed by varying the torque distribution. Cell-cell interactions produced by the different swimming modes are discussed.

REFERENCES

[1] Durham, W.M., Kessler, J. and Stocker, R., “Disruption of vertical motility by shear triggers formation of thin phytoplankton layer”, Science, Vol. 323 (2009), pp. 1067-1070.

[2] Matsui, H., Omori, T., and Ishikawa, T., “Hydrodynamic interaction of two deformable torque swimmers”, J. Fluid Mech. 894, A9 (2020).

[3] Skalak, R., Tozeren, A., Zarda, R. P. and Chien, S., “Strain energy function of red blood cell membranes”, Biophys. J., 13 (1973), pp. 245-264.

MODELING SPERM SUSPENSION IN THREE DIMENSIONS

*N. Taketoshi[1], T. Omori[1], T. Ishikawa[1]

[1] Tohoku University, Aramaki Aza Aoba 6-6-01, Sendai 980-8579, Japan
*nanami.taketoshi.q1@dc.tohoku.ac.jp

ABSTRACT

Sperm and their motility are an important part of the fertilization process. One of the recent social problems is the declining birth rate, which is partly due to the increase in the number of couples suffering from infertility. Various factors contribute to infertility, but in recent years, male infertility has been attracting attention in addition to female infertility. One of the symptoms of male infertility is oligozoospermia, a condition in which the number of sperm is lower than normal. Declining sperm numbers are thought to lead to lower egg encounter rates, but recently it has also been reported that sperm motility can vary with their population. For example, mammalian sperm have shown that bovine and mouse sperm swim in groups and that their swimming directions are aligned. These experimental results suggest that the swimming behavior of sperm differs between single swimming and group swimming. It is possible that the low sperm counts of oligozoospermia, as described above, and sperm in normal semen show different swimming behaviors, which may affect the difference in fertilization rates. Therefore, clarifying the relationship between sperm count and swimming may help to solve infertility problems.

In the previous hydrodynamical studies on sperm swimming have so far focused on single sperm and two sperm [(1,2)], but there have been only a few simulations of sperm population swimming [(3,4)]. Although these studies showed sperm forming clusters or large structures, the motions of sperm cells have restricted in a plane. In reality, however, they swim in three-dimensional space, and thus it is necessary to investigate how sperm behave in three-dimensional space.

In this study, we tried to represent sperm swimming in more realistic by making the sperm model swam freely in 3D space, and investigated how the swimming behavior differs when the concentration of sperm is changed.

REFERENCES

[1] Nanami Taketoshi, Toshihiro Omori and Takuji Ishikawa, “Elasto-hydrodynamic interaction of two swimming sperm”, Physics of Fluids, 32, 101901, 2020.

[2] Toshihiro Omori and Takuji Ishikawa, PHYSICAL REVIEW E., 93, 032402, 2016

[3] Kenta Ishimoto and Eamonn A. Gaffney, ”Hydrodynamic Clustering of Human Sperm in Viscoelastic Fluids”, Scientific Reports, 8:15600, 2018

[4] Simon F. Schoeller and Eric E. Keaveny,” From flagellar undulations to collective motion: predicting the dynamics of sperm suspensions”, Journal of Royal Society Interface, 15:20170834, 2018

EXPERIMENTAL STUDY OF MOVING-TRPIV ON DRAG REDUCTION CONTROL OF COHERENT STRUCTURES IN TURBULENT BOUNDARY LAYER OVER SUPERHYDROPHOBIC SURFACE

Yue Jinhui[1],Wang Xinwei[1], Jiang Nan[1,2]

[1] Department of mechanics, Tianjin University, 135 Yaguan Road, Haihe Education Park, Jinnan District,, Tianjin, Tianjin 300354, China

[2] Tianjin Key Laboratory of modern Engineering Mechanics, 135 Yaguan Road, Haihe Education Park, Jinnan District,, Tianjin, Tianjin 300354, China

ABSTRACT

The moving TR-PIV system was used to measure the velocity fields of the turbulent boundary layer on smooth surface and superhydrophobic surface as they moved downstream. The data is denoised by filtering, and the fluctuating velocity signal is obtained by combining the interpolation results of the direct numerical simulation data. It is found that the Reynolds shear stress, flow and normal turbulence of the superhydrophobic surface in the near wall region of the boundary layer are significantly lower than those of the smooth surface, which indicates that the superhydrophobic surface can suppress the high Reynolds stress events in the near wall region. Using wavelet decomposition to decompose the pulsation velocity, it is found that the maximum energy of the flow direction of the two surfaces is located in the near wall region, and the maximum energy scale is about 2δ. The spatial topological morphology of the coherent structure of the eject and sweep process of the maximum energy scale was detected separately using the conditional sampling and the spatial phase averaging methods, it is found that the vortex structure of the large-scale eject and sweep events is a large-scale vortex packet structure composed of several small vortices, and the streamline of each small vortex forms a local unstable dynamic system composed of saddle points and focal points. The dynamics of sweep event on superhydrophobic surfaces are simpler and less intense. λ_{ci} criterion is used to extract the vortex strength in the two surfaces, it is found that the vortex strength of the superhydrophobic surface is obviously weaker than the smooth surface near the wall. The number of uniform momentum zone (UMZs) on the superhydrophobic surface is lower, indicating that the large scale coherent structures has an effect on the number of uniform momentum zone. In the above study, the passive control of the coherent structure was used as the starting point to explore the influence of the superhydrophobic surface on the coherent structure. The results show that the energy of the burst coherent structure in the turbulent boundary layer of the superhydrophobic surface is reduced, thus achieving the drag reduction effect.

REFERENCES

[1] Adrian RJ, Meinhart CD, Tomkins CD. Vortex organization in the outer region of the turbulent boundary layer. Journal of Fluid Mechanics, 2000, 422: 1-54

Tomographic PIV measurements in the wake of a wall-mounted barchan dune

XiaoLei Han[1], Akira RINOSHIKA[1, 2](Corresponding author)

[1] School of Aeronautic Science and Engineering
Beijing University of Aeronautics and Astronautics
37 Xueyuan, Haidian District, Beijing, 100191, CHINA
[2] Department of Mechanical Systems Engineering
Graduate School of Science and Engineering, Yamagata University
4-3-16 Jonan, Yonezawa-shi, Yamagata 992-8510, JAPAN
e-mail: rinosika@yz.yamagata-u.ac.jp

Abstract

Barchan dunes with typical crescentic planform shape, commonly existing in nature, have been widely explored due to the great potential to reduce drag. However, the dynamic characteristics of this wake field is strong three-dimensional, time-dependent, turbulent. As such, a volumetric velocity measurement technique is required to capture the topological evolution of these coherent vortex structures. For this purpose, tomographic particle image velocimetry (TPIV) measurement is conducted to investigate flow structures behind a wall-mounted barchan dune model mostly immersed in a laminar boundary layer in a circulation water tunnel. The instantaneous three-dimensional arch-shaped vortex chains originating from the rear ridge of the barchan and the typical quasi-streamwise streak induced by the internal boundary layer development are first experimentally obtained in this work. Additionally, the time evolution of these dominant flow structures was investigated by use of time-dependent conditional averaging, which is a necessary step in topological analysis within the complex flow. Nevertheless, both conditionally averaged and typical instantaneous three-dimensional flow fields all provide credible proof for the fact that arch-shaped vortex populates the barchan dune wake, and the inclination angle of these structures aligned with the streamwise direction will decrease when advecting downstream. These results provide a comprehensive picture for us to understand the law of the temporal behavior of three-dimensional unsteady coherent flow structures within the wake fields of the barchan dune.

Key words: dynamic characteristics, arch-shaped vortex, vortex evolution

REVERSAL MOTION OF THE SOLID PARTICLES WITH HIGH THERMAL CONDUCTIVITY IN THE SOLID-DISPERSED RAYLEIGH-BÉNARD CONVECTION

Dai SHI*[1]

[1] Department of Mechanical Engineering, Osaka University, 2-1 Yamada-oka, Suita-City, Osaka 565-0871 Japan
*shi@fluid.mech.eng.osaka-u.ac.jp

ABSTRACT

The liquid-solid multiphase Rayleigh–Bénard (RB) system is one of the most typical non-equilibrium hydrodynamic systems in nature and industry, while most studies on granular flow in the multiphase RB system simplify the influence of thermal coupling from the solid phase to the fluid, viewing them as isothermal particles. The temperature gradient within the solid phase is studied by Takeuchi et al [1], and a regular oscillation mode of the granular flow is predicted to appear when the heat conductivity ratio between the solid and liquid phase increases to more than 10^1. A reversal mode is also predicted to exist when the convection in the same RB system becomes stronger [2].

In this study, an original experimental method to replace the vertical gravitational acceleration with a horizontal centrifugal acceleration is developed to reproduce the multiphase RB system in reality. The heat conductivity ratio between the solid and liquid phase is set to be 83.7 to highlight the influence of thermal coupling from the particles to the fluid. Experiments under the condition of Rayleigh number (Ra) = 2.7×10^4, 5.6×10^4, 1.1×10^5 in a square, closed RB system is carried out to observe the change in particle motion.

It is found that most particles remain stationary because of the static friction force in the system, even when Ra increases from 2.7×10^4 to 1.1×10^5 by varying the temperature difference between the hot and cold end of the RB system from 5K to 30K. However, some floating particles are found to show an obvious, coherent flow pattern of “reversal” under the Ra condition of 2.7×10^4. This large-scale “reversal” pattern survives even when the Ra increases to 1.1×10^5.

A physical explanation for this reversal phenomenon is attempted based on the particle mechanical and thermal coupling with the fluid. The imbalance in buoyancy momentum is believed to be the key to explain for the occurrence of the reversal phenomenon.

REFERENCES

[1] Takeuchi S. et al, “Effect of temperature gradient within a solid particle on the rotation and oscillation modes in solid-dispersed two-phase flows”, International Journal of Heat and Fluid Flow, 43 (2013) 15-25.

[2] Takeuchi S. et al, “Flow reversals in particle-dispersed natural convection in a two-dimensional enclosed square domain”, Physical Review Fluids, 084303 (2019) 1-22.

NEAR-LIMITING BEHAVIOR OF POOL FIRES WITH STRONG SWIRLING FLOW

*K. Tsukamoto[1], A. Odawara[1], T. Yamazaki[1], T. Matsuoka[1] and **Y. Nakamura[1]

[1] Toyohashi University of Technology, 1-1 Hibarigaoka, Tempaku, Toyohashi 441-8580 Japan

*tsukamoto.keita.kj@tut.jp; **yuji@me.tut.ac.jp

ABSTRACT

Near-limit dynamic behavior of non-premixed flame formed over the ethanol pool fires with prescribed swirling flow using rotational mesh screen has been examined experimentally. Experimental parameters were the diameter of the pool and the rotational speed of the mesh screen. Rotational mesh is hanged over the ceiling with rotor and its deformation during the rotation is prevented using the circular guide. In this way, the rotation axis is quite solid even we employed the high-speed of rotation; this enables us to investigate the near-limiting flame behavior at high circulation condition. Flame dynamics was recorded by the digital video camera and binary image of the flame is analyzed to capture the flame behavior character depending on the swirling flow.

It was experimentally confirmed that the flame height varies 1/3 power of the imposed circulation as suggested by the past works for various size of fire whirls [1-3]. However, it is clearly identified the trend does not sustain when higher circulation is imposed, especially with the smaller pools. Such critical conditions strongly depend on both pool diameter and the circulation. With careful observation and analysis, repetitive motion in the flame height in time (namely, large fluctuation of the flame dynamics) has been identified. With small-pool flame, the flame diameter is originally small. Once the circulation flow field is imposed, the flame responds to be narrower to form smaller flame diameter along the axis. Because the chemical reaction is finite process in time (even it is fast enough in some sense), too-small flame diameter is not possible to present, namely, local extinction shall occur. Once this happens (with strong circulation field), flame is suddenly "terminated" and the flame height is abruptly shorter. However, the unburned gas will remain in downstream due to the local extinction as described, re-ignition may occur once the condition is relaxed. In this way, the flame height may appear suddenly larger. Such local extinction and re-ignition may result in appearance of repetitive behavior of the flame dynamics as observed. At present, we are attempted to summarize the critical condition to appear the fluctuation by means of local extinction theory.

REFERENCES

[1] Kuwana, K., Sekimoto, K., Saito, K., and Williams, F.A., "Scaling Fire Whirls", Fire Safety Journal Vol. 43 (2008), pp. 252-257.

[2] Lei, J., Liu, N., and Zhang, L., "Experimental Research on Combustion Dynamics of Medium-scale Fire Whirl", Proceedings of The Combustion Institute Vol. 33 (2011), pp. 2407~2415.

[3] Hartl, K.A., and Smits, A.J., "Scaling of A Small-scale Burner Fire Whirl", Combustion and Flame Vol. 163 (2016), pp.202-208.

UNSTEADY FLOW INSTABILITY BETWEEN TWO PARALLEL CIRCULAR PLATES IN POWER GENERATOR

*Naho Kishizawa[1], Itsuro Honda[2], Naohisa Takagaki[2] and Osamu Kawanami[2]

[1] Graduate student, University of Hyogo, 2167, Shosha, Himeji city, Hyogo 671-2201 Japan
*ep18b034@steng.u-hyogo.ac.jp

[2]School of Eng., University of Hyogo, 2167, Shosha, Himeji city, Hyogo 671-2201 Japan

ABSTRACT

The electric generator has a weak point that electric energy is converted into heat energy. Cooling parts is important for safe operation, as high temperatures in parts can destroy the machine. A gap between two parallel circular plates regarded the cooling part of the electric generator, unstable flow velocity occurs. The reason this phenomenon is caused is unexplained. The purpose of this study is that the result of PIV experiments and that of numerical calculations is compared. In this paper, radial flow and tangential flow in the gap between two parallel circular plates was performed by PIV experiment and numerical simulation. In the experiment, two parallel circular plates having a diameter of 200 mm were used. The two-dimensional velocity field measured in the center plane of the wall-normal-span. In the numerical computations, high order finite difference scheme is applied to the Navier-Stokes equations. The numerical simulation was conducted over the wide Re numbers. Computational results showed the unsteady velocity field as the Re number rise to 10000. The computational results were animated to observe the unsteady state. From this animation, the frequency of the unsteady phenomenon between two parallel circular plates was obtained. The Re=10000, in which an instability region appeared in this study, is almost the same as Orszag's critical Reynolds number Rec=5772. These results show that the unsteady flow is rotating more slowly than the inlet rotation based on the inlet tangential velocity. The important findings are that unsteady flow obtained here is rotating stall similar to the vaneless diffuser of the centrifugal fan. As Re number increases, the shear layer was formed between inner boundary of the plate and the stationary its outer fluid. Therefore, rotating stall observed here is interpreted by the Kelvin-Helmholtz instability at shear layer.

REFERENCES

[1] Orszag, S. A., J. Fluid Mech, Vol. 50 (1971), pp.689-703

[2] G. K. Batchelor, An Introduction to Fluid Dynamics, Cambridge University Press, (1967), Chap. 7.

3D WAVELET TRANSFORM ON FLOW STRUCTURES BEHIND A WALL-MOUNTED SHORT CYLINDER WITH A FRONT INCLINED HOLE

H. Rinoshika

Yamagata University, 4-3-16 Jonan, Yonezawa-shi, Yamagata 992-8510, Japan
hrinoshika@gmail.com

ABSTRACT

This paper deals with the three-dimensional (3D) wake structures of a wall-mounted short circular cylinder, which are controlled by the flow issuing from a front inclined hole (FIH) drilled from the front side surface to the free end surface in the short cylinder. The 3D velocity fields are instantaneously measured at a Reynolds number of 10720 based on high-resolution tomographic particle image velocimetry (Tomo-PIV) in a water tunnel. The 3D velocity fields, Q criterion, tip vortices, and arch-type vortex are compared between the FIH cylinders and a standard cylinder. A 3D W-type arch vortex is observed behind the short FIH cylinder. Compared with the standard cylinder, the 3D W-type arch vortex exhibits a large peak or convex at center of the horizontal part and collapses more rapidly at the downstream for the FIH cylinders. The flow from the FIH may increase the rear recirculation region, and its height may effectively control the vortical structures. A 3D wavelet multiresolution analysis is employed to decompose the instantaneous 3D velocity fields measured by the Tomo-PIV. The large-scale streamwise vortices in the FIH cylinder wakes are smaller than those in the standard cylinder wake, and this becomes more evident as the hole height increases. However, 3D time-averaged large-scale arch vortical structures becomes stronger.

Keywords : 3D wavelet transform, Arch vortex, Flow control, Tomographic PIV, Vortex, Wake, Wall-mounted short cylinder

VISUALIZATION OF THERMAL NETWORKS IN LAYERED STRUCTURES BY MEANS OF PUMP-PROBE AND DECONVOLUTION TECHNIQUE

*T. Silveira Joao Vitor[1], H. Daiki[1] and F. Kazuyoshi[1]

[1] Tokyo Institute of Technology, Meguro-ku, Oookayama, 2-12-1, i6-3, 152-8850 Japan
*thomsen.j.aa@m.titech.ac.jp

ABSTRACT

In thermal management of electronics, the thermal parameters of components, such as the junction-to-case or junction-to-ambient thermal resistances are necessary for adequate thermal characterization of semiconductor packages. To estimate such parameters, the network identification by deconvolution method (i.e., NID) is widely applied in the semiconductor industry. Commonly, the NID method uses the thermal response obtained from inside the package (e.g., heat generation in the semiconductor die) to characterize the test subjects. In this context, the present work investigates the practical aspects of applying NID to the thermal characterization of electronics from an external excitation source.

The NID method consists of obtaining the time constant spectrum (i.e., TCS) of a given system by deconvolving its thermal response from a weight function. Further transformations utilizing the TCS allow for the construction of the structure function relative to the device under test, which maps the heat flow path from the source to the heat sink. This technique enables the identification of the thermal resistances in the system.

To achieve NID from an outside excitation, the temperature transients used as input for the NID are measured with the pump-probe transient thermoreflectance method (i.e., PPTTR). This methodology consists of using two lasers. The pump laser is applied to excite the sample, while the probe laser provides a signal to be measured by a photodetector. Thus, the reflectance transients obtained from the photodetector provide the thermal response used in the NID.

By employing heat transfer simulations, this investigation goes into detail about the theorical aspects of using NID from an external excitation. Experimentally, the thermal characterization of a thin film on the order of 100 nm on a substrate was successfully carried, which highlights the potential of the methodology to determine the thermal resistances of layered structures, such as semiconductor dies, from external excitation.

REFERENCES

[1] Székely, V. and Van. B. T., “Fine structure of heat flow path in semiconductor devices: A measurement and identification method”, Solid-State Electronics, Vol. 31, No. 9-(1988), pp. 1363-1368.

[2] Y. Ezzahri and A. Shakouri, "Application of network identification by deconvolution method to the thermal analysis of the pump-probe transient thermoreflectance signal", Review of Scientific Instruments 80, 074903, (2009), https://doi.org/10.1063/1.3176463

VISUALIZATION OF MIXING AND FLOW STRUCTURE IN BLADE-FREE PLANETARY MIXER WITH FREE SURFACE

*T. Yamagata[1], K. Sato[1], and T. Igarashi[1]

[1] Niigata University, 8050 Ikarashi 2-Nocho, Nishi-ku, Niigata, Niigata 950-2181 Japan
*yamagata@eng.niigata-u.ac.jp

ABSTRACT

Blade-free planetary mixer can agitate a liquid without a mixing blade by a flow induced by a precession of the container. The precessing flow is realized by the combined effect of the rotation of the container around an axis and the revolution around an off-axis distance away from the container. Clarifying the flow structure and mixing mechanism inside the container is necessary for improving mixing performance. In the precessing cylinder filled with fluid, flow instability increases [1] and the vortical structures are found to be in spiral forms [2]. The inclination and distortion of the rotation axis increases at a certain precession rate, at which the mixing performance increases [3][4]. These studies focused on flow structures in a liquid-filled cylinder. Therefore, the information on the flow structure with the free surface is insufficient, which is found in actual mixer use. The objective of this study was to investigate the flow structure and mixing performance in a precessing cylinder with a free surface of the fluid. The flow structure and mixing performance were numerically investigated with an unsteady-state solver for incompressible flows of Newtonian fluids and the volume-of-fluid method. The numerical results were validated by comparison with flow visualization experiments of mixing from particle dispersion. The precessing cylinder with a free surface increased mixing performance at a higher precession rate than liquid filled case. The deformation of the axis of the swirling flow due to the inclination of the free surface resulted a change in the precession rate that increased the mixing performance. The angle of the cylinder strongly affected the mixing performance, while the liquid level height in the cylinder and revolution radius had a minor effect.

REFERENCES

[1] Lagrange, R., Meunier, P., Nadal, F., Eloy, C., “Precessional instability of a fluid cylinder”, Journal of Fluid Mechanics, Vol. 666 (2011), pp. 104-145.

[2] Chergui, N., Lateb, M., Lacroix, E., Thermo, L. D., “CFD study of flow dynamics in a blade free planetary mixer (BFPM) – A qualitative flow study”, Chemical Engineering Research and Design, Vol. 102 (2015), pp. 100-115.

[3] Yamagata, T., Fujisawa, N., “Effect of Rotation and Revolution on Performance of Blade-Free Planetary Mixer”, Journal of Flow Control, Measurement & Visualization, Vol. 7 (2019), pp. 1-10.

[4] Yamagata, T., Sugisawa, H., Fujisawa, N., “Experimental study on laminar mixing in planetary mixer”, Experiments in Fluids, Vol. 62 (2021), 28 (12 pages).

EXPERIMENTAL INVESTIGATION ON FLOW CHARACTERISTICS AND HEAT TRANSFER OF NONCIRCULAR SYNTHETIC JETS

L. Wang[1], *L.H. Feng[1], Y. Xu[1] and J.J. Wang[1]

[1]Fluid Mechanics Key Laboratory of Education Ministry, Beijing University of Aeronautics and Astronautics, Beijing 100191 China
*lhfeng@buaa.edu.cn

ABSTRACT

This study reveals the flow characteristics and heat transfer of noncircular synthetic jets through studying their interaction with a cross-flow [1] and a heated wall [2]. For the interaction with a cross-flow, tomographic particle image velocimetry (PIV) is used to measure the flow field of a rectangular synthetic jet with aspect ratio (AR) of 3 interaction with a laminar boundary-layer. Two orifice orientations (spanwise and streamwise) are examined at a synthetic-jet to free-stream velocity ratio of 1. The results display complex three-dimensional vortical structures and axis switching of vortex rings. Due to axis switching, the spanwise configuration yields vortical structure interaction, causing faster jet transition and lower velocity deficit than the streamwise case. In addition, a stronger spanwise vortex in the near-wall region is induced by the spanwise configuration, which transfers more momentum and initial disturbance into the boundary layer, suggesting greater efficiency of promoting boundary-layer transition. For the impingement onto a heated wall, thermographic phosphor thermometry and two-dimensional PIV are used to acquire the wall temperature and flow field, respectively. Five orifice configurations (circular, square, AR = 3 elliptic, and AR = 3 and 5 rectangular) are examined at an orifice-to-wall distance of 7.5. The results demonstrate heat transfer enhancement of impinging synthetic jets using noncircular orifices. The square orifice displays better stagnation cooling performance due to the larger impinging velocity and deeper penetration into the wall shear layer. The elliptic and rectangular orifices display axis-switching of vortex rings before they impinge onto the wall, which enhances the near-wall mixing and turbulent kinetic energy and thus heat transfer compared with the circular case. In particular, strong secondary structures in the trailing jet are induced by the AR = 5 rectangular orifice, resulting in the double-peak distribution of wall temperature in the major-axis direction. Furthermore, the AR = 5 rectangular orifice yields a stronger wall jet in the minor-axis direction. As a result, the AR = 5 rectangular orifice achieves the optimal cooling performance with the maximum time-averaged cooling area.

REFERENCES

[1] Wang, L., Feng, L.H., “The interactions of rectangular synthetic jets with a laminar cross-flow”, Journal of Fluid Mechanics, Vol. 899, (2020), A32.

[2] Wang, L., Feng, L.H., Xu, Y., Xu, Y., Wang, J.J., “Experimental investigation on flow characteristics and unsteady heat transfer of noncircular impinging synthetic jets”, International Journal of Heat and Mass Transfer, Vol. 190, (2022), 122760.

VISUALIZATION OF A FRESHWATER-SPONGE SNEEZING

*Kei Kawashima[1], Kenji Kikuchi[1], Takuji Ishikawa[1]

[1]Tohoku University, 6-6-01 Aoba, Aramaki, Aoba-ku, Sendai 980-8579 Japan
*kei.kawashima.s8@dc.tohoku.ac.jp

ABSTRACT

Sponges, which are sessile animals and widely distributed under water, take a filter feeding using flagellar beat in the choanocytes. Choanocytes are the cells in the body of a sponge that undertake both pumping and filtering functions. Filter feeding of sponges accumulates the organic particles, which is one of the sources of water pollution[1]. Hence, sponges play a role in purifying water in nature[2]. Interestingly, sponges can clear clogged incurrent canal by a sneezing-like behavior in spite of having no muscles or nerves[3]. In order to apply the sponge's filtration function to engineering filter systems, we investigated sneeze-like discharge behaviors that support the constant water purification action of sponges.

To induce sneezing, we compared the sneezing frequency of a sponge in different amounts of dissolved oxygen in the culture medium, and found that the frequency of sneezing was predominantly greater for a sponge in the medium with less dissolved oxygen.

Previous studies have induced sneezing by applying mechanical agitation to a sponge or by adding amino acids such as GABA and L-glutamate[4]. In the current study, we newly found that the sponge increases the frequency of sneezing by decreasing the concentration of dissolved oxygen. This suggests that a sponge may perceive a decrease in oxygen concentration as an incurrent canal blockage.

We fed the cultured sponge with fluorescent particles and took time-lapse photographs, which showed that the sponge sneezed, but the number of particles in their bodies did not change after the sneeze. However, we succeeded in observing particles transported from the choanocytes to the amoebocytes during the sneezing.

This study suggests that the sneezing behavior of a sponge may not only backflow water to clear the incurrent canal, but may also help transport small particles from choanocytes to amoebocytes.

In the future, we intend to measure discharging effects of sneezing and develop a mathematical model of the constant water purification action of a sponge that takes into account the sneezing and other particle emission behaviors.

REFERENCES

[1] Steven E. McMurray, Zackary I. Johnson, Dana E. Hunt, Joseph R. Pawlik, Christopher M. Finelli, Selective feeding by the giant barrel sponge enhances foraging efficiency, *Limnology and Oceanography*, Vol. 61, (2016), pp.1271-1286

[2] Nathan Grant, Eugueni Matveev, Amanda S. Kahn, Sally P. Leys, Suspended sediment causes feeding current arrests in situ in the glass sponge *Aphrocallistes vastus*, *Marine Environmental Research*, Vol. 137,(2018), pp. 111-120

[3] Glen R. D. Elliot, Sally P. Leys, Evidence for glutamate, GABA and NO in coordinating behavior in the sponge, *Ephydatia muelleri* (Demospongiae, Spongillidae), *The Journal of Experimental Biology*, Vol. 213, (2010), pp. 2310-2321

[4] Kornelia Ellwanger, Andre Eich, Michael Nickel, GABA and glutamate specifically induce contractions in the sponge *Tethya wilhelma*, *Journal of Comparative Physiology*, Vol. 193,(2006), pp. 1-11

INFLUENCE OF SYNTHETIC JET ON MULTISCALE CHARACTERISTICS IN WALL-BOUNDED TURBULENCE

*Biaohui Li[1] and Nan Jiang[2]

[1] School of Mechanical Engineering, Tianjin University,135 Yaguan Road, Jinnan District, Tianjin 300354, China
*libiaohui915@outlook.com
[2]Tianjin Key Laboratory of Modern Engineering Mechanics, 135 Yaguan Road, Jinnan District, Tianjin 300354, China

ABSTRACT

The research focuses on the impacts of synthetic jet on a fully-developed turbulent boundary layer (TBL), particle image velocimetry (PIV) system is utilized to capture the two components velocity signals in the flow field. The near-wall mean velocity profile obtained by single-pixel ensemble correlation (SPEC) [1] algorithm evidence the decline of streamwise velocity gradient in the range of about 30~60mm downstream of the synthetic jet outlet, achieving an average drag reduction rate of 19.05%. In addition, under the upwash action of synthetic jet, streamwise vortices is lifted, the activity of the near wall turbulence fluctuation is reduced.

Proper orthogonal decomposition (POD) [2] analysis reveals that the injection of the perturbation notably attenuates the induction effect of prograde vortex on the low-speed fluid in the large-scale fluctuation velocity field, thereby weakening the bursting process of near-wall turbulent events related to the skin-friction drag. The power spectral densities of the time coefficients for different mode indicate that the synthetic jet redistributes the turbulent kinetic energy, and the energy is concentrated to the small-scale coherent structure.

The time coefficients are decomposed into signals of different frequency bands by the method of complementary ensemble empirical mode decomposition (CEEMD) [3], it is found that the signal is transferred from low-frequency to-high frequency due to disturbance, thus revealing the relationship between scale and frequency band. Then, a new method of scale decomposition is proposed, that is, the high-frequency noise data is eliminated by the continuous mean square error (CMSE) [4] criterion, and the large-scale fluctuation velocity field can be obtained.

REFERENCES

[1] Sugisaki, H., Lee, C., Ozawa, Y., Nakai, K., Saito, Y., Nonomura, T., Asai, K. and Matsuda, Y., “Single-pixel correlation applied to background-oriented schlieren measurement”, Experiments in Fluids, Vol. 63, No. 1 (2022), pp. 1-7.

[2] Taira, K., Hemati, M. S., Brunton, S. L., Sun, Y. and Yeh, C. A., “Modal Analysis of Fluid Flows: Applications and Outlook”, AIAA Journal, Vol. 58, No. 11 (2019), pp. 1-25.

[3] Yeh, J. R., Shieh, J. S. and Huang, N. E., “Complementary Ensemble Empirical Mode Decomposition: A Novel Noise Enhanced Data Analysis Method”, Advances in Adaptive Data Analysis, Vol. 2, No. 2 (2010), pp. 135-156.

[4] Wu, H., Fang, Q., Peng, Y., Gong, Z. M. and Kong, X. Z., “Hard projectile perforation on the monolithic and segmented RC panels with a rear steel liner”, International Journal of Impact Engineering, Vol. 76 (2015), pp. 232-250.

STUDY OF UNDEREXPANDED SUPERSONIC JETS FROM AXISYMMETRIC MICRO-NOZZLES IN LOW REYNOLDS NUMBER REGIONS

*T. Tashiro[1], S. Nakao[1], Y. Miyazato[1] and Y. Ishino[2]

[1] The University of Kitakyushu, 1-1 Hibikino, Wakamatsu-ku, Kitakyushu, 808-0135 Japan
* c1mba013@eng.kitakyu-u.ac.jp
[2] Nagoya Institute of Technology, Gokiso-cho, Showa-ku, Nagoya 466-8555, Japan

ABSTRACT

There is a significant interest in studying shock-containing microjets due to the numerous applications such as the cooling of micro-electro mechanical systems, fine particle removal in inkjet printer heads, and so on. It has been challenging issues to investigate experimentally the shock structure in the near-field region of supersonic microjets. However, recently, Sugawara et al. [1] captured experimentally the fine structure of a shock-containing microjet issued from an axisymmetric convergent round nozzle with an inner diameter of 1000 μm at the exit by the Mach-Zehnder interferometer. Tashiro et al. [2] investigated experimentally the density field of a Mach 1.5 supersonic microjet by the Twyman-Green interferometer. In the present study, flow structure of supersonic microjet at low Reynolds numbers are investigated by the Reynolds-averaged Navier-Stokes (RANS) simulations. Typical two turbulence models including the Menter k-ω shear-stress transport (SST) and the Spalart Allmaras (SA) are used for the simulations because they have been widely used and trusted models in areas of aerospace community. Flow fields of jets issued from an axisymmetric convergent-divergent micro-nozzle with a Mach number of 1.5 and an inner diameter of 1000 μm at the exit are simulated by solving the RANS equations. In simulations, the nozzle pressure ratio (NPR), which is defined as the ratio of the stagnation pressure, p_{os}, just upstream of the nozzle to the back pressure, p_b, is held constant at NPR = 4.0, but both the p_{os} and p_b are changed simultaneously to vary the Reynolds number estimated using the diameter and flow properties at the nozzle exit. Simulations are validated by a quantitative comparison with the jet density field obtained previously using the Twyman-Green interferometer [2]. Effects of the Reynolds numbers on the microjet structure are demonstrated with the density fields of the jet and Mach number contours on the cross-sectional area including the jet centreline. The density fields simulated using the SST and SA turbulence models are in good quantitative agreement with each other and they show a reasonable agreement with the experimental density field. Effects of the Reynolds number on the near-field shock system of the microjets are clarified numerically.

REFERENCES

[1] Sugawara, S., Nakao, S., Miyazato, Y., Ishino, Y. and Miki, K., “Three-Dimensional Reconstruction of a Micro Jet with a Mach Disk by Mach-Zehnder Interferometers”, J. Fluid Mechanics, Vol. 893, (2020), A25.

[2] Tashiro, T., Nakao, S., Miyazato, Y. and Ishino, Y., “Preliminary Experiments of Supersonic Microjets by TGI, Proceedings of 33rd International Symposium on Space Technology and Science, (2022).

NUMERICAL SIMULATION OF MASS TRANSFER AND VELOCITY IN T-JUNCTION PIPING SYSTEM

Shun Watanabe[1], Ryo Morita[1], *Masaaki Satake[1] and Takayuki Yamagata[2]

[1] Central Research Institute of Electric Power Industry, 2-6-1 Nagasaka, Kanagawa, 240-0196 Japan

*m-satake@criepi.denken.or.jp

[2]Niigata University, 8050 Ikarashi 2-Nocho, Nishi-ku, Niigata, Niigata 950-2181 Japan

ABSTRACT

Prediction and management of pipe-wall thinning in fossil and nuclear power plants are important for safety management and stable supply of electricity. One of the main causes of the pipe-wall thinning is flow accelerated corrosion (FAC) [1], which is a mass transfer phenomenon from the pipe-wall surface into the fluid. Its management has been conducted with conservative evaluation of the thinning rate and the residual lifetime of the piping based on the wall thickness measurements. However, noticeable case of the wall thinning was occurred in branch and junction piping (T-junction pipe). It is a problem to manage the section beneath the reinforcing plate of the T-junction pipe; the region where measurement is difficult to conduct with ordinary ultrasonic testing device. Numerical analysis for the T-junction pipe was conducted, and the wall thinning profile due to the FAC was evaluated by calculating the mass transfer coefficient[2]. In this study, numerical simulation with the same methods[2] is conducted to investigate the influence of the flow rate ratio of the main pipe and branch pipe on the mass transfer and velocity field in T-junction pipe. The mass transfer coefficient of the experiment showed two peaks after the main pipe and the branch pipe merged. The maximum value increased as the flow rate from the branch pipe increased. These results are in qualitative agreement with the experimental results. From these results, the validity of the numerical simulation methods[2] is confirmed.

REFERENCES

[1] Sanchez-Caldera, L.E., Griffith, P., Rabinowicz, E., “The mechanism of corrosion-erosion in steam extraction lines of power stations”, ASME Journal of Engineering for Gas Turbines and Power, Vol. 110 (1988), pp. 180-184.

[2] Watanabe, S., Morita, R., “Characteristic of pipe wall thinning by flow accelerated corrosion at drift region in junction piping”, Transactions of the JSME, Vol. 86 (2020), 20-00110 (16 pages).

INERT GAS FLOW RELEASED FROM BURSTING SAUSAGE-SHAPED RUBBER BALLOON AND ITS EXTINGUISHING CHARACTARISTICS

*A. Hinako Mikami and B. Hiroyuki Torikai

Hirosaki University, Bunkyo Town 1, Hirosaki, Aomori 036-8560 Japan
*A.h21ms540@hirosaki-u.ac.jp and B.torikai@hirosaki-u.ac.jp

ABSTRACT

An inert gas is one of clean and environmentally friendly gaseous extinguishing agents. Basically, the inert gas is supplied to fire area as a high velocity jet and easy to mix with air. As a result, it is difficult to deliver the inert gases over long distances without its concentration reduction. The authors have proposed to use a rubber balloon for transporting of gaseous extinguishing agents [1,2]. When an inert gas is filled into a rubber balloon, the rubber membrane does not allow contact the inert gas with an ambient gas, and we can easily carry and deliver the extinguishing capsule to combustion region while maintaining the high concentration. Moreover, when a rubber balloon contacts with a diffusion flame, it bursts instantly, and then the inert gas filled in it is released to the flame at high speed and close range. As a result, fires are considered to be extinguished without the need for large amounts of the inert gas by using the rubber-balloon extinguishing method.

In this study, in order to clarify the extinguishing characteristics, the authors have carried out the extinguishing experiments of a methane-air diffusion flame formed on a round porous burner using a sausage-shaped rubber balloon filled with nitrogen gas. The sausage-shaped rubber balloon was transported at an arbitrary speed to the extinguishing target using an electric linear slide actuator. To measure the extinguishing effectiveness, an extinguishing probability was measured varying the transporting speed of the rubber balloon and the height of the rubber balloon from the burner surface. To elucidate the extinguishing mechanisms, the released flow from the bursting balloon was visualized with a laser light sheet method. Furthermore, the comparisons of the extinguishing effectiveness and the released flow behaviours between the sausage-shaped and elliptical-shaped rubber balloons were done.

From the experimental results, it is found that the sausage-shaped rubber balloon always has the higher extinguishing effectiveness than the elliptical-shaped rubber balloon because the nitrogen gas flow released from the sausage-shaped rubber balloon is more effectively supplied to the flame base of the diffusion flame. This means that the sausage shape is thought to be suitable as a fire-extinguishing capsule.

REFERENCES

[1] Hiroyuki Torikai, Extinguishing Characteristics of a Pool Fire with a Rubber Balloon Filled with Inert Gases. Fire Technology, Vol. 56, pp.385-399 (2019).

[2] Yasushi Iwatani and Hiroyuki Torikai, Improvement of Fire Extinguishing Performance by Decentralized Supply of Fire-Fighting Agents, Open Journal of Safety Science and Technology.Vol.7, pp.87-95 (2017).

THE FIBER-ORIENTATION OF CELLULOSE NANOFIBER SUSPENSIONS IN A PLANAR EXPANSION FLOW

*A. Kusano[1], A. Ushida[2], and T. Sato[3]

[1] Graduate school of Science and Technology, Niigata University, 8050, 2-no-cho, Ikarashi, Nishi-Ward, Niigata City, 950-2181, Japan
*f21b080h@mail.cc.niigata-u.ac.jp
[2]Faculty of Engineering, Niigata University, 8050, 2-no-cho, Ikarashi, Nishi-Ward, Niigata City, 950-2181, Japan
[3] Institute for Research Promotion, Niigata University, 8050, 2-no-cho, Ikarashi, Nishi-Ward, Niigata City, 950-2181, Japan

ABSTRACT

TEMPO-oxidized cellulose nanofiber (CNF) [1] is one of the materials expected be applied in various fields. In this study, we investigate the fiber-orientation of CNF suspensions to apply it as reinforced resin composite materials by injection molding. In recent years, items molded from plastic are mass produced and increasing [2], then they are discarded. CNF is a fiber which has biodegradable; therefore, it can be an environmentally friendly raw material for composites. To practicalize CNF as a composite, it is crucial to understand how flow influences both the direction and degree of orientation of fibers. Because the fiber orientation induced by flow in the molding process affects the properties of molded items. In fiber suspension, fibers occur mechanical interference such as entanglement as increasing. In molding flow, we have known that glass or carbon fiber is much larger than polymer of matrix so that flow field, other fibers, and mold cause mechanical interference with each fiber [3]. Flows with abrupt expansions occur often during mold processing. However, only a few studies have been conducted on the flow of CNF suspensions through a planar expansion flow. Thus, the fiber-orientation of CNF suspensions in these complex flow fields is yet to be elucidated. The aim of this study is to analyze the flow-induced orientation of CNF suspensions in a planar channel with an abrupt expansion. To achieve this, flow-induced birefringence was used. Since birefringence is correlated with the strength of the fiber-orientation, high birefringence indicates a highly oriented fiber state. As the result, the birefringence is rapidly decreased at the abrupt expansion but after that it drastic increasing. Thereafter, the birefringence gradually decreased and indicated an approximately constant value. These phenomena qualitatively did not depend on the flow rates qualitatively. The results of this study suggested that the strength of fiber orientation depends on the flow rates but is not affected by the flow rate with respect to the orientation fields.

REFERENCES

[1] Saito, T., Kimura, S., Nishiyama, Y., Isogai, A., “Cellulose Nanofibers Prepared by TEMPO-Mediated Oxidation of Native Cellulose”, Biomacromolecules, Vol. 8, (2007), pp. 2485-2491.

[2] Geyer, R., Jambeck, J.R., Law, K.L., “Production, use, and fate of all plastics ever made”, Science Advances, Vol. 3, No 7 (2007), e1700782.

[3] Kiziltas, A., Nazari, B., Gardner, D.J., Bousfield, D.W., “Polyamide 6–Cellulose Composites: Effect of Cellulose Composition on Melt Rheology and Crystallization Behavior”, Polymer Engineering & Science, Vol. 54, No. 4 (2014), pp. 739-746.

INFLUENCE OF THE PERIODIC FLOW ON THE FLOW STRUCTURE AROUND THE ROBOTIC FISH.

*M. Kawano[1], Y. Tanaka[2] and Y. Isoda[1]

[1] Graduate School of Science and Technology, Kyoto Institute of Technology,
Matsugasaki, Sakyo-ku, Kyoto, 606-8585, Japan
[2] Faculty of Mechanical Engineering, Kyoto Institute of Technology,
Matsugasaki, Sakyo-ku, Kyoto, 606-8585, Japan
*m1623106@edu.kit.ac.jp

ABSTRACT

In this study, we investigate the influence of the difference between periodic and steady flow on Q value and the surface pressure of the robotic fish. We conducted calculations of the flow around the robotic with open-source computational fluid dynamics (CFD) software OpenFOAM. In this study, the Reynolds number is about 15600, and the mean velocity is 0.04 [m/sec]. A periodic flow is a flow in which the velocity varies sinusoidally in the freestream direction. The robotic fish is composed of three sections: front as body, middle as actuated tail, and rear as caudal fin. The front section is fixed, and the middle and rear sections are oscillating. Each oscillating section is controlled its angle sinusoidally. In steady flow, the time-average lift coefficient is almost zero. In contrast, it is non-zero in a periodic flow.
We visualize the Q value to investigate the vortex structure to reveal the difference from the flow field. Moreover, we investigate the surface pressure of the robotic fish to reveal the influence of the pressure field. The pressure gradient of a periodic flow causes changing the flow velocity in the flow direction. It is difficult to understand the pressure change on the surface of the robot. So, we subtract the pressure caused by a periodic flow from the pressure field. From the isosurface of the Q-value, the vortex behind the robotic fish is symmetric in a steady flow with respect to the flow direction. However, in a periodic flow, it is asymmetric with respect to the flow direction with a larger vortex on one side. Comparing the surface pressure between steady and periodic flow, the amplitude of pressure in periodic flow is smaller than that of the pressure in steady flow on the same side as Q-value. In this way, the steady and periodic flow are different flow fields on one side of the robotic fish.

REFERENCES

FLOW BOILING HEAT TRANSFER IN A PLATE HEAT EXCHANGER WITH MIXED CHEVRON ANGLE PLATES

Mu-Ting Hsieh, Chien-Yuh Yang* and Fu-Chen Lin

National Central University, No. 300, Jhongda Rd., Jhongli, Taoyuan 32001, Taiwan
*cyyang@ncu.edu.tw

ABSTRACT

Plate heat exchangers (PHE) have been widely used in refrigeration and air conditioning system, chemical reaction processes and many other industrial applications. While applying the plate heat exchanger as an evaporator in an air-condition system, refrigerant flows into the heat exchanger at two-phase condition. Since the length of flow path the in the channels near the inlet port is shorter than that in the channels farther away from the inlet port, most of the fluid will flow through the channels closer to the inlet. This will cause a flow maldistribution and degrade the heat exchanger performance.

This study provides an experimental investigation of refrigerant R-410A boiling in a brazed plate heat exchanger with mixed chevron angle plates. 60 plates with 65^{o} chevron angle were used in the first half and the other 60 plates 35^{o} used in the second half to compose a 120 plates PHE. Since the flow resistance in the lower chevron angle plates is smaller than it in the higher chevron angle plates, it is expected to reduce the difference of flow resistance between these parts and relieve the flow maldistribution of two-phase flow in a multi-plates PHE.

Infrared (IR) thermography was used to observe the PHE side temperature distribution. The observation results showed that the temperature uniformity was improved at low Reynolds conditions by applying mixed chevron angle plates. While Reynolds is high, there is no significant difference of temperature distribution for these two heat exchangers. The heat transfer coefficient in the mixed chevron angle plates PHE is 5.8% higher than that in the single-chevron angle PHE and the pressure drop is 22.9% lower. The overall heat exchanger performance has been significantly improved by the mixed chevron angle design.

CFD ANALYSIS AND VISUALIZATION OF CAVITATING BUBBLY FLOW STRUCTURE IN A CENTRIFUGAL PUMP USING MULTI-PROCESS CAVITATION MODEL

* Akihisa Yamada[1], Taiki Takamine[2], Shin-ichi Tsuda[2] and Satoshi Watanabe[2]

[1] Graduate School of Engineering, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka, 819-0395, Japan
* yamada.akihisa.679@s.kyushu-u.ac.jp
[2]Department of Mechanical Engineering, Kyushu University, 744 Motooka, Nishi-ku, Fukuoka, 819-0395, Japan

ABSTRACT

Numerical prediction of cavitation with higher accuracy is one of the important issues for the development of more reliable high-speed hydraulic machines, and it can also bring us many useful visualization data which are difficult to be obtained experimentally. Currently, a homogeneous model as one of the cavitation models is often used in the field of fluid machinery due to the low computational cost with its simplicity [1]. However, the quantitative accuracy of typical one-equation homogeneous models only focusing expansion/shrinkage of bubbles or the evaporation/condensation is not high.
Multi-Process (MP) model is a four- or five-equation homogeneous model developed by Tsuda et al. (2015) [2]. MP model can take account of main elementary processes in cavitation using the moment method for a size distribution function of cavitation bubbles. It solves several equations for moment variables corresponding to the number density of the bubbles, the densities of the radius and the surface area, and so on. Therefore, it is expected to improve the prediction accuracy compared with typical one-equation models. In this study, unsteady RANS (Reynolds Averaged Navier-Stokes) simulation using MP model was carried out targeting the final stage model of a three-stage centrifugal pump.
We have conducted the simulations at various NPSH (Net Positive Suction Head) conditions to obtain the head breakdown curve at the design flow rate. Good agreement was obtained between the experiment and the simulation in the trend of them. Axial thrust force was almost constant before the head breakdown, then decreased sharply. At lower NPSH, local velocity increase was observed due to blockage of the flow path in the impeller by the growing cavity. In addition, separation occurred downstream of the cavity in the impeller after the head drop. We also visualized each moment variable in MP model, and a size polydisperity of cavitation bubbles as one of the characteristics in MP model was clearly confirmed, indicating that it qualitatively works well also in terms of a useful visualization.

REFERENCES

[1] Singhal, A.K. et al., "Mathematical basis and validation of the full cavitation model", J. Fluid Eng., Vol. 124, No. 3 (2002), pp. 617-624.

[2] Tsuda, S. and Watanabe, S., 2015 Application of multi-process cavitation model for cavitating flow in cold water and in liquid nitrogen around a hydrofoil, Proc. ASME/JSME/KSME 2015 Joint Fluids Eng. Conf. (Seoul, South Korea), (2015), AJKFluids2015-05532.

COLOR PIV FOR MEASURING THREE-DIMENSIONAL WAKE STRUCTURES OF A DELTA WING FROM 0 TO 90 DEGREE IN ANGLE OF ATTACK

*K. Ochi[1], Y. Horimoto[1], H. J. Park[1], Y. Tasaka[1] and Y. Murai[1]

[1] Hokkaido University, Nishi-8-chome, Kita-13-jo, Kita-ku, Sapporo, Hokkaido 060-8628 Japan
*ochi.k@ring-me.eng.hokudai.ac.jp

ABSTRACT

In recent years, the demand for measurement of wing characteristics in the relatively low Reynolds number range ($Re = O(10^3–10^5)$) has increased due to the rapid development of unmanned aerial vehicles. Since the wing characteristics of a delta wing depend on its wake structures, it is necessary to visualize the structures and clarify the mechanism that influences flying stability. In order to measure the wake structure in multiple dimensions, we employ particle image velocimetry with multiple-color-layer illumination (color PIV), which is suitable for 3D measurements for large-scale structures in air flows. In particular, we attempt to visualize the wake structure widely in the angle of attack (AoA) from 0 to 90 degree, where longitudinal vortex transits to vortex shedding via very complicated multi-polar vortical structures.

Using the set-up of three-layer color-coded PIV in a wind tunnel facility, we obtained velocity vector distribution in the three layers simultaneously as a time series. Analyzing vorticity and streamlines from the PIV data, we confirmed a pair of steady leading-edge vortex separation behind the delta wing for AoA less than 25 degree. For AoA close to 90 degree, alternative vortex shedding similar to von Kármán vortex was found with Strouhal number of around 0.2, which was also validated by a hot-wire anemometer by calculating the power spectrum of velocity fluctuations. At an intermediate range of AoA from 30 to 60 degree, the highest turbulent intensity in the examined region of AoA was detected due to the 3D flow separation occurring on the delta wing surface. The wake consists of alternations of an asymmetric vortex pair that produced central downwash intermittently. In the presentation, we will show 3D isosurfaces of such a wake structure obtained from the 3D color PIV data combined with a spatio-temporal correlation analysis.

REFERENCES

[1] Murai, Y., Yumoto, T., Park, H. J. and Tasaka, Y., “Color-coded smoke PIV for wind tunnel experiments improved by eliminating optical and digital color contamination”, Experiments in Fluids, Vol. 62, No. 11 (2021), 231.

[2] Li, J., Zhao, X. and Graham, M., “Vortex force maps for three dimensional unsteady flows with application to a delta wing”, Journal of Fluid Mechanics, Vol. 900 (2020), A36.

EVALUATION OF THE INFLUENCE OF READING PIXEL IN THE DRAG COEFFICIENT CALUCULATED FROM HIGH-SPEED VIDEO CAMERA IMAGES

*T. Miyazaki[1], T. Kikuchi[1] and A. Muramatsu[1]

[1] Nihon University, 7-24-1 Narashinodai, Funabashi, Chiba 274-8501, Japan
*cstm21032@g.nihon-u.ac.jp

ABSTRACT

In an experiment using a ballistic range which launches a projectile, Reynolds number and Mach number can be set independently by changing the pressure of the test range and the projectile velocity. The drag coefficient can be also obtained without the model support interference which is a problem in a wind tunnel experiment. It is used for ground-based experiments of re-entry into the atmosphere [1] and is expected to become a promising experimental technique. The influence of error due to low resolution of high-speed video camera [2] was statistically evaluated to precisely calculate the drag coefficient from the high-speed images measured in a ballistic experiment. The location history of a free-flight projectile was calculated by a true drag coefficient arbitrary setting and photographic condition. The drag coefficient, which was calculated from the projectile location history containing error generated by reading error of pixel adding to this true location history, differs from the true drag coefficient. The average of this difference which was drag coefficient error due to reading error converged by the Monte Carlo simulation which in the reading error of normal distribution random in ranging from $-\sigma$ to $+\sigma$. The influence of reading errors on the calculation of the drag coefficient correlated better with the visualization area and camera resolutions and did not correlate with the resolution per pixel. This prediction was verified by ballistic experiments, in which the flow fields around 6.35 mm diameter polyacetal projectiles launched by a light gas gun were visualized by the shadowgraph method. The standard deviation of the drag coefficient for each conditions indicated similar trends to the simulation. Producing a quantitative index which estimates experimental error enables to set photographic condition of highest precision and to give a statistically expected error bar for a single experiment.

REFERENCES

[1] Hideki. K. et al, “Wake Flow Analysis of a Free Flight Capsule at Transonic Speeds”, Symposium on Shock Waves in Japan, 2019

[2] Tomoharu. K, “Shock Standoff Distance around a Near-Sonic”, graduation thesis, 2017

EXPERIMENTAL INVESTIGATION OF SHEAR-INDUCED BIREFRINGENCE PROPERTIES OF RHEOSCOPIC FLUIDS FOR VISUALIZATION OF STRESS FIELD OF FLUID FLOW

*K. Hayashi[1], S. Tanaka[1], and K. Shirai[1]

[1] Shibaura Institute of Technology, 3-7-5 Toyosu, Koto, Tokyo 135-8548 Japan
*kshirai@shibaura-it.ac.jp

ABSTRACT

Fluid shear stress is one of the fundamental parameters in fluid mechanics. Shear stress at the surface of a solid wall is used for calculating the friction force. In Newtonian fluid, shear stress is proportional to the velocity gradient, which is closely related to the production of turbulence. Measurement of shear stress requires either to use certain analogy or to acquire the velocity gradient in Newtonian fluid. The former method provides surface distributions of the wall shear stress such as liquid crystal coating, shear sensitive polymer film or micro pillar array. In case of the liquid crystal coaring, the wall shear stress is evaluated based on the colour variation of the coated surface, however, these methods are restricted to the surface shear stress. In the latter method, one needs to obtain fluid velocities at two closely located independent positions. The measurement suffers from the uncertainties of the spatial positions as well as the fluid velocities. Hence, direct evaluation of shear stress field is expected in a fluid flow. In the present study, we investigate the feasibility of imaging the stress field of fluid flow. The method relies on the photoelasticity of rheoscopic fluid. In a fluid flow with rheoscopic fluid, elongated colloidal particles align to the induced shear stress and the alignment produces birefringence. The flow-induced birefringence is correlated with shear stress of the flow. Therefore, we have a possibility of evaluating the stress field through proper analysis of the birefringence state. The purpose of this study is to evaluate the birefringence and extinction angle of rheoscopic fluids at different shear conditions. To this end, the known shear stresses are applied to the fluids by using a rotating Couette flow. We used two different working fluids known as rheoscopic fluid. We discuss on the dependencies of the photoelastic coefficients on different shear conditions towards quantitative imaging of fluid stress field.

PSFVIP13-47

INFLUENCE OF THE THIRD STREAM TO A THREE-FEED NON-PREMIXED COMBUSTION SYSTEM

*S. Yadav[1], P. Yu[1], and H. Watanabe[1]

[1] Kyushu University, 6-1 Kasuga-koen, Kasuga, Fukuoka 816-8580 Japan

*imsujeet@tse.kyushu-u.ac.jp

ABSTRACT

Three-stream non-premixed combustion systems have been widely applied in the industry, and the corresponding flamelet-based models have also been developed. However, besides the fuel stream and oxidizer stream, the influences of the third stream to the whole system and the flamelet-based models need further research. In this work, influences of two different third streams, such as pure inactive diluent and oxidizer contained stream, to the chemistry are investigated by means of two-dimensional (2D) direct numerical simulations (DNS). It is confirmed that for a pure inactive third stream the influence of its scalar dissipation rate to the chemistry can be neglected when its mixture fraction is not very large, for instance, less than 0.8. Data obtained from large-eddy simulations (LES) with identical probability density function for mixture fractions further supports this conclusion. As the third stream is oxidizer-contained, its scalar dissipation rate and the cross-scalar dissipation rate affect the chemistry significantly. It is observed in the flamelet library the deviation for temperature distribution from the DNS data cannot be overlooked if the cross-scalar dissipation rate for this oxidizer-contained stream and fuel stream is ignored. It is also found as the mixture fraction of the oxidizer stream increased, the influence of the cross-scalar dissipation rate between the oxidizer-contained third steam and fuel stream is diminished. It is found that with the cross-scalar dissipation rate for this oxidizer-contained third stream and the oxidizer stream is ignored there is almost no difference.

SPATIAL SUPERRESOLUTION BASED ON PROPER ORTHORGONAL DECOMPOSITION AND BAYSIEN ESTIMATION OF SUBSONIC JET FLOW MEASURED BY TWO MAGNIFICATION PIV SIMULTANEOUS MEASUREMENT

*H. Honda[1], Y. Ozawa[1] and T. Nonomura[1]

[1] Department of Aerospace Engineering, Tohoku University, Sendai, Miyagi, 980-8579, Japan
*honda.harutaka.p8@dc.tohoku.ac.jp

ABSTRACT

A subsonic jet generates strong acoustic waves while propelling the airplane. This acoustic wave triggers fatigue failure to the structures and generates noise to the ambient. The researches aiming to predict and reduce the acoustic wave have been conducted for more than half a century[1]. These acoustic waves are generated by the convection and fluctuation of vortices in turbulent flow, and its intensity increases in proportion to the eighth power of the jet velocity[2]. Particle image velocimetry (PIV) is one of the methods for investigating the behaviour of vortices in turbulent flows and has been applied to the subsonic jet[3, 4]. The spatial resolution of the PIV measurement is defined by the field of view of the camera, and the accuracy of the PIV analysis is affected by the resolution of the particles within the image. Since the spatial resolution of locally small flow fields such as vortex structures near the shear layer degrades in a wide field of view, it is difficult to capture the entire flow in a wide field of view while keeping the high spatial resolution. This research aims to address the issue by designing the spatial super-resolution framework to expand the limitation of the PIV measurements.

The reconstruction framework based on proper orthogonal decomposition and Bayesian estimation was designed for the spatial super-resolution of a subsonic jet, and the simultaneous two PIV measurements of a subsonic jet with different magnifications were conducted for obtaining datasets for training and testing the framework. The designed framework was evaluated using the artificial data generated from the average pooling from the high-resolution training data. The reconstruction error of the proposed framework was estimated to be 71% at the resolution ratio of 1:10 using 300 POD modes. In the simultaneous two-magnification PIV measurement, pair images of broad and close-up views of the jet in the same plane were obtained successfully.

REFERENCES

[1] Jordan, P., and Gervais, Y., “Subsonic jet aeroacoustics: associating experiment, modelling and simulation,” Experiments in Fluids, Vol. 44, No. 1, 2008, pp. 1–21.

[2] Lighthill, M. J., “On sound generated aerodynamically I. General theory,” Proceedings of the Royal Society of London. Series A. Mathematical and Physical Sciences, Vol. 211, No. 1107, 1952, pp. 564–587.

[3] Bridges, J., and Wernet, M. P., “The NASA subsonic jet particle image velocimetry (PIV) dataset,” 2011.

[4] Rodríguez, D., Cavalieri, A. V., Colonius, T., and Jordan, P., “A study of linear wavepacket models for subsonic turbulent jets using local eigenmode decomposition of PIV data,” European Journal of Mechanics-B/Fluids, Vol. 49, 2015, pp. 308–321.

NUMERICAL ANALYSIS OF INTERNAL FLOW IN A SEPARATE HERRINGBONE PLATE HEAT EXCHANGER

*D. Fujiwara[1] and S. Torii[1]

[1] Kumamoto University, 2-39-1, Kurokami, Kumamoto 860-8555 Japan
188t2228@st.kumamoto-u.ac.jp

ABSTRACT

The purpose of this study is to visualize the flow inside a heat exchanger with a separate herringbone plate and to clarify the phenomenon of heat transfer enhancement. The analysis software used is STRA-CCM+, a three-dimensional thermal-fluid analysis software. The method used for discretization was the finite volume method (FVM). The AKN model (Abe-Kondoh-Nagano low-Re k-epsilon model) of Reynolds averaged Navier-Stokes (RANS) is used in the analysis. Reliability of the solution is ensured by comparison with experimental values. Separate herringbone plate is plate with the center of the turbulence promoter of the herringbone plate smoothed. Finally, the shape of the plate has been shown to have superior heat transfer and flow performance compared to commonly used plates. In the analysis, high and low temperature fluids are flowed along the wall of a titanium plate for heat exchange. The working fluid is water, and heat is exchanged by counter flow. The high temperature fluid flows upward on the convex side of the plate and the low temperature fluid flows downward on the concave side of the plate. The analysis results shows that the overall heat transfer coefficient is consistent with the experimental values. The velocity distribution between the ribs of the high temperature channel confirms the presence of reattachment points due to flow separation. The temperature distribution shows a high heat transfer coefficient in the region near the reattachment point. Consequently, it can be confirmed that heat transfer enhancement took place inside the heat exchanger due to the high heat transfer coefficient in the reattachment point region.

REFERENCES

[1] Torii, S. and Izumi, K., “Effect of Plate Shape and Size on Thermal Fluid Flow in Channel”, JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER, Vol. 26, No. 4 (2012), pp. 598-601.

VISUALIZATION OF TRANSIENT HEAT TRANSFER BY A DROPLET BOUNCING ON A HEATED SURFACE VIA THERMOGRAPHY

*M. Yoshida[1], S. Yamada[2], Y. Funami[2] and H. Nakamura[2]

[1] Department of Equipment and Structural Engineering, Graduate School of Science and Engineering, National Defense Academy, 1-10-20 Hashirimizu, Yokosuka, Kanagawa 239-8686 Japan
*ed21011@nda.ac.jp
[2] Department of Mechanical Engineering, School of Systems Engineering, National Defense Academy, 1-10-20 Hashirimizu, Yokosuka, Kanagawa 239-8686 Japan

ABSTRACT

In this study, we attempted to measure the transient heat transfer of the Leidenfrost phenomenon caused by a single falling ethanol droplet colliding with a high temperature heat transfer surface. Visualization was done with two visible cameras and an infrared (IR) camera. The droplet behavior was photographed by two visible cameras, one from diagonally above and the other through the heat transfer surface. Simultaneously, we measured the transient temperature distribution by the IR camera. For the measurement, the calcium fluoride (CaF_2) on which an indium tin oxide (ITO) film was formed was used as the heat transfer surface. CaF_2 is transparent to visible light and IR radiation. On the other hand, ITO film is transparent to visible light and opaque to IR radiation. So, it is possible to acquire a visible image of the droplet dynamics seen from above and below, and to measure the transient temperature distribution of the heat transfer surface at the same time. Furthermore, by numerically solving the three-dimensional transient conduction equation, the transient surface heat flux was calculated from the measured temperature distribution. Thus, we were able to capture the dynamics and the transient heat transfer of the Leidenfrost phenomenon. The contact time of the droplet on the heated surface was compared with the empirical formula [1]. In addition, the heat transfer characteristics were compared with the previous study [2].

REFERENCES

[1] Richard, D., Clanet, C., Quéré, D., “Contact Time of a Bouncing Drop”, Nature, Vol. 417, (2002), pp. 811.

[2] Jung, J., Jeong, S., Kim, H., “Investigation of Single-droplet/wall Collision Heat Transfer Characteristics Using Infrared Thermometry”, International Journal of Heat and Mass Transfer, Vol. 92 (2016), pp. 774–783.

VISUALIZATION OF TEMPERATURE FIELD IN A CHANNEL BY LASER INDUCED FLUORESCENCE AND TEMPERATURE DEPENDENCE OF PARTICLE BEHAVIOR IN THE CHANNEL

*K. Hirai, M. Yajima, H. Kato and K. Shirai

Department of Mechanical Engineering, Shibaura Institute of Technology 3-7-5 Toyosu, Koto, Tokyo 135-0045 Japan
*md21098@shibaura-it.ac.jp

ABSTRACT

It has been reported that mixing fine particles into the solution improves the thermal conductivity over that of the original solution.[1] The operational potential of nanoparticles as high-energy carriers has also been shown.[2] From the above, it is important to observe the particle movement near the boundary surface of different phases where heat exchange is active, considering the effects of Brownian motion and thermophoresis. Although the colloidal particle velocity have been measured near the solid-liquid boundary using Evanescent Wave Particle Tracking Velocimetry (EWPTV)[3], no measurements have been made under different temperature conditions. In this study, we measured the temperature field in a microchannel using thermocouples and Laser Induced Fluorescence (LIF) and the colloidal particle velocity in a microchannel near the solid-liquid boundary using EWPTV. The experimental device used in this study was shown to be capable of forming a stable temperature field in the channel. The particle velocity in the channel was found to follow the Hagen-Poiseuille flow as an approximation, but it was found that an apparent slip velocity occurs as it approaches the vicinity of the wall surface. The particle velocity and fluorescence intensity values near the wall within the EW penetration depth range, which is shorter than a hundred micro meter, were proportional to the distance from the wall, yielding linearity. In addition, the particle fluorescence brightness values were found to be inversely proportional to the wall distance. From the above results, we expect that the particle velocities aren't only measured near the wall using EWPTV under various temperature conditions, but also the exponential decay of the EW light intensity was used to feasibly locate particles in three dimensions with a single camera.

REFERENCES

[1] Choi, S.U.S., Eastman, J.A., "Enhanced Heat Transfer using Nanofluids", Patent, United States (2001).

[2] Yang, C., Li, W., Sano, Y., Mochizuki, M. Nakayama, A., "On the Anomalous Convective Heat Transfer Enhancement in Nanofluids: A Theoretical Answer to the Nanofluids Controversy". Journal of Heat Transfer Vol.135 (2013), 054504.

[3] Kato, H., Hirai, K., Maruyama, K., Shirai, K., "Experimental observation of near-wall motions of colloidal particles based on particle tracking using with evanescent light illumination", The 32nd International Symposium on Transport Phenomena (2022).

PSFVIP13-52

VISUALIZATION-BASED MEASUREMENT OF FLOW VELOCITY IN A MOLDED PIPE WITH MATCHED REFRACTIVE INDEX

S. ISHII[1], S. FUKUSHIMA[1] and K. SHIRAI[1]

[1] Department of Mechanical Engineering, Shibaura Institute of Technology
3-7-5 Toyosu, Koto, Tokyo 135-8548 Japan

ABSTRACT

In human body, oxygen and … are transported through thousands of blood or lymph vessels. The flows can be simulated with round shaped pipe flows with small diameters. However, flow velocity in a round pipe is difficult to be measured due to its curvature. Even when the pipe is made of a transparent material, difference of the working fluid, pipe material, and the surrounding environment causes refractions and reflections at their boundaries. These effects lead to image distortions and strong scattering at the boundaries. In the present study, we match the refractive indices of the working fluid, pipe material and the surrounding. We creased the flow tube with silicone elastomer in the laboratory. An advantage of the self-made tube is the flexibility of creating a tube with any internal and external diameters, which is sometimes necessary to meet the required conditions of the experiment. As the working fluid, we used a binary mixture of water and glycerin to match the refractive index of the elastomer. The test section was immersed in a square-shaped box filled with the same liquid as the working fluid. With the refractive-index matching, we use visualization-based method for measuring the flow velocities in the tube. The test section was illuminated with a thin laser sheet. From the images of the seeded tracer particles velocities are obtained by simple particle tracking. The average velocities in the tube is measured and compared to the theoretically estimated ones. So far, the agreement of the measurement and the theoretical ones are within 2 percent. The cause is thought to originate from the fact that refractive index matching reduces distortion due to scattered light, thereby reducing measurement uncertainty. This will be confirmed by an independent measurement using a laser Doppler velocimetry.

Keywords: flow visualization, velocity measurement, pipe flow, flexible tube, silicone, curvature, spatial average, refractive index matching, particle tracking, particle image velocimetry

DIFFERENCES IN FLOW AROUND GOLF BALLS WITH DIFFERENT DIMPLE OCCUPANCY, VOLUME RATIO AND DEPTH

*K. Moriyama[1], H. Okanaga[1]

[1] Tokai University, 4-1-1 Kitakaname, Hiratsuka, Kanagawa, 259-1292, Japan
*1cemm098@mail.u-tokai.ac.jp

ABSTRACT

The surface of a golf ball has many dents called dimples to improve the flight distance. The dimple occupancy and dimple volume ratio to the golf ball is very important in particular to the aerodynamics characteristics. Therefore, we made 15 model balls with different occupancy and volume ratio. The occupancy was varied using the diameter and number of dimples. In addition, the dimple depth was changed to the volume ratio. We conducted lift and drag measurement experiments by wind tunnel experiments. Then, a flight trajectory simulation was performed using the obtained lift and drag data, and it was clarified how the occupancy and the dimple volume ratio affect the flight distance. The most flying model ball had a high occupancy of 83.1% and a dimple volume ratio of 11.3×10^{-3}. This model ball is characterized by a larger dimple diameter and a shallower dimple depth than other spheres. We will visualize the flow of each of the spheres using the wind tunnel experiments by the spark tracking method. Then, we clarify the relationship between the aerodynamic characteristics and to the difference in the occupancy and the dimple volume ratio from flow visualization.

REFERENCES

[1] K. Aoki, M. Nonaka, T. Goto: Effect of dimple structure on the flying characteristics and flow patterns of a golf ball, The Engineering of Sports 5, Vol.1, pp.49-55, 2004.

[2] 2. Bin Lyu, Jeff Kensrud, Lloyd Smith, Taylor Tosaya: Aerodynamics of Golf Balls in Still Air †, The 12th Conference of the International Sports Engineering Association, Proceedings 2018, 2, 238

[3] 3. A.J. Smits & D.R. Smith: A new aerodynamic model of a golf ball in flight, Science and Golf II, E.&F.N. Spon, London (1994), pp. 340-347.

A TIME-RESOLVED THREE-DIMENSIONAL DENSITY MEASUREMENT FOR AN ASYMMETRICAL UNSTEADY SUPERSONIC IMPINGING JET USING A FIBRE BOS TECHNIQUE

*T. Kuroda[1], M. Kobayashi[1] and T. Ukai[1]

[1] Osaka Institute of Technology, 5-16-1 Omiya, Asahi, Osaka 535-8585, Japan
*kuroda3939@gmail.com

ABSTRACT

A supersonic jet with long shock cells impinges on the planetary surface at a high altitude for a long time [1], which results in the diffusion of sands. The diffused sand causes damage to the exploration rover's equipment. To reduce such kind of risk, a jet-jet interaction technique, which can shorten the length of shock cells, would be useful. Kobayashi *et.al.* investigated the effect of jet-jet interaction on supersonic impinging jets, and the surface flow of a plate was visualized by an oil flow visualization technique [2]. They showed that an asymmetric surface flow was observed in the jet-jet interaction case, and a curved flow pattern, which is a linearly symmetric pattern, appeared. From several previous studies [3, 4], we deduce that the asymmetric flow pattern on the plate is caused by an asymmetric vortex, but further investigations are necessary to fully understand vortex motion on the plate. In this study, we used a Fibre BOS technique that temporal variation of a quantitative three-dimensional density field can be measured using a single camera [5], to visualize the Mach 2 supersonic jet with jet-jet interaction impinging a plate vertically. A Fibre BOS technique can obtain the same number of projections as the number of image fibre used, and nine projections were captured in this experiment. The flow visualization captured the structure of the asymmetrical unsteady flow in the supersonic impinging jet with jet-jet interaction. In other words, jet-jet interaction causes the unsteady flow field on the plate. Additionally, jet-jet interaction reduced the density change on the plate. Prospects are to reconsider the experiment after confirming the reproducibility of the image reconstruction of the Fibre BOS technique.

REFERENCES

[1] Mehta, M., Renno, N.O., Marshall, J., Grover, M.R., Sengupta, A., Rusche, N.A., Kok, J.F., Arvidson, R.E., Markiewicz, W.J., Lemmon, M.T. and Smith, P.H., “Explosive erosion during the Phoenix landing exposes subsurface water on Mars,” Icarus, Vol. 211, No.1 (2011), pp. 172, 194.

[2] Kobayashi, M. and Ukai, T., “Surface and spatial flow structure of an underexpanded jet with vertical jet interactions impinging on perpendicular and inclined plates”, AIAA Paper 2022-2582 (2022).

[3] Geers, L.F.G., Hanjalic, K. and Tummers, M. J. “Wall imprint of turbulent structures and heat transfer in multiple impinging jet arrays,” Journal of Fluid Mechanics Vol. 546 (2005).

[4] Thielen, L., Jonker, H.J.J. and Hanjali, K., “Symmetry breaking of flow and heat transfer in multiple impinging jets,” International Journal of Heat and Fluid Flow, Vol. 24, No. 4 (2003), pp. 444-453.

[5] Ukai, T., “The principle and characteristics of an image fibre Background Oriented Schlieren (Fibre BOS) technique for time-resolved three-dimentional unsteady density measurements”, Experiments in Fluids, Vol. 62 (2021).

PRELIMINARY STUDY OF MICROJETS FROM RECTANTULAR CONVEREGNT NOZZLES WITH HIGH-ASPECT RATIOS

*S. Yoshimi[1], S. Nakao[1], and Y. Miyazato[1]

[1] The University of Kitakyushu, 1-1 Hibikino, Wakamatsu-ku, Kitakyushu, 808-0135 Japan
* acadress443@gmail.com

ABSTRACT

Microjets issued from rectangular nozzles with high aspect ratios (AR: the ratio of the major dimension to the minor dimension at the nozzle exit) have been widely used for gas wiping in continuous galvanizing line. However, little is known about the flow characteristics of such microjets because of challenging issues. Handa et al. [1] applied the molecular tagging velocimetry (MTV) for supersonic microjets issued from a convergent-divergent nozzle with a design Mach number of 2.0 and a rectangular shape of 500 μm × 5000 μm (AR = 10) at the nozzle exit. They clarified effects of Reynolds number on the microjet structure including the supersonic core length and the velocity decay process. Aniskin et al. [2] investigated effects of the Pitot microtube diameter on the main parameters of plane underexpanded microjets such as the shock-cell spacing and the supersonic core length. Two convergent nozzles with sizes of 22.3 μm × 2593 μm (AR = 116) and 83.3 μm × 3823 μm (AR = 46) at the nozzle exit and two glass Pitot tubes with inner diameter (μm) /outer diameter (μm) of 24/70 and 16/42 were used in the experiments. As a result, they found that the length of the first shock-cell measured by the Pitot microtube is overestimated in comparison with that visualized by the shadowgraph method, but the Pitot tube diameter has a minor effect on the supersonic core length.

In the present study, the flow structure of a shock-containing microjet issued from a rectangular convergent nozzle with a high-aspect ratio is investigated experimentally by the rainbow schlieren deflectometry. A convergent nozzle with a rectangular shape of 300 μm × 3000 μm (AR = 10) at the nozzle exit is used in the experiments and the nozzle pressure ratio (NPR: the ratio of the plenum pressure upstream of the nozzle to the back pressure) is held constant at 4.0 to produce a Mach stem in the first shock-cell of the jet plume. In addition, the flow field of rectangular microjets is simulated by solving the Reynolds-averaged Navier-Stokes (RANS) equations to investigate the near-field shock structure of the jets. The Menter k-ω shear-stress transport (SST) model is used for the simulations because it has been widely used and trusted model in areas of aerospace community. Three-dimensional flow structure of shock-containing jets issued from a rectangular convergent nozzle with a high aspect ratio is displayed by computer flow visualizations.

REFERENCES

[1] Handa, T., Mii, K., Sakurai, T., Imamura, K., Mizuta, S. and Ando, Y., " Study of Supersonic Rectangular Microjets Using Molecular Tagging Velocimetry", Experiments in Fluids, (2014), 55: 1725.

[2] Aniskin, V. M., Timofeev, I. V., Maslov, N. A., and Tsibulskaya, E. O., "Effect of the Pitot Microtube Diameter on Pressure Measurement in Plane Supersonic Microjets", Flow Measurement and Instrumentation, Vol. 70 (2019), 101655.

IMPROVEMENT OF PARTICLE TRACKING VELOCIMETRY WITH DEEP LEARNING

*T. Yano[1] and Y. Nakanishi[1]

[1]Kanagawa University, 3-27-1 Rokkakubashi, Kanagawa-ku, Yokohama, Kanagawa 221-8686 Japan
*t-yano@kanagawa-ua.c.jp

ABSTRACT

Understanding of the velocity distribution in a fluid flow is important in the field of fluid engineering, and the measurement techniques of flow velocity have been studied for a long time. The flow measurement with video camera and fine tracer particles, which measures the flow velocity based on the traceability of particles to the fluid motion, is an effective technique. This technique can obtain multipoint velocities simultaneously, and it can be classified into particle tracking velocimetry (PTV) and particle image velocimetry (PIV) depending on the method for deriving the velocity. When the number density of tracer particles is the same, the spatial resolution of PTV is supposed to be higher than that of PIV because the former determines the velocity from the motion of individual particles and the latter determines from the correlation of the pattern of particle distribution. However, there is a problem in PTV that the measurement accuracy deteriorates with increase in the number density of tracer particles. This problem has been challenged by several researchers (e.g., Schanz et al. [1] developed the technique known as "Shake-The-Box"). The purpose of this study is to improve the accuracy of PTV for densely seeded particle images by using the deep learning.

The main procedures of PTV are as follows: (1) detect tracer particles on the image, (2) track motions of tracer particles during short time intervals, and (3) convert results from image coordinates to absolute coordinates. Currently, the deep learning is applied to procedure (1). Firstly, as many candidates of tracer particles on the image are obtained by Hough transform method, which is an analysis technique for extracting circles on the image. After that, the correctness/incorrectness of the obtained candidates is classified by the supervised deep learning with convolution neural network (CNN) [2]. The accuracy of particle detection can be improved by eliminating the candidates judged to be incorrect. As a result of the verification using synthetic particle images, the detection of tracer particles on dense particle images were realized with good accuracy (e.g., approximately 98% and 86% of particles were correctly detected for images with 0.01 and 0.05 particles per pixel (ppp), respectively), and the effectiveness of the present method was demonstrated.

REFERENCES

[1] Schanz, D., Gesemann, S. and Schröder, A., "Shake-The-Box: Lagrangian Particle Tracking at High Particle Image Densities", Experiments in Fluids, Vol. 57, No. 5 (2016), 70.

[2] LeCun, Y., Bottou, L., Bengio, Y. and Haffner, P., "Gradient-Based Learning Applied to Document Recognition", Proceedings of the IEEE, Vol. 86, No. 11 (1998), pp. 2278–2324.

EXPERIMENTS ON FLOW FIELD CONTROL USING A UNIDIRECTIONAL ACOUSTIC FLOW GENERATOR DEVICE

* H. Kawashima[1], S. Miyazaki[1] and T. Hiroi[1]

[1] National Maritime Research Institute, 6-38-1 Shinkawa, Mitaka Tokyo 181-0004 Japan
*kawasima@m.mpat.go.jp

ABSTRACT

A flow field control device using an acoustic streaming is being developed to control the turbulent flow field in a turbulent boundary layer from the wall surface area. By inputting high-frequency power to a transducer installed on a piezoelectric substrate, surface acoustic waves (SAW) can be radiated on the piezoelectric substrate to generate acoustic streaming in a liquid adjacent to the substrate. A commonly used method of generating acoustic streaming has been to use a SAW device composed of a bidirectional interdigitated electrodes (IDT) on a piezoelectric material base of single crystal lithium niobate (LiNbO3) [1][2][3]. This method was not suitable as a flow field control device because the acoustic streaming was generated at an angle of 67.5 degrees to the wall surface due to the sound speed of the piezoelectric substrate being too fast compared to the sound speed of water, and the vertical component of the acoustic stream to the wall surface was larger than the horizontal component. Also, because the IDT is bidirectional, the acoustic streaming is generated in a V-shape, and the horizontal component of the flow velocity is the same back and forth, which was another reason why it was not suitable as a flow field control device.

The flow field control device employs BGO (Bi12GeO20) [4], a piezoelectric substrate with surface wave sound velocity closer to sound velocity of water, to reduce the angle of the generated acoustic streaming to the wall surface to about 30 degrees. By applying a unidirectional circuit [5] for generating surface waves, the acoustic streaming can be generated in almost one direction. Acoustic streaming tests were conducted using the flow field control device in a water tank. The acoustic streaming was then generated using the developed flow field control device and measured by PIV. The angle of acoustic streaming was 29.9 degrees to the wall surface, and the maximum velocity was 31.6 mm/s when power was applied at a voltage of 40 V. Input voltage and maximum flow velocity were generally proportional.

REFERENCES

[1] Shiokawa, S., Matsui, Y., Ueda T., “Study on SAW Streaming and its Application to Fluid Devices”. *Proceedings of 10th Symposium on Ultrasonic Electronics 1989, Japanese Journal of Applied Physics,* Vol. 29 (1990), pp.137-139.

[2] Maezawa, M., Kamakura, T., “Sound Field Evaluation and Flow Visualization in Water by SAW device”. *IEICE technical report,* (2007), pp.47-51.

[3] Kong, D., Kurosawa, M. K., “A Novel Approach to Swimmer Actuation via Surface Acoustic Wave”. *Proceedings of Symposium on Ultrasonic Electronics,* Vol. 39 (2018)

[4] Cambell, C., “Surface Acoustic Wave Devices and Their Signal Processing Applications”. *Academic Press,* (1989), pp.20

[5] Kondo, J., “Development of Liquid-Phase Sensing System Using Acoustic Wave Devices”. *Technical Report of Industrial Technology Research Grant Program in FY2006,* (2007)

PRELIMINARY STUDY OF UNDEREXPANDED SONIC JETS FROM ELLIPTIC CONVERGENT NOZZLES

*T. Nagata[1], M. M. Islam[1,2], T. Miyaguni, S. Nakao[1], and Y. Miyazato[1]

[1] The University of Kitakyushu, 1-1 Hibikino, Wakamatsu-ku, Kitakyushu, 808-0135 Japan
* ntatsuya2000@outlook.com

ABSTRACT

It has been recognized that elliptic nozzles provide improvements in flow mixing within the supersonic jets when compared with the axisymmetric nozzles. Menon and Skews [1] experimentally investigated the gross features of shock-containing elliptic jets by the shadowgraph technique and simulated the near-field shock structure by solving the Reynolds-averaged Navier-Stokes (RANS) equations. Most recently, Rao et al. [2] applied the proper orthogonal decomposition (POD) and dynamic mode decomposition (DMD) to the time-resolved schlieren data of under-expanded supersonic jets issuing from a high aspect ratio elliptic nozzle and extracted the coherent information associated with the screeching behaviour and closure mechanism of a feedback loop for the flapping mode. However, the flow characteristics of the near-field shocks of elliptic underexpanded jets are still elusive. In the present study, shock-containing elliptic jets are investigated experimentally by the rainbow schlieren deflectometry where a rainbow filter with fine colour gradations is placed perpendicular to the nozzle axis in the schlieren cut off plane in such a manner that the streamwise density gradient can be captured. An elliptic convergent nozzle with an aspect ratio of AR = 2 at the exit (AR is defined as the ratio of the major dimension to the minor dimension at the nozzle exit) is used in the present study. The semi-major and semi-minor dimensions at the nozzle exit are 5 mm and 2.5 mm, respectively. The nozzle pressure ratio (NPR = p_{os} / p_b) of the plenum pressure p_{os} upstream of the nozzle to the back pressure p_b is held constant at NPR = 3.0 to produce a slightly underexpanded sonic jet. Rainbow schlieren pictures of the elliptic jets are taken independently from the jet major and minor axis directions by rotating the nozzle about its central axis. The flow structures of the elliptic jets at the jet major and minor axis plane views are demonstrated with fine colour gradations that correspond to the density gradient in the streamwise direction. The shock-cell spacing of the first shock-cell of the elliptic jet plume is obtained from the hue distribution along the jet centreline and it is compared quantitatively with that estimated by the vortex sheet model [3] for elliptic underexpanded jets.

REFERENCES

[1] Menon, N. and Skews, B. W., “Shock Wave Configurations and Flow Structures in Non-Axisymmetric Underexpanded Sonic Jets, Shock Waves, Vol. 20, No. 3 (2010), pp.175-170.

[2] Rao, A. N., Kushari, A. and Mandal, A. C., “Screech Characteristics of Under-Expanded High Aspect Ratio Elliptic Jet”, Physics of Fluids, Vol. 32, No. 7 (2020), 076106.

[3] Tam, C. K. W., “The Shock-Cell Structures and Screech Tone Frequencies of Rectangular and Non-Axisymmetric Supersonic Jets, J. Sound and Vibration, Vol. 121, No. 1 (1988), pp. 135-147.

OSCILLATORY CHARACTERISTICS OF SHOCK TRAINS IN SQUARE DUCTS

*T. Inadomi[1], R. Fukunaga[1], T. Takeshita[1], S. Nakao[1], and Y. Miyazato[1]

[1] The University of Kitakyushu, 1-1 Hibikino, Wakamatsu-ku, Kitakyushu, 808-0135 Japan
* takatoac@outlook.jp

ABSTRACT

Shock trains appear in a variety of flow devices such as supersonic wind tunnels, supersonic ejectors, and isolators of scramjet engines [1]. It is well known that shock trains oscillate and induce strong pressure fluctuations, leading to vibration of the fluid machines or significant noise. Recently, understanding of the behaviour of an unsteady shock train is of great need in the context of the design of scramjet engines [2] since a higher engine thrust can be achieved when a shock train is confined inside the isolator of the scramjet engine. Furthermore, shock train oscillations have a significant effect on the unstart problem. Takeshita et al. [3] applied the high-speed Mach-Zehnder interferometry for an isolator shock train and performed the fringe-shift analysis for the unsteady interferograms of the shock train to acquire the oscillatory characteristics of the first, second and third shocks in the shock-train.

In the present study, the oscillatory characteristics of a shock train in a square duct is investigated by the high-speed Mach Zehnder interferometry with the finite-fringe setting where a He-Ne laser with a wavelength of 632 nm is used as a light source. A Laval nozzle with a design Mach number of 1.5 and a square shape of 10 mm × 10 mm at the exit is used in the present experiments and it is followed by a square duct with a streamwise length of 50 mm. Interferograms of the shock train are recorded in 1,000 pictures with 21,600 frames per second by a high-speed digital camera (Photron, FASTCAM SA1.1). The effective spatial resolution of the present imaging system is 24 pixel/mm or 41 μm/pixel.

The time-resolved density fields of the shock train are reconstructed by the fringe-shift analysis of the interferograms of the shock train and the density calculation from the fringe shift. The two-dimensional fluctuating density field including the shock train, the following mixing region, and the boundary layer flows within the interaction region are demonstrated quantitatively.

REFERENCES

[1] Matsuo, K., Miyazato, Y. and Kim, H. D., “Shock Train and Pseudo-Shock Phenomena in Internal Gas Flows”, Progress in Aerospace Sciences, Vol. 35, No. 1 (1999), pp. 33-100.

[2] Gnani, F., Zare-Behtash, H. and Kontis, K.,”Pseudo-Shock Waves and Their Interactions in High-Speed Intakes, Progress in Aerospace Sciences, Vol. 82 (2016), pp. 36-56.

[3] Takeshita, T., Nakao, S., and Miyazato, Y., “Application of Mach-Zehnder Interferometers for Isolator Shock Trains”, ACC J., Vol. 25, No. 1 (2019), pp. 68-77.

VISUALIZATION OF BUBBLE DYNAMICS INSIDE THE PROOFING BREAD DOUGH USING X-RAY MICROTOMOGRAPHY

*K.Kimura[1], K.Kikuchi[1], K.Numayama[2], T.Ishikawa[2] and T.Miyagawa[3]
[1]Department of Finemechanics, Graduate School of Engineering, Tohoku University
6-6-01 Aramaki Aza Aoba, Sendai, Miyagi 980-8579 Japan
[2]Graduate School of Biomedical Engineering, Tohoku University
6-6-01 Aramaki Aza Aoba, Sendai, Miyagi 980-8579 Japan
[3]Graduate School of Science and Technology, Hirosaki University
3, Bunkyo-cho, Hirosaki, Aomori 036-8561 Japan

ABSTRACT

Bread has a bubble structure that is thought to affect its mechanical properties and texture. The bubble structure is formed when yeast ferments and releases carbon dioxide gas. In the previous research, it was reported that the relative density, bubble diameter, and open cell ratio affect mechanical properties such as Young's modulus and yield stress in baked bread [1]. The purpose of this study is to clarify the relationship between the three-dimensional bubble structure and mechanical properties in bread dough during fermenting and proofing.

We made a dough sample of 3 grams following the previous research [2] (strong flour, water, salt and yeast were added as the main ingredients and the stirring time was 60 rpm for 10 minutes.) and packed it to the bottom of a polystyrene tube bottle. We used X-ray microtomography to visualize the internal structure of opaque bread dough. We used MorpholibJ, a plugin for Fiji to detect the volume and number of bubbles from image data. In the preliminary survey, by using this analysis method, we succeeded in detecting more than 65% of bead particles with a uniform size of about 0.5 mm to 0.71 mm in diameter.

As a result, we succeeded in detecting bubbles from the X-ray tomographic image data and obtaining the number and volume of bubbles. Comparing the data at 10 minutes and 60 minutes after the start of fermentation, both the number and volume of bubbles increased. In conclusion, we succeeded to evaluate non-destructively the growth of bubbles in the bread dough due to yeast fermentation.

REFFERENCES

[1] M. C. Zghal, M. G. Scanlon, H. D. Sapirstein, "Cellular Structure of Bread Crumb and its Influence on Mechanical Properties", Journal of Cereal Science, 36, (2002), 167-176

[2] S. Chevallier, R. Zúñiga, A. Le-Bail, "Assessment of Bread Dough Expansion during Fermentation", Food Bioprocess Technol, 5, (2012), 609-617

Improvement of Robustness on Real-time Flow Field Measurement using Sparse Processing PIV

*Chihaya Abe[1], Naoki Kanda[1], Sayumi Kaneko[1], Kumi Nakai[1] and Taku Nonomura[1]

[1] Tohoku University, 6-6-1 Aramaki, Sendai, Miyagi, 980-8579, Japan
*chihaya.abe.p1@dc.tohoku.ac.jp

ABSTRACT

Robustness of sparse processing particle image velocimetry (SPPIV) was improved, and the flow velocity field was measured in real time by improved SPPIV, where SPPIV is a method to estimate the entire flow field from limited results of sparsely located PIV-analysis interrogation window but was suffered from being high resolution because of outliers appearing in the PIV analysis. In this study, high-resolution velocity field estimation was conducted by shrinking the interrogation window from 32×32 pixel to 16×16 and 8×8 pixel, and the robustness of the improved SPPIV was investigated. In the improved SPPIV, recursive local-correlation PIV and robust principle component analysis (RPCA) based on gappy proper orthogonal decomposition (gappy-POD) are applied to reduce the noise and improve the spatial resolution of training data. Robust Kalman filter was applied for state estimation of real-time measurements, which involves an optimization problem that is iteratively calculated by fast iterative shrinkage-thresholding algorithm (FISTA). In the wind tunnel test, the PIV measurement and real-time measurement using improved SPPIV were conducted for the flow velocity field around the NACA0015 airfoil model. The results of this test show that the high-resolution flow velocity field can be estimated by the improved SPPIV. It was found that robust Kalman filter is effective to reduce the estimation error significantly when the interrogation window is small. It is because the more the number of observation points increases, the more estimated vectors are affected by noise in difficult measurement conditions using small interrogation window. On the other hand, the hyperparameter in the optimization problem is highly relevant to the results. Hence, parameter settings are important. In terms of calculation time of SPPIV, it was found that the processing time of FISTA in robust Kalman filter is a small percentage of the total. Therefore, improved SPPIV proposed in this study is sufficiently feasible for real-time flow field measurement.

REFERENCES

[1] Kanda, N., Nakai, K., Saito, Y., Nonomura, T. and Asai, K., “Feasibility study on real-time observation of flow velocity field using sparse processing particle image velocimetry”, Transactions of the Japan Society for Aeronautical and Space Sciences, Vol. 64, No. 4 (2021), pp. 242-245.

[2] Mattingley, J. and Boyd, S., “Real-time convex optimization in signal processing –Recent advances that make it easier to design and implement algorithms”, IEEE Signal Processing Magazine (2010)

DEPENDENCE OF MEASUREMENT ACCURACY AND TIME DELAY OF LIFETIME-BASED DUAL-LAYER PSP/TSP ON FILM THICKNESS

*H. Mori[1], T. Kuroki[1] and H. Nishiyama[1]

[1] Kyushu University, 744 Motooka, Nishi-ku, Fukuoka 819-0395 Japan
*hide-m@mech.kyushu-u.ac.jp

ABSTRACT

Dual-layer PSP/TSP (DL-PTSP)[1] technique is a method to compensate for the temperature sensitivity of pressure-sensitive paint (PSP) by composite measurement of pressure and temperature, which is based on the combination of PSP and temperature-sensitive paint (TSP) technique. Applying lifetime-based calibration to DL-PTSP enables simultaneous measurement of pressure and temperature by DL-PTSP with a single-camera equipment, if the lifetime of PSP luminophore is very different from that of TSP luminophore[2]. However, nonzero thickness of intermediate paint layers of DL-PTSP, separating the upper PSP layer and the lower TSP layer physically, may cause temperature difference between the PSP layer and the TSP layer, and the delay of time response of temperature measurement by the TSP layer. Because the intermediate layers of DL-PTSP have the role to avoid photophysical interaction between PSP luminophore and TSP luminophore, such as quenching of luminescence, the intermediate layers are indispensable for DL-PTSP, although it is desirable to decrease their thickness. It means that there is "optimum" thickness of the intermediate layers of DL-PTSP. In this study, the relation between the thickness of the intermediate layers and the properties of DL-PTSP with the lifetime-based calibration method, such as the sensitivity of pressure and temperature and the time response of temperature measurement, are examined. It is clarified that the independence of pressure measurement and temperature measurement are kept for DL-PTSP by using the lifetime-based calibration method, when the total thickness of the intermediate layers is as thin as 12μm. The time delay of the temperature measured by the TSP layer from that of the PSP layer is as small as milliseconds if the thickness of the intermediate layers is 12μm.

REFERENCES

[1] Moon, K.J., Mori, H., Ambe, Y. and Kawabata, H., "Development of Dual-layer PSP/TSP System for Pressure and Temperature Measurements in Low-speed Flow Field", Proc. of ASME-JSME-KSME Joint Fluids Engineering Conference 2011, AJK2011-11020 (2011).

[2] Moon, K.J., Mori, H. and Furukawa, M., "Simultaneous measurement method of pressure and temperature using dual-layer PSP/TSP with lifetime-based method", Meas. Sci. Technol., 29-12, DOI: 10.1088/1361-6501/aae408 (2018).

IMAGE PROCESSING OF FLOW VISUALISATION PICTURES TO DETERMINE THE STRUCTURE OF TRANSITIONAL CHANNEL FLOW OF AQUEOUS POLYMER SOLUTIONS

* T. Okuda[1], X. Song[1], S. Yimprasert[1], P. H. Alfredsson[2] and M. Matsubara[1]

[1] Shinshu University, 4-17-1 Wakasato, Nagano, Nagano 380-8553 Japan
21w4016j@shinshu-u.ac.jp
[2] KTH Royal Institute of Technology, Stockholm, SE-100 44 Sweden

ABSTRACT

It is well known that the addition of polymer to a wall-bounded turbulent flow reduces drag, which is hypothesized to be due to the suppression of the turbulent eddy motion by the dashpot-like behaviour of the polymer [1]. The effect of polymer addition to transitional channel flow was investigated by flow visualization [2], and it was confirmed that the transition is delayed by polymer addition. In the present study, we quantify the effect of polymer addition by using image processing to obtain the intermittency ratio, which is the fraction of turbulence in the flow, as well as the scales of turbulence and streaky structures.

The experimental apparatus is a channel consisting of two flat glass walls. The total length is 7320 mm and the height 7 mm. After an initial length where a trip wire is attached to one of the walls, the channel width decreases thereby increasing the Reynolds number and the flow becomes turbulent. The fully turbulent flow then enters a section where the width increases which makes the Reynolds number decrease until it flows into the constant-width channel test section where it becomes intermittent with mixed laminar and turbulent regions.

Polyacrylamide was used as the polymer. The flow was visualized by suspending reflective pearl flakes into the water/polymer mixture and illuminating the flow downstream the exit of the enlarged section with LED lights. The visualized flow is captured with a high-resolution digital camera. Two moving averages of the visualized images were taken, and the local standard deviation of the difference of the two moving averages was used to distinguish turbulent from laminar flow and to determine the intermittency ratio. Furthermore, by taking autocorrelations of the laminar and turbulent parts cut out from the images, the streamwise and spanwise scales of flow structures were estimated from the correlation values.

The results show that the transition to turbulent flow is delayed with polymer addition. It was also found that the length of the streak structures appearing around the turbulent patches increases with increasing polymer concentration. These results indicate that polymer addition does not only delay transition, but also significantly affects the turbulent structure in the transitional flow.

REFERENCES

[1] Den Toonder, J., Hulsen, M., Kuiken, G., and Nieuwstadt, F., “Drag reduction by polymer additives in a turbulent pipe flow: Numerical and laboratory experiments”, Journal of Fluid Mechanics, Vol. 337 (1997), pp. 193-231.

[2] Yimprasert, S., Alfredsson, P. H. and Matsubara, M., “Effects of polymer addition on transition and length scales of flow structures in transitional channel flow”, 7th International Conference on Jets, Wakes and Separated Flows, (2022).

THREE-DIMENTIONAL DENSYTY MEASUREMENT USING SIMULTANEOUS MULTI-ANGLE BOS MEASUREMENT SYSTEM IN A SUPERSONIC WIND TUNNEL

*Masato Yamagishi[1], Yusuke Hirose[2], Shinsuke Udagawa[3], Tatsuro Inage[4] and Masanori Ota[1]
[1] Chiba University, 1-33 Yayoi Inage Chiba, 263-8522, Japan
*masa0326yama@chiba-u.jp
[2] Tokyo Salesian College of Technology,
4-6-8 Oyamagaoka, Machida City, Tokyo, 194-0215, Japan
[3] Tokyo Metropolitan College of Industrial Technology, Arakawa, Tokyo, 1168523, Japan
[4]Shonan Institute of Technology, Fujisawa, Kanagawa, 2518511, Japan

ABSTRACT

The 3D-BOS technique is the three-dimensional density field visualization method for fluid. In the past studies, the flow field around jets has been measured [1], and we succeeded in measuring the density around the projectile in the ballistic range for the first time [2]. On the other hand, the accuracy verification of the 3D-BOS technique is performed by comparing it with the CFD result or conventional schlieren method, and there is no example of simultaneous measurement of force and pressure in a supersonic wind tunnel. The 3D-BOS measurement in the supersonic wind tunnel requires repeated experiments and the wind tunnel disturbance is significant. Therefore, the measurement procedure is complex.

In this study, the 3D-BOS technique using the simultaneous multi-angle BOS measurement system for density measurement is applied in a supersonic wind tunnel. With the proposed method, the number of projections can be increased for the same number of repetitions, and more accurate 3D reconstruction results are expected.

The experiment is conducted in a supersonic wind tunnel at JAXA / ISAS, and the flow around the Cone Model installed in a uniform flow with a Mach number of 2.0 is measured. In the conventional method, measurement with a single camera is mainly used. In contrast, the proposed method enables the measurement of the multi-angle projection data at once by the simultaneous multi-angle BOS measurement system using 12 digital cameras. In the projection data measured by the BOS method, the wind tunnel disturbance is observed in the entire image, and the result of the conventional method shows that the shock wave shape was distorted. On the other hand, the proposed method succeeds in suppressing the distortion of the shock wave shape due to the influence of the disturbance.

REFERENCES

[1] F. Nicolas, D. Donjat, F. Micheli, G. L. Besnerais, A. Plyer, P. Cornic,F. Champagnat, Y. Michou “EXPERIMENTAL STUDY OF A COUNTER-FLOW JET IN ONERA’S S1MAWIND TUNNEL BY 3D BACKGROUND ORIENTED SCHLIEREN” 18th International Symposium on Flow Visualization Zurich, Switzerland (June 26-29, 2018)

[2] Y. Hirose, K. Ishikawa, Y. Ishimoto, T. Nagashima, M. Ota, S. Udagawa, T. Inage, K. Fujita, H. Kiritani, K. Fujita, K. Ohtani, H. Nagai “The Quantitative Density Measurement of Unsteady Flow around the Projectile” Journal of Flow Control, Measurement & Visualization, Vol. 7 No.2, (2019)

HIGH ACCURATE DENSITY MEASUREMENT OF TRANSONIC FLOW FIELD AROUND THE REENTRY CAPSULE MODEL

*Yutaro Katagiri[1], Nao Kosaka[1], Masato Yamagishi[1], Yusuke Hirose[2], Masanori Ota[1]
Masayuki Nonura[3], Koji Hujita[3], Kiyonobu Kiyota[3] and Hiroki Nagai[3]
[1] Chiba University, 1-33 Yayoi Inage, Chiba, 263-8522, Japan
*ykatagiri.17@chiba-u.jp
[2]Salesian Polytechnic, 4-6-8 Oyamagaoka, Machida, 194-0215, Japan
[3]Institute of Fluid Science, Tohoku University, 2-1-1 Katahira Aoba, Sendai, 980-8577, Japan

ABSTRACT

Quantitative measurement of unsteady flow field around the test model installed in the wind tunnel is investigated using Background Oriented Schlieren (BOS) technique. The re-entry capsule is planned to use in the sample return mission, which plans to collect the astronomical sample and bring it back to our planet. Understanding the unsteady and complex flow field around flying objects like the re-entry capsule in transonic flow is very important [1]. In the previous study, computational fluid dynamics (CFD) and qualitative experiments such as the schlieren method are mainly used to find out flow field characteristics. We have studied the BOS technique that is remarked as a quantitative experimental method using a simple optical setup and image analysis [2]. However, this method was hardly used in the transonic flow because of its sensitivity. Therefore, the BOS technique with telecentric optical system [3] and digital projector is introduced in this study. They enable not only to obtain a parallel projection of the flow field and test model but to adjust the distance from the background to the test model. As a result, the sensitivity of the BOS technique is improved, and quantitative experiment visualization is achieved. Visualization of the unsteady flow field around the re-entry capsule model shows flow separation from the front shoulder part of the capsule and flow disturbance in the wake region. Thanks to quantitative visualization, unsteady flow characteristic around the capsule model can be compared with oscillation of the model, and the relation between them is revealed.

REFERENCES

[1] Masayuki Nomura, Koji Fujita, and Hiroki Nagai., “Experimental Study of a Re-entry Capsule to Reduce Self-excited Oscillation at Transonic Speed”, the 39th Aerospace Numerical Simulation Symposium, 2021

[2] Meier GEA. Computerized background-oriented schlieren. Exp. Fluids Vol. 33, 2002, pp 181-187.

[3] Masanori Ota, Friedrich Leopold, Ryusuke Noda, Kazuo Maeno. Improvement in spatial resolution of background-oriented schlieren technique by introducing a telecentric optical system and its application to supersonic flow. Exp. Fluids Vol. 56, 2015

ULTRASOND VECTOR PROJECTILE IMAGING OF URINARY FLOW DYNAMICS

Takuro Ishii

Frontier Research Institute for Interdisciplinary Sciences, Tohoku University
6-3 Aramaki Aza Aoba, Aoba ward, Sendai, Miyagi 980-8578 Japan
takuro.ishii@tohoku.ac.jp

ABSTRACT

Lower Urinary Tract Symptoms (LUTS) is a major age-related disease caused by changes in mechanical and physiological properties of the bladder and the urethra, such as benign prostatic hyperplasia (BPH). Yet, it is difficult to directly observe the shape of the urinary tract and their hydraulic effects to the internal urinary flow because the present in-vivo flow imaging modality (e.g., color Doppler imaging) does not have a sufficient time resolution to image the large deformation of the urinary tract in the beginning and ending phases of the voiding or is not capable of quantitatively measuring the multi-directional flow in the curved and obstructed urinary flow tract. Thus, this study aimed to develop a novel urodynamic flow imaging modality based upon an ultrasound high-frame rate vector flow imaging technique, Vector Projectile Imaging (VPI) [1].

The urodynamic flow imaging system was built using a research-purpose ultrasound platform (Vantage 128, Verosnics, Inc., WA, USA) equipped with a custom-made transrectal linear probe (L10-5TR192V, Okusonic corp., Saitama, Japan). In addition, an original ultrasound imaging framework, Uro-VPI [2], was implemented on the system. In general, the ultrasound platform was configured to emit unfocused ultrasound pulses from -10 and 10 degrees of transmit steering angles alternatively to acquire Doppler shift maps of respective beam direction, and those multiple Doppler data was processed to visualize time-resolved flow vectors that depicted the spatio-temporal distribution of flow direction and speed. The efficacy of the developed system was tested in male patients with lower urinary tract symptoms.

The experiments showed the Uro-VPI system was capable of clearly visualizing the transient and complex urinary flow dynamics, such as formation of jet and vortex flow patterns at the obstruction in the urethra. This new flow visualization modality in urology would provide deeper insights into the hydrodynamic mechanisms causing LUTS.

REFERENCES

[1] Yiu, B., Lai, S., Yu, A., “Vector Projectile Imaging: Time-resolved Dynamic Visualization of Complex Flow Patterns”, Ultrasound Med Biol, Vol. 40 (2014), pp. 2295-2309.

[2] Ishii, T., Nahas, H., Suzuki, K., Yiu, B., Chee, A., and Yu, A., “Contrast-Enhanced Urodynamic Vector Projectile Imaging (CE-UroVPI) for Urethral Voiding Visualization: Principles and Phantom Studies”, Urology, Vol. 140 (2020), pp. 171-177.

COMPARISON OF STRAIGHT-SWIMMING PERFORMANCE OF PITCHING WING BETWEEN STEADY AND PERIODIC FLOWS

*Y. Isoda[1], Y. Tanaka[2]

[1] Graduate School of Science and Technology, Kyoto Institute of Technology, Kyoto-shi, Kyoto-fu 606-8585 Japan

[2] Faculty of Mechanical Engineering, Kyoto Institute of Technology, Kyoto-shi, Kyoto-fu 606-8585 Japan

*d1821003@edu.kit.ac.jp

ABSTRACT

In this study, we investigated the difference in the straight-swimming performance of the pitching wing between steady and periodic flows, where the periodic flow is a time-varying freestream velocity in the mainstream direction, and the pitching wing is a NACA 0012 airfoil rotating about its quarter chord point. The swimming performance is evaluated using mean thrust and lift coefficients. For instance, the pitching wing can swim straight in the forward direction at positive thrust and the zero lift coefficient. The thrust and lift coefficients were calculated by open-source computational fluid dynamics (CFD) software OpenFOAM. The CFD was carried out under the conditions of Reynolds number of 4130. The Strouhal number of periodic flow, St_p, is 0.5, and that of the pitching wing, St_w, is 0.5, 1, 1.5, and 2. The most important parameter is the frequency ratio, k (= St_w/St_p), between the pitching wing and periodic flow. In present conditions, the frequency ratio, k, becomes 1, 2, 3, and 4. We consider the steady flow as a special case of periodic flow at a velocity amplitude of zero.

In steady flow, the thrust coefficient increases with increasing frequency ratio, k, and the lift coefficient are 0.00, 0.00, 0.00, and 0.89 at each frequency ratio, k. At k = 4 (or St_w = 2), the lift coefficient switches from zero to 0.89, and the wake becomes asymmetry. This phenomenon is called wake deflection [1]. In periodic flow, the thrust tendency to frequency ratio is the same as the steady case. The lift coefficient is 0.32, 0.00, 0.00, and 0.80 at each frequency ratio. At k = 4 (or St_w = 2), the lift coefficient is non-zero (0.80). That is the same as the steady case. However, lift coefficient is also non-zero (0.32) at k = 1. In conclusion, we found that the pitching wing in steady flow can swim straight at frequency ratio of 1, 2, and 3. In contrast, the pitching wing in a periodic flow can swim straight at frequency ratio of 2 and 3.

REFERENCE

[1] Yu, M., Hu, H., and Wang, Z., “Experimental and Numerical Investigations on the Asymmetric Wake Vortex Structures of an Oscillating Airfoil”, In 50th AIAA aerospace sciences meeting including the new horizons forum and aerospace exposition, (2012), p. 299.

INFLUENCE ON THERMAL IMPEDANCE ALONG HEAT TRANSFER PATH OF POWER SEMICONDUCTOR PACKAGE BY COPPER PAD ON PRINTED CIRCUIT BOARD

*K. Nishi[1]

[1] Ashikaga University, 268-1 Omae-cho, Ashikaga, Tochigi 326-8558 Japan
*nishi.koji@g.ashikaga.ac.jp

ABSTRACT

Power electronics technologies are becoming more important than before with motor application expansion. Motor drive system with three-phase inverter circuit is becoming common to meet high controllability and lower power consumption demands. Especially, adequate thermal design is vital to enable inverter integrated motor drive system with light weight and small size. To realize adequate thermal design, heat transfer structure of the system is required to be grasped and its visualization technique is important.

There is one way to visualize heat transfer structure of the system by utilizing thermal impedance distribution. In previous research [1], a methodology to obtain thermal impedance distribution along heat transfer path is proposed. Transient junction temperature is measured by utilizing temperature-sensitive parameter and thermal impedance distribution is obtained by optimizing thermal resistance and heat capacitance parameters of Cauer thermal network which minimize junction temperatura prediction error.

In this paper, this methodology is applied to two test boards with power semiconductor devices which employ a type of surface mount type package, “DPAK”. Each test board has different copper pad configuration and the influence on thermal impedance by copper pad configuration is discussed.

REFERENCES

[1] Nishi, K., “A Methodology to Extract Thermal Impedance Distribution along Whole Heat Transfer Path”, Journal of The Japan Institute of Electronics Packaging, Vol. 24, No. 6 (2021), pp. 551-559 (In Japanese).

EXPERIMENTAL STUDY OF THERMAL CONVECTION BY HEATING AND COOLING THE SIDE OR TOP SURFACE OF A ROTATING CYLINDRICAL VESSEL

K. Shimoura[1], *T. Ihara[1], T. Noguchi[1] and K. Hirata[1]

[1] Doshisha University, 1-3 Tatara-miyakodani, Kyotanabe-shi, Kyoto 610-0321, Japan
*cguc4017@mail4.doshisha.ac.jp

ABSTRACT

The objective of this study is to experimentally investigate a rotating flow field in a rotating cylindrical vessel. Among the secondary flows from the most basic rigid-body rotational motion generated in the rotating fluid in the cylindrical vessel, one has the columnar structure that appears from the water surface to the bottom we suppose that the former secondary flow could be related with the cooling of the top surface, and the latter secondary flow could be related with the heating and cooling of the side wall, and another has the vortex structure that appears just near the side wall.

As well as by Fowlis & Hide (1965) and Hide & Mason (1970), we made a regime diagram with two dimensionless quantities such as the thermal Rossby number Ro_T and the Taylor number Ta, observing visualized flow using Aluminum flakes. We also check whether the columnar flow structure is thermal convection or not, and investigate methods to prevent the generation of the two secondary flows.

We summarized the experimental results with various vertical temperature differences in a map of depth versus rotation speed, comparing with a transformed equation of the approximate equation for the critical Rayleigh number by Chandrasekhar (1953). We summarized the experimental results with various horizontal temperature differences in a regime diagram of Ro_T against Ta, comparing with the experimental results by Fowlis & Hide (1965) and Hide & Mason (1970).

As a result, it was found that the columnar structure is related with the thermal convection due to vertical temperature difference caused by vapor cooling from the top water surface, and that the occurrence of the columnar structure could be suppressed by placing the lid on the top water surface in the cylinder. It was also found that the vortex structure just near the side wall in one of several regimes is related with thermal convection due to the horizontal temperature difference. The thermal convection due to the horizontal temperature difference may be in different regimes due to heating or cooling of the side wall, and the temperature difference must be controlled by heating and cooling of the side wall below 0.05 K except for the condition where Ta is more than 10^8.

REFERENCES

[1] Fowlis, W. W. and Hide, R., Thermal convection in rotating annulus of liquid: effect of viscosity on the transition between axisymmetric and non-axisymmetric flow regimes, Journal of the Atmospheric Sciences, Vol. 22, No. 5 (1965), pp. 541–558.

[2] Hide, R. and Mason, P., Baroclinic waves in a rotating fluid subject to internal heating, Philosophical Transactions of the Royal Society of London, Series A, Vol. 268 (1970), pp. 201–232.

[3] Chandrasekhar, S., The instability of a layer of fluid heated below and subject to Coriolis forces, Proceedings of the Royal Society of London, Series A, Vol. 217 (1953), pp. 306–327.

STUDY ON DRAG COEFFICIENT FOR CIRCULAR PLATE WITH HOLES

*K. Namba[1], Y. Kubota[1] and O. Mochizuki[1]

[1] Toyo University, 2100 Kujirai, Kawagoe, Saitama 350-8585 Japan
*mangetu2015@gmail.com

ABSTRACT

The estimation of drag coefficient can be written as $C_D = 2D/(\rho u^2 A)$. Here, C_D is drag coefficient, D is drag force, ρ is density of fluid, u is flow velocity and A is the characteristic area. The drag coefficient is different with the shape of objects. Furthermore, the drag coefficient of annular plate as circular plate with hole is larger than the circular plate without hole [1]. The size of hole relates the magnitudes of drag coefficient. The previous study obtained the drag coefficient of circular plate with holes during the free-falling [2]. They reported that the drag coefficient increased with the increase of aperture ratio and the number of holes in a circular disk. Now we have a question, why does the drag coefficient of circular disk with holes increasing compared with circular plate without holes? We have carried out the drag force measurement and visualization under the steady condition to answer this question.

In our experiment, drag force measurement and flow visualization around the circular disk with holes were investigated. The experimental models were obtained from the number of holes and aperture ratio. Number of holes is 3, 4 and 6, and aperture ratio are 0.1, 0.2 and 0.3 respectively. Measurement of drag coefficient was carried out with towing in 4000x 500x 500mm water tank and momentum sensor.

The results show that the increase of both the number of holes and the aperture ratio showed the increase of drag coefficient. This result is agreed with previous result [2]. The results from flow visualization showed that the jet flow from the holes in a circular plate. This suggested the reason of increasing the drag coefficient of circular plate with holes. From the comparison of these results, the drag coefficient was increased with the summation of circumference of holes in model. This result suggests the effect of shear flow caused by the jet ejected from the holes in a circular disk. The details of these results will be shown in the presentation.

REFERENCES

[1] The Japan Society of Mechanical Engineering., Mechanical Engineers' Handbook Fundamentals, 1st Edition, Maruzen Publishing Co., Ltd, (2007), pp. α4-85.

[2] Naoki, H. and Yoshihiro, K., "Flow around circular plate with holes", The Japan Society of Mechanical Engineering Journal, (2020).

EFFECT OF SURFACE WETTABILITY FOR DISK FRCTION ON ROTATING DISK IN CENTRIFUGAL IMPELLER

*Kento Sakai [1], Yumeno Inaba[1] and Kazuyoshi Miyagawa[1]

[1] Waseda University, 3-4-1 Okubo, Shinjuku-ku, Tokyo, 169-8555, Japan
*kent2916@fuji.waseda.jp

ABSTRACT

The disk friction loss is a major issue for high efficiency for industrial pumps since the disk friction loss increases remarkably in the low specific-speed pump. Flow fields in narrow clearance between rotating disks and stationary disks are theoretically analysed [1]. Moreover, Watanabe et al. [2] developed a drag-reduction method using the spiral grooves on the disk surface. However, it has few examples of the practical application though various kinds of study.

This research focuses on the relationship between surface wettability, internal flow, and disk friction loss. The hypothesis that the effect of disk friction reduction could be obtained by controlling the wettability of the disk surface was tested by the experimental method.

The test disks were chemically coated with a thickness of about 3μm. The Ad-tech coating and the Cytop coating were selected for hydrophilic coating and for hydrophobic coating, respectively. The measured contact angles were about 14° for the hydrophilic coating and about 110° for the hydrophobic coating. Using the rotating disk with these coating and without coating, the torque and pressure measurement test was performed, and the results are compared. The closed-loop experimental apparatus simulates the clearance flow between a rotating impeller and a stationary casing as such in centrifugal pumps. Measurements of friction torque and pressure distribution in radial direction were conducted in various testing conditions changing rotational speed, flow rate and water temperature. In addition, the flow field was simulated by solving the unsteady RANS equations by three-dimensional computational fluid dynamics simulations to compare with the experimental results. The Shear Stress Transport k-omega model was used to include the effects of turbulence model.

Experimental results show radial pressure distribution are almost same between the surface with hydrophilic/hydrophobic coating and without coating. These results suggest that the flow field in the clearance are nearly unchanged depending on the surface wettability. On the other hand, when we organized the disk friction coefficient calculated from the measured torque by the rotating Reynolds number, the disk friction coefficient with coating is smaller than without coating, regardless of the hydrophilic/hydrophobic effects. It should be additionally noted that effects of coating for friction reduction is greater when the Reynolds number is smaller.

In conclusion, our experimental results provide the evidence for the reduction of the disk friction loss by controlling the wettability of the surface.

REFERENCES

[1] Daily, J.W. and Nece, R.E.,“Chamber Dimension Effects on Induced Flow and Frictional Resistance Enclosed Rotating Disks”, ASME Journal of Fluid Engineering, Vol.82-1(1960), pp.217-230.

[2] Watanabe, K., Budiarso, Ogata, S. and Uemura, K., “Drag Reduction of an Enclosed Rotating Disk with Fine Spiral Grooves”, Journal of Environment and Engineering, Vol.2(2007), No.1, pp.97-107.

VISUALIZATION OF SHOCK WAVES ABOVE WATER DRIVEN BY WATER ENTRY USING SHADOW GRAPH METHOD

*Y. Ebii[1], T. Kikuchi[1] and A. Muramatsu[1]

[1] Nihon University, 7-24-1 Narashinodai, Funabashi, Chiba 274-8501, Japan
*csys21009@g.nihon-u.ac.jp

ABSTRACT

The high-speed water entry phenomenon is a complicated unsteady flow, which involves a water splash, an underwater shock wave, an underwater projectile, and an underwater cavity. We are interested in physics of this complicated phenomena. A shock wave in air above water also occurs even when the entry speed is less than the speed of sound in the air[1]. We guess that this shock wave affects phenomena above water phenomena and causes a threshold value between low-speed and high-speed. In order to confirm the entry speed at which a shock wave above water begun to be generated, 5.97 mm diameter plastic resin spheres were launched at different entry speeds. The flow fields above water were optically visualized by the shadow graph method. Although a week shock wave above water was observed at the entry speed 55.19 m/s, this shock wave attenuated and disappeared on next video frame. The propagation velocity of the shock waves above water at entry speeds of 69.15-152.8 m/s were calculated from the resulting sequence photographs. The shock propagation velocities were almost constant at less than 130 m/s of the projectile entry speed. The shock propagation velocities increased linearly with respect to the entry speed at greater than 130 m/s of the entry speed. Therefore, the influence of shock waves above water on water-surface phenomena may not be negligible at entry speeds of 130 m/s or higher. The entry point was enlarged and optically visualized the inception of shock waves above water by the same way. Some initial shock waves were attached to the water splash at entry speeds of 125-152.8 m/s. It was guessed that the initial speed of the water splash exceeded the speed of sound in air.

REFERENCES

[1] Kikuchi, T. and Takayama, K. The influence of the entry speed to flow field above water surface on the water entry phenomenon, International Journal of scientific and technical research in engineering, Vol 2, Issue 12, 2017.

DEVELOPMENT OF HIGH-RESOLUTION PSV USING CODED ILLUMINATION AND FEATURE MATCHING ALGORITHM

*Yusaku Tsukamoto[1], Shumpei Funatani[1]

[1] University of Yamanashi, 4-3-11 Takeda, Kofu, Yamanashi 400-8511, Japan
*g21dts03@yamanashi.ac.jp

ABSTRACT

PSV is a high-resolution flow velocimetry method using a digital SLR camera, and its spatial dynamic range is one order of magnitude higher than that of PIV. Several recent studies proposed new PSV methods to measure flow fields that cannot be measured by PIV or PTV. Wang et al. developed a new measurement technique named CSPSV using a color illumination system and applied it to a large-scale 3D measurement (1.5 m × 1.4 m × 0.8 m in width × height × depth) using two digital SLR cameras [1].

We developed an analysis method to detect particle trajectories using a feature matching algorithm to improve the resolution of PSV. Feature matching algorithms are well known as one of the image recognition techniques, and this study used the KAZE algorithm [2] to find key points and matched them by the *k*-nearest neighbor algorithm (*k*-NN algorithm). In addition, the direction of particle flow was recorded in the visualized image by using the coded illumination method proposed by Khalighi et al [3].

In this study, we first verified the validity of the trajectory detection methods using the KAZE and *k*-NN algorithms in advance by using synthetic images that artificially simulate the trajectory of particles captured by the coded illumination method. In the next validation, the applicability of the KAZE and *k*-NN algorithms to actual airflow measurements was evaluated using a device that generates a swirling airflow. The optical system consists of a semiconductor laser (continuous wave laser, 532 nm, 1 W) and a high- resolution digital SLR camera (image size: 6000 × 4000 pixels), and the illumination timing of the laser is controlled by a TTL signal.

The validation results show that the coded illumination method can successfully visualize asymmetric particle trajectories from swirling flows, and that the KAZE and *k*-NN algorithms can detect trajectories in the visualized images.

REFERENCES

[1] Wang, H., Li, X., Shao, X., Wang, B., and Lin, Y. “A colour-sequence enhanced particle streak velocimetry method for air flow measurement in a ventilated space.”, Building and Environment, Vol. 112, No. 1 (2017), pp. 77-87.

[2] Alcantarilla, P.F., Bartoli, A., and Davison, A.J. “KAZE features.”, In: 12th European conference on computer vision, Italy, (2012), pp. 214-227.

[3] Khalighi, B., and Lee, Y.H. “Particle tracking velocimetry: an automatic image processing algorithm.”, Applied Optics, Vol. 28, No. 20, (1989), pp. 4328-4332.

DETECTION OF LARGE-SCALE STRUCTURES IN A TEMPORAL ROUND JET USING MULTIDIMENSIONAL DYNAMIC MODE DECOMPOSITION

*M. Takahashi[1], R. Fukui[1], K. Tsujimoto[1], T. Ando[1] and T. Shakouchi[1]

[1] Mie University, 1577 Kurimamachiya-cho, Tsu-shi, Mie 514-8507 Japan
*takahashi@mach.mie-u.ac.jp

ABSTRACT

Numerical simulations reproducing the turbulent flows generated in the laboratory using wind tunnels or water channels are called spatially-developing simulations, which often require large computational domains and long computation times for convergence of statistics. In this respect, temporally-developing simulations can save computational resources and have been used in many studies of shear flows.

Round jet in a temporally-developing simulation (temporal round jet) obtains the streamwise homogeneity in exchange for temporal stationarity. In a cylindrical coordinate system, the temporal round jet is not only homogeneous but also periodic in both streamwise and azimuthal directions. Thus, the flow field of the temporal round jet is expected to be decomposed into spatially oscillating modes. Dynamic mode decomposition (DMD) provides the modal decomposition according to the temporal or spatial oscillation. In recent years, DMD has been further generalized for multidimensional oscillation analysis, which is originally documented as Spatio-temporal Koopman decomposition[1]. This enables us to analyze the dominating structures in a temporal round jet based on the streamwise and azimuthal oscillations.

It is considered that the helical large-scale structures are dominant in the far-field of round jets[2]. In this study, we use the multidimensional DMD to detect the dominant large-scale structures in a temporal round jet. The jet is generated in a cubic computational domain whose length in all three directions is $4\pi D_E$, where D_E is the initial diameter of the jet core. The jet Reynolds number is 2500. Consequently, we demonstrate that the large-scale structure in the present temporal round jet consists of single- and double-helical structures, which is in good agreement with that supposed by the existing experimental results.

REFERENCES

[1] Le Clainche, S. amd Vega, J. M., “Spatio-Temporal Koopman Decomposition”, Journal of Nonlinear Science, Vol. 28, No. 5 (2018), pp. 1793-1842.

[2] Samie, M., Lavoie, P. and Pollard, A., “Quantifying eddy structures and very-large-scale motions in turbulent round jets”, Journal of Fluid Mechanics, Vol. 916, No. 10 (2021), A2.

HEAD-ON VORTEX RING COLLISIONS UPON SMALL LATTICE POSTS

*Daniel Tze How New[1], Chuanhua Liu[2], Darius Koi Yik Tham[2], Bowen Xu[1]and Haotian Li[3]

[1] School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, 639798, Singapore
[2] NUS High School of Mathematics and Science, 20 Clementi Avenue 1, 129957, Singapore
[3] VOMMA (Shanghai) Technology Co. Ltd, Nerobay 1, 951 Jianchuan Road, Shanghai, China

ABSTRACT

An experimental time-resolved particle-image velocimetry study had been conducted in water on round vortex rings colliding head-on with surfaces with lattice posts. The vortex rings were generated by an impulsively started piston driven by a stepper motor, where "cylindrical water slugs" were ejected out of a d=20mm round nozzle. The Reynolds numbers of the vortex rings range from $Re=Ud/\nu=1000$ to 4000, where U is the mean piston velocity and ν is the kinematic viscosity of water. The lattice posts comprised of circular cylinder with diameters of 1mm and 2.5mm, as well as spaced at 5mm and 10mm apart in a square grid. The research goal is to study how effective these lattice posts and their different arrangements are in terms of enhancing the breaking up of the primary vortex rings and mitigating the formations of secondary and tertiary vortex rings. Earlier study by [1] have shown that it is possible to control head-on collisions between vortex rings and flat walls via lattice posts and thus motivated the present study. Findings from the study will have relevance in better understanding how strong counter-rotating vortices, such as those produced by aircraft wing-tips, may be encouraged to dissipate faster through the use of similar physical structures strategically located by the sides of airport take-off runways. Results from the study reveal that the lattice posts are able to dissipate the various vortex ring structures faster than when no lattice posts are present (see [2-4] for latter), with the dissipation rate/behaviour depending on the post diameter and spacing.

References

[1] Li, Q., & Bruecker, C. H. (2018). Vortex interaction with a rough wall formed by a hexagonal lattice of posts. Physics of Fluids, 30(5), 054107.

[2] Verzicco, R., & Orlandi, P. (1994). Normal and oblique collisions of a vortex ring with a wall. Meccanica, 29(4), 383-391.

[3] Cheng, M., Lou, J., & Luo, L. S. (2010). Numerical study of a vortex ring impacting a flat wall. Journal of Fluid Mechanics, 660, 430-455.

[4] New, T. H., Shi, S., & Zang, B. (2016). Some observations on vortex-ring collisions upon inclined surfaces. Experiments in Fluids, 57(6), 1-18.

ADAPTIVE WINDOW TECHNIQUE FOR LIFETIME-BASED TEMPERATURE AND VELOCITY SIMULTANEOUS MEASUREMENT USING THERMOGRAPHIC PARTICLE TRACKING VELOCIMETRY WITH A SINGLE CAMERA

*T. Cai, J. Han, M. Kim, J. Jung and †K.C. Kim

School of Mechanical Engineering, Pusan National University, Busan, 46241, Korea.
†kckim@pusan.ac.kr

ABSTRACT

The simultaneous measurement of temperature and velocity in the fluid has been attracted increasing interest as it could provide synchronized temperature and velocity information, which is important in the basic research on fluid mechanics and thermodynamics. As one of the effective velocity and temperature simultaneous measurement methods, thermographic PIV/PTV[1] has experienced rapid development in recent years as its outstanding capability in various application areas, wide temperature range, and in-situ measurement.

The most widely used thermographic PIV/PTV method is the intensity-ratio-based method[2]. Although this method is relatively mature and has been applied in film cooling, combustion, and other applications, it still has many obvious shortcomings including 1) susceptible to surrounding light such as multiple scatters and near-wall reflections, 2) its expensive solution in which several lasers and cameras are needed, 3) cumbersome process: aligning several laser sheets will be difficult, and since two different viewing angles are used, a correction is needed.

Compared with the intensity ratio method, the lifetime method[3] has been proven to have higher temperature measurement accuracy in the field of surface temperature measurement because of its self-referencing characteristics. In addition, the single camera-based measurement could reduce the cost and cumbersome arrangement to a certain extent. With the continuous development of high-precision measurement requirements, it is necessary to develop a lifetime-based thermographic PTV/PIV. It will greatly reduce the difficulty of the measurement setup and the cost of equipment while improving the measurement accuracy.

In this study, we proposed an adaptive window technique for lifetime-based temperature and velocity simultaneous measurement using thermographic PTV with a single camera. This method generates windows for calculating the phosphorescence decay lifetime constant by capturing the position and velocity of phosphorescent particles. It overcomes the problem of inaccurate temperature acquisition of dynamic particles in the traditional lifetime method. Despite the disadvantages of sacrificing resolution and adding post-processing steps, the high-precision measurement results, single-camera realization, and application of moving targets make it an alternative method for thermographic PTV measurement.

REFERENCES

[1] Abram. C., Fond. B., Beyrau. F., Temperature measurement techniques for gas and liquid flows using thermographic phosphor tracer particles, Progress in energy and combustion science, 64(2018) 93-156.

[2] Omrane. A., Juhlin. G., Ossler. F., and Aldén. M., Temperature measurements of single droplets by use of laser-induced phosphorescence, Appl Opt, 43(2004) 3523-9.

[3] Someya. S., Uchida, M., Tominaga, K., Terunuma, H., Li. Y., and Okamoto. K., Lifetime-based phosphor thermometry of an optical engine using a high-speed CMOS camera, International journal of heat and mass transfer, 54(2011) 3927-32.

EXPERIMENTAL INVESTIGATION ON THE FLOW STRUCTURE OF BUOYANCY VORTICES USING PARTICLE IMAGE VELOCIMETRY

*Z. Wang[1], N.A. Hawkes[1], M. MacDonald[1], J.E. Cater[2] and R.G.J. Flay[1]

[1] Department of Mechanical and Mechatronics Engineering, University of Auckland, 1010, New Zealand
*zwan472@aucklanduni.ac.nz
[2]Departement of Engineering Science, University of Auckland, 1010, New Zealand

ABSTRACT

A buoyancy vortex is similar to many naturally occurring vortices, such as dust devils. For dust devils, the source of buoyancy is from intense surface heating from solar radiation that can result in strong winds with peak tangential velocities of approximately 10 m/s near ground level [1, 2]. Field measurements of atmospheric vortices have been complicated by unsteady temperature gradients and unpredictable winds. To obtain a better understanding of the flow structure of a buoyancy vortex, a laboratory investigation in a controlled environment is required without the influence of ambient conditions.

One of the earliest laboratory buoyancy vortex studies was by Mullen and Maxworthy [3], where the flow was visualised and measured using soap bubbles. However, the accuracy of the early measurements and the flow visualisation near the vortex core were not ideal due to the short survival time of the tracer particles and the limitations of the recording system. Therefore, this study is intended to create a more holistic understanding of the flow characteristics around the vortex core, the potential flows at the outer region, and the temperature-driven flow structures using the particle image velocimetry (PIV) technique.

This research carried out a quantitative analysis to explore an artificial buoyancy-driven columnar vortex at a laboratory scale. Based on experimental investigations using flow visualisation [4] and PIV, the horizontal cross-section of the vortex at 0.6 m above the ground was captured. The formation and decay stages of the vortex were visualised, and both instantaneous and time-averaged tangential and radial velocity profiles were derived from the PIV results. The temperature profiles were also measured from thermocouples mounted at the same height. A correlation was investigated between the temperature profile in the radial direction of the vortex and the location of the peak velocity.

REFERENCES

1. Sinclair, P.C., Ph.D. Thesis, *A quantitative analysis of the dust devil*. 1966, The University of Arizona.
2. Sinclair, P.C., *The Lower Structure of Dust Devils.* Journal of the Atmospheric Sciences, 1973. **30**(8): p. 1599-1619.
3. Mullen, J. and T. Maxworthy, *A laboratory model of dust devil vortices.* Dynamics of Atmospheres and Oceans, 1977. **1**: p. 181-214.
4. Hawkes, N. & Flay, R. & Cater, J. & MacDonald, M., *Scaling of Experimental Buoyancy Vortex Structures with Respect to Power Generation*, 2020. Journal of Physics: Conference Series

FREQUENCY RESPONSE OF CARBON-NANOTUBE TEMPERATURE-SENSITIVE PAINT

*T. Ikami[1, 2], K. Takahashi[1], Y. Konishi[1] and H. Nagai[1]

[1]Institute of Fluid Science, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi-prefecture 980-8577, Japan
*tsubasa.ikami.s7@dc.tohoku.ac.jp
[2]Department of Aerospace Engineering, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi-prefecture 980-8577, Japan

ABSTRACT

Temperature-sensitive paint (TSP) is a flow measurement technique that optically measures global temperature distribution. TSP allows visualization of flow fields by measuring the temperature difference caused by laminar and turbulent flows. The turbulent flow has a higher heat transfer coefficient than the laminar flow. Therefore, the temperature distribution is formed due to the heat transfer between the test model and flow when there is a temperature difference between them.

Recently, cntTSP has attracted much attention. cntTSP is a combination of TSP and a carbon nanotube (CNT) layer for the thin internal heater. The CNT layer supplies constant heat to the model surface and a temperature difference occurs between the model and flow when a voltage is applied to the CNT layer. As a result, the laminar-to-turbulent boundary layer transition can be measured as the temperature difference. In previous studies, cntTSP has been applied to unsteady flow field measurements in models in motion, such as pitch oscillations. However, the quantitative response of cntTSP has not yet been evaluated to apply to unsteady flow fields.

In this study, the frequency response of cntTSP was quantitatively evaluated by making changes in the flow field by periodically introducing local disturbances to the boundary layer on a flat plate using a loudspeaker. The turbulent wedge caused by the introduction of the local disturbance was visualized as a low-temperature region, which indicates a large heat transfer coefficient. Moreover, the temperature in the wedge-shaped region varied according to the period of introduction of the local disturbance. Furthermore, the amplitude of the temperature fluctuation was quantitatively evaluated. The temperature amplitude decreased with the increase in the frequency of the introduction of the local disturbance, and it reached $O(10^{-3}$ K) at 10 Hz.

REFERENCES

[1] Ikami, T., Fujita, K., Nagai, H. and Yorita, D., “Measurement of Boundary Layer Transition on Oscillating Airfoil using cntTSP in Low-speed Wind Tunnel”, Measurement of Science and Technology, 32 (2021), 075301, 11pp.

[2] Simon, B., Filius, A., Tropea, C. and Grundmann, S., “IR Thermography for Dynamic Detection of Laminar-turbulent Transition”, Experiments in Fluids, 57:93 (2016).

LOCAL AND TRANSIENT HEAT TRANSFER MEASUREMENT FOR DROPWISE CONDENSATION USING TEMPERATURE SENSITIVE PAINTS

*K. Matsushima[1], T. Uchimura[1], N. Sakoda[1,2] and Y. Kita[1,2]

[1] Department of Mechanical Engineering, Kyushu University, 819-1395 Japan
* k.matsushima@heat.mech.kyushu-u.ac.jp
[2]International Institute for Carbon-Neutral Energy Research (WPI-I^{2}CNER), Kyushu University, Japan

ABSTRACT

Condensation is a phase-change phenomenon in which vapor turns into liquid and used in many applications such as electronic cooling, air conditioners and thermal power plants. On a solid surface, condensation can occur in two modes: filmwise condensation, in which a liquid film is constantly present, and dropwise condensation, in which the condensate liquid collects in the form of droplets. Dropwise condensation has been of considerable interest due to higher heat transfer coefficients than those of filmwise condensation. Although research is underway for the industrial application of dropwise condensation, numerous researchers have focused on droplet evolution and shapes, departure mechanisms and jumping phenomena without completing the task of actually evaluating local and transient heat transfer phenomena which should come along with the droplet behavior [1]. Understanding of such microscopic and transient nature of dropwise condensation heat transfer is needed for the development of a comprehensive predictive model.

In this contribution, we develop a technique capable of simultaneously measuring temperature distributions on a heat transfer surface and monitoring droplet behavior using TSPs (Temperature-Sensitive Paints). TSPs are fluorescent paints that are excited by UV light and emit light with a longer wavelength. The temperature can be determined based on the intensity of the emitted light. As it can measure through materials that are opaque to IR with high spatial and temporal resolution at low cost [2], TSP is more suitable for condensation experiment than other methods such as IR cameras or thermocouples.

In our experiments, a thin titanium plate was used as the condensing surface and attached to a sapphire substrate using an adhesive. Two TSP layers were created across the adhesive, and the local heat flux was calculated using the temperatures measured at those two layers. As a result, we have succeeded in capturing the TSP brightness changes coupled with droplet motion with high spatial resolution. We have demonstrated the potential of TSP-based local heat transfer measurement for condensation research.

REFERENCES

[1] El Fil, B., Kini, G. and Garimella, S., “A review of dropwise condensation: Theory, modeling, experiments, and applications”, Int. J. Heat Mass Transfer, Vol. 160 (2020), 120172.

[2] Al Hashimi, H., Hammer, C.F., Lebon, T.M., Zhang, D. and Kim, J., “Phase-Change Heat Transfer Measurements Using Temperature-Sensitive Paints”, Journal of Heat Transfer, Vol. 140, No. 3 (2018), 031601.

VIDEO ANALYSIS ABOUT THE GLIMMER SYNCHRONIZATION OF *LUCIOLA PARVULA* FIREFLIES

*Shiro Kobayashi[1], Nao Ninomiya[1], Masayuki Iigo[2] and Rion Tada[1]

[1] Deptartment of Optical Engineering, Utsunomiya University, 7-1-2 Yoto, Utsunomiya 321-8585 Japan
*nino@utmu.jp
[2]Center for Bioscience Research and Education, Utsunomiya University, 350 Mine, Utsunomiya 321-8505 Japan

ABSTRACT

It is known that the *Luciola parvula* fireflies clearly emit lights at regular time intervals and synchronize between multiple males are observed, but the details of synchronization have not yet been clarified. Presently, we took videos of the luminescence phenomenon of multiple male *Luciola parvula* fireflies and performed a correlation analysis on the details of the time change of the luminescence brightness of each male *Luciola parvula* fireflies. The 12 *Luciola parvula* fireflies are kept in glass wells so that each *Luciola parvula* fireflies can only moves in restricted areas and their luminescence is recorded by a digital movie camera from above at the video rate of 30 fps. According to the preliminary study, each glimmer lasts about 3 frames and thus the time series brightness is fitted by a parabolic function in order to give a precise timing of maximum brightness. First of all, time intervals between glimmers of each *Luciola parvula* fireflies are evaluated. Secondly, the glimmer correlation between different *Luciola parvula* fireflies are investigated. As a result, it was clarified that the light emission cycle of *Luciola parvula* fireflies is almost constant at 0.54 seconds, more than 70% of all light emissions are synchronized and the order of synchronization is almost fixed with very small delay. Consequently, it has been shown quantitatively that the glimmers of male *Luciola parvula* fireflies are well synchronized. The synchronization of fewer individuals and the synchronization to the external stimulus and many other aspects remain for the future study.

REFERENCES

[1] Nakagawa, M. and Tanaka, H., “A Plausible Model of Synchrony in *Pteropyx effulgens* and Its Occurrence Condition for Traveling Waves”, Journal of the Japan Society for Industrial and Applied Mathematics, Vol. 29, No. 3 (2019), pp. 250-293.

[2] Hanson, F. E., Case, J. F, Buck, E. and Buck, J., “Synchrony and flash entrainment in a New Guinea firefly”, Science, 174 (1971), 161–164.

TIME-RESOLVED PIV MEASUREMENTS OF FLOW FIELD AROUND A BADMINTON SHUTTLECOCK DURING TURNOVER PROCESS

*Y. Sakurai[1], K. Nakagawa[1] and H. Hasegawa[1]

[1] Utsunomiya University, 7-1-2 Yoto, Utsunomiya-city, Tochigi 321-8585 Japan
*mc216725@cc.utsunomiya-u.ac.jp

ABSTRACT

Badminton is one of the most popular sports in the world. The shuttlecock is used in badminton game has the unique shape. The shuttlecock is truncated cone-shaped and consists of a cork, gaps and a skirt portion. The shuttlecock has aerodynamic properties which differ from the ball used in other racquet sports. As an example of unique aerodynamic property, the shuttlecock shows high deceleration. It is known that the initial velocity immediately after smashing may reach up to 137m/s (493 km/h) at maximum. The mechanism of high deceleration of the shuttlecock was clarified in a previous study [1]. In addition, turnover refers to the flipping experienced by a shuttlecock when undergoing heading change from nose pointing against the flight path at the moment of impact and a shuttlecock indicates the aerodynamically stable feature for the flip movement just after impact [2]. In a previous study, the flow field around the shuttlecock during turnover process was visualized [3]. However, the mechanism of turnover stability of the shuttlecock has not been fully understood.

In this study, unsteady flow field during turnover process was investigated to clarify the mechanism of turnover stability. In the present, we simulated the flipping motion by wind tunnel experiments and visualized the flow field around the shuttlecock by a PIV technique.

It was confirmed that the flipping time in turnover process for the standard shuttlecock (with gaps) was shorter than that for the shuttlecock without gaps [3]. In the standard shuttlecock, the flow through the gap produces a vortex behind the shuttlecock, which increases the pressure difference between the front and back of the shuttlecock and generates a restoring force. This restoring force relates to the turnover stability.

REFERENCES

[1] Hasegawa, H., Kitta, S., and Obayashi, S., “Flow analysis and aerodynamic characteristics of a badminton shuttlecock with spin at high Reynolds numbers”, Sports Engineering, Vol.16 (2011), 91-98.

[2] Cohen C., Texier B.D, Quere D., and Clanet C., “The physics of badminton”, New Journal of Physics, Vol.17 (1989), 063001.

[3] Nakagawa K., Hasegawa H., Murakami M., and Obayashi S., “Aerodynamic stability of a badminton shuttlecock”, Transactions of JSME (in Japanese), Vol.83(2017), No856.

UNSUPERVISED DEEP LEARNING OF THE STRUCTURAL CHARACTERISTICS IN A TURBULENT CHANNEL FLOW AT A LOW REYNOLDS NUMBER

*Mohammad Javad Sayyari[1], Jinyul Hwang[1], Kyung Chun Kim[1]

[1] School of Mechanical Engineering, Pusan National University, Busan 46241, Republic of Korea
*sayyari@pusan.ac.kr

ABSTRACT

In this study, we examine the capability of an unsupervised deep learning network to capture the spatial organization of large-scale structures in a turbulent channel flow at a low friction Reynolds number. For this purpose, the Wasserstein generative adversarial network with gradient penalty (called GAN here for simplicity) is trained based on the cross-stream dataset of a direct numerical simulation (DNS) at the friction Reynolds number $Re_\tau \approx 180$ to generate artificial flow fields. Then, we explore the turbulence statistics and the corresponding coherent structures using the generated flow fields. The generated flow fields show quite similar one-point and two-point statistics when compared with those obtained from the DNS data. Further analysis reveals that the GAN learns important structural information hidden in the training data, such as the length-scale of the energetic motions, the spanwise distance of the low-speed streaks in the logarithmic region, and the wall-normal variation of the integral length scales. The success of GAN to predict these features inspires us to evaluate its capability to learn the spatial organization of the coherent structures. In this regard, we measure the size distribution of all the intense velocity structures identified in the instantaneous flow fields and find a very good agreement between the DNS and GAN results. Then, we focus our study on the large-scale structures and investigate their spanwise waviness and inclination angle with respect to the horizontal axis as the cross-stream imprints of their large-scale meandering motion in the streamwise direction. The relations among the waviness, inclination angle, width and height of the structures are discussed by computing their joint probability density functions. The results show increasing waviness behaviour as the size of the structures increases. Also, it is observed that the structures are almost symmetrically distributed around 0° when they are averaged unconditionally. This is further analysed by conditionally averaging the identified structures, categorizing them into negative and positive clusters. The success of GAN to learn the structural characteristics hidden in the turbulent flow fields in the low Reynolds number suggests its potential to predict this information at a high Reynolds number and thus be utilized as an inflow turbulence generator to provide instantaneous boundary conditions for more complicated problems such as spatially evolving turbulent boundary layers. This has a great potential for reducing the computational costs of DNS related to a required large computational domain at high Reynolds numbers. However, further studies are required to assess the GAN capability for predicting the vortical structures associated with the coherent structures studied in this work.

VISUALIZATION AND MEASUREMENT OF INTERNAL FLOW IN A THIN VORTEX CHUCK RESEARCH ON IMPROVEMENT OF SILICON WAFER ADSORPTION PERFORMANCE

* Naoyuki Fukaya [1] , Shumpei Funatani[1], Katsuhiro Koshiishi[2] , Hitoshi Iwasaka[2]

[1] University of Yamanashi, 4-3-11 Takeda, Kofu, Yamanashi 400-8511, Japan
*g22tm022@yamanashi.ac.jp
[2]HARMOTEC CO., LTD., 4-1-32 Sumiyoshi, Kofu, Yamanashi 400-0851, Japan

ABSTRACT

Conventionally, silicon wafers and other semiconductor products have been transported by suction or gripping in direct contact. However, direct contact with a wafer adversely affects the quality and characteristics of semiconductor products because it causes contamination of the wafer surface by fine particles and generates static electricity. Therefore, nowadays, non-contact transfer equipment using a swirling flow (hereinafter referred to as a vortex chuck) is often used for transporting such precision semiconductors. In addition to the feature of non-contact work holding, vortex chucks have the advantages of not generating heat, not generating a magnetic field, and having a simple structure. However, the effects on air flow and workpieces have not been clarified for the thin vortex chucks that have been developed in recent years. In this study, turbulent flow during suction by a vortex chuck was measured visually, and the locations where turbulence occurs were evaluated.

A transparent acrylic plate was used as the work material for visualization. The air compressor is connected to the vortex chuck via a regulator, pressure gauge, and valve. The supply pressure was set at 0.2 MPa. The surface of the workpiece is uniformly coated with high-viscosity oil and sprinkled with titanium dioxide powder as tracer particles. The gap between the workpiece and the vortex chuck was maintained at approximately 0.2 mm using a micrometer to adjust the height of the workpiece.

Negative values tend to be observed near the wall of the vortex chuck, indicating that large turbulence is generated by the sudden change in vorticity. At one location, the vorticity was -0.007 /ms, which is larger than the vorticity at other locations, indicating that the pressure is concentrated. This suggests that the pressure distribution during suction by the vortex chuck is uneven, leading to a narrower suction range and reduced stability during suction retention.
As a result of visualization and measurement of turbulence during suction by the vortex chuck, it was confirmed that there were variations in turbulence energy due to the structure of the vortex chuck. In the future, we will study and fabricate a vortex chuck structure that improves the stability of silicon wafer adsorption and retention based on the conditions of turbulence generation.

REFERENCES

[1] Li, X. Tokunaga, H., Cai, M., Funaki, T., Kawashima, K., Kagawa, T. “Research on a non-contact handling system using swirling flow (1st Report: Basic characteristics of a vortex chuck)”, Trans JFPS, Vol. 38, No. 1 (2007), pp. 1-6 (Japanese).

DEVELOPMENT OF VISUALIZATION TECHNIQUE FOR HYDRODYNAMIC STRESS FIELD UTILIZING FLUORESCENT FORCE PROBE

*R. Kuriyama[1], R. Okamoto[2], W. Yamamoto[1], H. Kitakado[3], S. Saito[3], K. Tatsumi[1] and K. Nakabe[1]

[1] Graduate School of Engineering, Kyoto University, Kyotodaigaku-katsura, Nishikyo-ku, Kyoto 615-8540 Japan
* kuriyama.reiko.2m@kyoto-u.ac.jp
[2] Faculty of Engineering, Kyoto University, Kyotodaigaku-katsura, Nishikyo-ku, Kyoto 615-8540 Japan
[3] Graduate School of Science, Kyoto University, Kitashirakawa Oiwake, Sakyo-ku, Kyoto 606-8502 Japan

ABSTRACT

The present study investigates the fluorescence response of a molecular force probe called FLAP (flexible and aromatic photofunctional systems), which undergoes stress-induced conformational change in elongational flow. We aim at developing a technique for visualizing the hydrodynamic stress field in channel flow through fluorescence imaging of the FLAP force probe. FLAP is composed of two rigid fluorescent aromatic wings fused with a flexible eight-membered ring (cyclooctatetraene). This molecule flexibly changes its conformation between V-shape and planar shape in response to the external force. FLAP is excited by UV light and emits blue or green fluorescence depending on its conformation. We have visualized the tensile force distribution during deformation of a polymer film through ratiometric fluorescence imaging of FLAP doped in the film[1]. In the present study, the response of FLAP to hydrodynamic elongational stress was experimentally evaluated by measuring spatially averaged fluorescence spectra and two-dimensional fluorescence images of FLAP solution in microchannel flow. We used three types of FLAP whose molecular ends were chemically functionalized with PEG (polyethylene glycol) chains of different molecular weights (5000, 10000, and 20000), and their stress-responses were compared. Each FLAP-PEG was dissolved in a dimethyl sulfoxide/glycerol mixture, and the solution was supplied to a sudden contraction part of the microchannel at constant flow rates in the range of 2.2–446 μL/min. These flow rates correspond to the elongational stress of approximately 0.1–20 kPa at 40 μm upstream of the entrance of the contraction part. The spectral measurements showed that the intensity ratio of green (529 nm) to blue (494 nm) fluorescence increased monotonically with the flow rate, which indicates that the number of FLAP in planar shape increased with increasing elongational stress. In addition, FLAP bearing the longest PEG chain showed the highest increasing rate of the intensity ratio, which suggests the possibility of adjusting the stress sensitivity of FLAP by changing the length of the PEG chain. In fluorescence imaging, two-dimensional distributions of the fluorescence intensity ratio were visualized around the entrance of the contractions part. The intensity ratio varied significantly in the region where the velocity gradient was large in the streamwise direction.

REFERENCES

[1] Kotani, R., Yokoyama, S., Nobusue, S., Yamaguchi, S., Osuka, A., Yabu, H. and Saito, S., “Bridging pico-to-nanonewtons with a ratiometric force probe for monitoring nanoscale polymer physics before damage”, Nature Communications, Vol. 13, No. 303 (2022), 9pp.

UNSTEADY PRESSURE DISTRIBUTION MEASUREMENT ON CYLINDER SURFACE USING ANODIZED ALUMINUM PRESSURE-SENSITIVE PAINT DURING SHOCK WAVE PASSAGE

*S. Tamakuma[1,2], K. Yomo[1,2], T. Ogawa[1] and H. Nagai[1]

[1] Institute of Fluid Science, Tohoku University 2-1-1 Katahira, Aoba-ku, Sendai 980-8577, Japan

*shintaro.tamakuma.r7@dc.tohoku.ac.jp

[2] Department of Aerospace Engineering, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai 980-8577 Japan

ABSTRACT

Pressure-sensitive paint (PSP) is an optical sensor that can measure the global pressure distribution using the oxygen quenching of dye molecules. Anodized-aluminum pressure-sensitive paint (AA-PSP) has excellent oxygen permeability and responsivity among various PSPs and has been applied to visualize high-speed fluid phenomena. In order to capture such ultrafast phenomena using PSP and a high-speed camera, the frame rate of the camera must be fast, and the exposure time must be short to avoid blur. It is desirable for AA-PSP to have bright luminescence, high-pressure sensitivity, and fast response time. The time-response of AA-PSP depends mainly on the lifetime of luminescence and oxygen diffusion in a pore of an anodized aluminum layer[1].

Fujii et al. investigated the relationship between the electrolyte temperature and the pore size and depth[2]. The 90% rise time of their AA-PSP, the fastest time response to date is 0.81 μs. It has also been applied to visualize the interaction of a shock wave with a cylinder. However, the signal-to-noise ratio was poor, and image intensifiers were used due to the lack of luminescence intensity.

In this study, we conducted the time response tests using the shock tube based on the results of luminescence intensity and pressure sensitivity investigations at different anodizing times and dipping concentrations. In this test, the shock wave passing through the specimen coated with the PSP was visualized, and the rise time was calculated. When calculating the time constant, the delay in the rise time caused by blurring of the shock wave surface due to exposure time was corrected to calculate a more accurate time constant.

As a result, the 90% response time of this AA-PSP was approximately 2.6 μs.
This AA-PSP was applied to measure the unsteady pressure distribution on the cylinder surface when the shock wave passed over the cylinder. This resulted in the time series visualizations of the pressure distribution when the shock wave passed over the cylinder even under shooting conditions with a frame rate of 50,000 fps and an exposure time of 1.0 μs.
Future work will include measuring the unsteady pressure distribution as the shock wave passes through the cylinder and calculating the unsteady air force.

REFERENCES

[1] Kameda, M., Tezuka, N., Hangai, T., Asai, K., Nakakita, K., and Amao, Y. “Adsorptive Pressure-Sensitive Coatings on Porous Anodized Aluminium.” Measurement Science and Technology, Vol. 15, No. 3, 2004, pp. 489–500.

[2] Numata, D., Fujii, S., Nagai, H., and Asai, K. “Ultrafast-Response Anodized-Aluminum Pressure-Sensitive Paints for Unsteady Flow Measurement.” AIAA Journal, Vol. 55, No. 4, 2017, pp. 1118–1125.

DEVELOPMENT OF PRESSURE DISTRIBUTION MEASUREMENT TECHNOLOGY ON FREE-FLIGHT OBJECT SURFACE AT TRANSONIC SPEED

*Y. Hosono[1,2], K. Yomo[1,2], K. Takahashi[1], T. Ogawa[1], K. Ohtani[1] and H. Nagai[1]

[1] Institute of Fluid Science, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi 980-8577 Japan
*hosono.yota.q6@dc.tohoku.ac.jp
[2] Department of Aerospace Engineering, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi 980-8577 Japan

ABSTRACT

A measurement technology using Pressure-Sensitive Paint (PSP) is developed to capture the surface pressure distribution of an object during free flight at transonic speed. In a PSP measurement, the pressure distribution is calculated by substituting the ratio of the pressure-independent image (reference image) and the pressure-dependent image (signal image) into the calibration equation. However, in a measurement of unsteady motions, the reference image required for the signal image changes in time. Therefore, it is necessary to adjust the position and attitude of the object in the images. As a solution to this issue, a two-color PSP measurement technology has been proposed, which enables simultaneous capturing of both images [1]. On the other hand, this measurement is performed with a color camera, which is inferior in ISO sensitivity, and the signal-to-noise ratio is not sufficient.

Thus, this study aims to develop a measurement technology using the intensity-based method, which allows us to use a monochrome camera and measure with a higher signal-to-noise ratio. Then, to capture a large number of reference images corresponding to the position and attitude of the signal image in advance, we developed a system that automatically collects reference images with 3-axis attitude control. The models measured were cylinder models ($\phi 30 \times 15$ mm), and Hayabusa capsule models (1/16 scale), the surfaces of which were coated with anodized aluminum PSP (AA-PSP). The models were flighted at a speed of M=1.1, and the luminescence emissions of the PSP in flight were captured with a high-speed monochrome camera at a rate of 100 kHz. The captured data showed that the signal-to-noise ratio was more than 10 when the exposure time was more than 3 μs.

REFERENCES

[1] Sakaue, H., Miyamoto, K., and Miyazaki, T., “A motion-capturing pressure-sensitive paint method”, J. Appl. Phys. 113, 084901, (2013).

VISUALIZATION STUDY OF NANO PARTICLE BEHAVIOR WITHIN PORES

*Y. Abe[1], Y. Matsuda[1]

[1] Department of Modern Mechanical Engineering, Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo 169-8555, Japan
*ya-jupiter0309@toki.waseda.jp

ABSTRACT

Porous media are materials that have many pores inside, and they have the property of adsorption of particles. Due to this property, porous media are widely used in industries. Therefore, it is important to clarify how particles behave within pores. In this study, we visualize nano particle behaviour within pores using single-particle tracking. Single-particle tracking enables us to observe particles within the measurement object directly. For example, single-particle tracking is used to reveal particle diffusion in a cell [1] and polymer layer [2],[3]. Moreover, we analyze particle behaviour using physical property values and find out unique features of particle behaviour within pores.

REFERENCES

[1] Saxton M and Jacobson K, “Single-particle tracking: Applications to Membrane Dynamics”, Annual Review of Biophysics and Biomolecular Structure, Vol. 26 (1997), pp. 373-399.

[2] Wöll D, Braeken E, Deres A, Schryver F, Uji-I H, and Hofkens J, “Polymers and single molecule fluorescence spectroscopy, what can we learn?”, Chemical Society Reviews, Vol. 38 (2009), pp. 313-328.

[3] Iwao R, Yamaguchi H, Niimi T, Matsuda Y, “Single-molecule tracking measurement of PDMS layer during curing process”, Physica A: Statical Mechanics and its Applications, Vol. 565 (2021), p.125576.

Development of Paddlewheel Aerator Performance Predictions Model

* P. N. A. Nugroho[1], M.A. Mustaghfirin[1], D.S.A. Pambudi[1], E. Novianarenti[1], D.A.Wulandari[2] and S. Torii[3]

[1] Politeknik Perkapalan Negeri Surabaya, 60111, Indonesia
*priyambodo@ppns.ac.id
[2] Department of Mechanical Engineering, Faculty of Engineering, Universitas Negeri Jakarta, Indonesia.
[3] Department of Advanced Mechanical System Engineering, Kumamoto University, Kurokami, 2-39-1, Kumamoto 860-8555, Japan

ABSTRACT

The intensive aquaculture industry is mainly affected by the ability to maintain the water quality, and low dissolved oxygen could be improved through the aeration process. The paddlewheel aerator is one of the supporting devices required for the intensive aquaculture pond system. The paddlewheel aerator still has a low aeration performance, resulting in higher operational costs. This paper presents an analytical model to predict the optimal performance of a paddlewheel aerator. The model considers the most influential factors affecting the performance of the paddlewheel aerator. Then, Computational Fluid Dynamics (CFD) is employed to simulate and validate the developed analytical model. The experimental model is also performed to validate the result. The simulation results demonstrate that the model is accurate enough to be used for estimating the paddlewheel aerator's optimal operational condition.

Keywords: paddlewheel aerator, analytical model, experimental model

REFERENCES

[1] World Resources Institute 2012 Coastal and Marine Ecosystems - Marine Jurisdictions: Coastline length
[2] Leung P, Carole R 2006 Shrimp Culture: Economics, Market, and Trade *Wiley-Blackwell* 5
[3] Kooiman J 2005 Fish for Life: Interactive Governance for Fisheries *Amsterdam University Press* 94
[4] Fast A 1992 Penaeid Growout Systems: An Overview *Marine Shrimp Culture* Amsterdam Elsevier 345-353
[5] Boyd C 1998 Pond water aeration systems *Aquacultural Engineering* 18 1 9-40,
[6] Moore J, and Boyd C 1992 Design of small paddlewheel aerators *Aquacultural Engineering* 11 1 55-69
[7] Boyd C, Torrans E, and Tucker C 2018 Dissolved oxygen and aeration in ictalurid catfish aquaculture *Journal of the World Aquaculture Society* 49 1 7-70
[8] Boyd C, Darryl E, Chamberlain G Global Aquaculture 2006 Aerated pond management in Operating Procedures for Shrimp Farming: Global Shrimp OP Survey Results and Recommendations, 1st ed. St. Louis, MO, USA: *Global Aquaculture Alliance* 68-75
[9] Zhu D, Cheng X, Sample D, and Yazdi M 2020 Effect of intermittent aeration mode on nitrogen concentration in the water column and sediment pore water of aquaculture ponds *Journal of Environmental Sciences* 90 331-342
[10] Hoagland R, Rouse D, Teichert-Coddington, and Boyd C 2001 Evaluation of automated aeration control in shrimp ponds *Journal of Applied Aquaculture* 11 3 45-55

STUDY THE EFFECT OF DOWNSTREAM BLUFF BODY TO ENHANCE FIV-BASED ENERGY HARVESTING, A VISUALIZATION STUDY

*S. Hosseini[1], *A. Afsharfard[2], K.C. Kim[1]

[1]School of Mechanical Engineering, Pusan National University, Busan 46241, Republic of Korea
sajjad_hosseini@pusan.ac.kr
[2] Mechanical Engineering Department, Ferdowsi University of Mashhad, Mashhad, Iran
afsharfard@um.ac.ir

ABSTRACT

With the global population swelling and industrialization on the rise in developing nations, humanity's hunger for energy has reached to unprecedented levels. As a results different types of the energy harvesters are developed by researchers within the last three decades and it is remained as a necessary topic for global needs. In this study a modified Flow Induced Vibration (FIV) based energy harvester is presented. In this system, effect of employing the second bluff body behind of a nature-inspired FIV-based energy harvester is studied. In doing so multidisciplinary research that studies the following contents is presented:

(a) experimentally studying the approaches to increase amplitude of FIV
(b) utilizing the enhanced vibrations to extract energy

In this way, to increase the oscillation amplitude of bluff body and consequently increase in the energy harvesting, the near-wake flow is studied comprehensively using smoke and laser flow visualization and Particle Image Velocimetry (PIV). Then the electromechanical equation of motion for the piezoelectric vibration-based energy harvester is derived. The experimentally validated electromechanical equations of motion are used to study the FIV-based energy harvesting.

REFERENCES

[1] Lam, K., Wang, F. H. & So, R. M. C., 2004. Three-dimensional nature of vortices in the near wake of a wavy cylinder. Journal of Fluids and Structures, Volume 19, pp. 815-833.

[2] Xia, C. et al., 2018. POD analysis of the wake behind a circular cylinder coated with porous media. Visualization Society of Japan, Volume 21, pp. 965-985.

FSI-BASED DESIGN OF A DVA FOR DECREASING UNDESIRED VIBRATIONS OF A ROTATING BEAM, A VISUALIZATION STUDY

*A. Shahsavari[1], *A. Afsharfard[2], K.C. Kim[1]
[1]School of Mechanical Engineering, Pusan National University, Busan 46241, Republic of Korea
amirreza @pusan.ac.kr
[2] Mechanical Engineering Department, Ferdowsi University of Mashhad, Mashhad, Iran
afsharfard@um.ac.ir

ABSTRACT

In this study, a Dynamic Vibration Absorber (DVA) will be designed to decrease undesired lateral vibrations of a rotating beam like wind turbine blade. To do so, the effect of aerodynamic forces on the dynamic response of the rotating beam will be studied. Then the DVA is designed according to the Fluid Solid Interaction (FSI) content. In other words, the dynamic behaviour of the rotating beam is studied from the following perspectives:

- Effects of attaching or detaching the DVA on its dynamic response
- Effect of using the DVA on the aerodynamics of the rotating beam

For studying the dynamic effect of the DVA on the vibratory behaviour of the rotating beam, it is analytically modelled using the Euler Bernoulli model with presence of lateral excitation. The dynamic effect of DVA is considered as an external force, and its application for decreasing lateral vibration is studied. Then, the aerodynamic effect of attaching the DVA to the rotating beam is experimentally studied. The experiments were performed in a wind tunnel at a speed of 1.1 m/s on two models of the antenna with and without DVA on the surface of the rotating beam. Smoke and laser flow visualization was used to better understanding the effect of attaching the DVA on the flow over the rotating beam. The flow visualization tests were performed in two sections, one section perpendicular to the surface of the antenna, and the antenna surface section at three different angles. Changes in flow over the rotating beam and the wake region were also examined and compared by flow visualization results. Finally, the geometry of DVA is changed to improve the aerodynamic properties of the rotating beam and decrease its undesired lateral vibration.

REFERENCES

[1] Zhang, Mingjie, and Fuyou Xu. "Tuned mass damper for self-excited vibration control: Optimization involving nonlinear aeroelastic effect." Journal of Wind Engineering and Industrial Aerodynamics 220 (2022).
[2] Zhang, Zili. "Optimal tuning of the tuned mass damper (TMD) for rotating wind turbine blades." Engineering Structures 207 (2020).

IMPROVEMENT IN ACCURACY OF THE BACKGROUND ORIENTED SCHLIEREN TECHNIQUE WITH PARALLEL PROJECTION

*N. Kosaka[1], Y. Igari[1], Y. Katagiri[1], M. Yamagishi[1], M. Ota[1]

[1] Chiba University, 1-33 Yayoi Inage, Chiba, 263-8522, Japan
*nao.kosaka@chiba-u.jp

ABSTRACT

Visualization of supersonic flow field around the test model is important in the development of aircraft and spacecraft. The schlieren or shadowgraph methods have been commonly used as visualization methods. These methods measure the refractive index change in the medium as brightness change in the image plane due to the refraction of the light ray. If the density can be written with refractive index using Glad-stone Dale relation, density information can also be obtained. High sensitivity measurement is possible with a parallel light observation however, the schlieren method is difficult to measure the density quantitatively and is used for qualitative visualization as a projected two-dimensional image. We have studied the Background Oriented Schlieren (BOS) technique which is a new visualization method developed by Meier.[1] The BOS technique measures the displacement of the background image caused by refractive index change as for the schlieren method. The big advantage of the BOS is that quantitative measurement is possible with a very simple setup consist of a background and a digital camera. Quantitative density measurement is achieved with image analysis of the acquired images. However, drawbacks are that the sensitivity is lower than that of the Schlieren method and that the camera cannot focus on both the background image and the test model. In order to overcome these drawbacks, the telecentric optical system [2] and digital projector are introduced into BOS measurement. The parallel projection of the test model with the flow field can be captured with this setup and the background position can be set at any position. Experiments are carried out at suction-type supersonic wind tunnel at Chiba University to evaluate the proposed BOS setup. As a result, the experimental data were confirmed to be proportional to the theoretical values.

REFERENCES

[1] G.E.A. Meier. Computerized background-oriented schlieren. Experiments in Fluids, Vol.33, pp181-187, 2002

[2] Masanori Ota, Friedrich Leopold, Ryusuke Noda, Kazuo Maeno. Improvement in spatial resolution of background-oriented schlieren technique by introducing a telecentric optical system and its application to supersonic flow. Exp. Fluids Vol. 56, 2015

STEREO-PIV MEASUREMENT OF A BI-STABLE WAKE BEHIND A SIMPLIFIED SEDAN-TYPE VEHICLE MODEL IN A CROSSWIND

*C. Hasegawa[1], T. Hiraoka[1], T. Nakashima[1], T. Kanehira[1],
H. Mutsuda[1], K. Shimizu[2] and Y. Nakamura[2]

[1] Hiroshima University, 1-4-1 Kagamiyama, Higashi-Hiroshima, Hiroshima 739-8527 Japan
*m220112@hiroshima-u.ac.jp
[2]Mazda Motor Corporation, 3-1, Shinchi, Fuchu-cho, Aki-gun, Hiroshima 730-8670 Japan

ABSTRACT

In the automotive aerodynamics field, development efforts are underway to reduce aerodynamic drag and improve driving performance. Conventionally, these performances are evaluated under yaw zero wind conditions. However, in recent years, particular attention has been paid to aerodynamic performance in the real world, where the apparent wind direction changes due to a natural wind on the road, another vehicle traveling around. Especially, the wind direction change sometimes causes sudden change in aerodynamic characteristics at a certain wind direction. When such aerodynamic change occurs, each wake flow structure causing the two aerodynamic characteristics may randomly appear around the critical wind direction. Furthermore, transient change in wind direction across the critical wind direction cause long-time delay of the change in the wake flow structure and aerodynamic characteristics. Then, undesirable characteristics due to a short disturbance sometimes maintain for a long time. These phenomena make an aerodynamic performance of a vehicle worse and they must be suppressed. In order to develop aerodynamic techniques to suppress them, it is necessary to clarify the wake structure of the two stable states in detail.

In the present study, wake flow structures of a simplified sedan-type automobile model [1], for which bi-stability has been reported in previous studies, was experimentally clarified by a stereoscopic PIV measurement. The bi-stable velocity distributions under the same wind direction condition were measured in a number of parallel planes, respectively. Then, by stacking the results in their normal direction, three-dimensional wake structures in each stable state were visualized in detail. From a comparison between the two stable wake structures, it was shown that the main difference of flow feature regarding the sudden change in aerodynamic characteristics was the entrainment flow from the side of the model to the base of the model. It was also found that the increased entrainment flow caused the formation of longitudinal vortices, which may significantly increase the aerodynamic drag.

REFERENCES

[1] Nakamura, Y. et al., “Identification of wake vortices in a simplified car model during significant aerodynamic drag increase under crosswind conditions, Journal of Visualization (2022), DOI: 10.1007/s12650-022-00837-8.

NUMERICAL ANALYSIS ON THE TEMPERATURE OF AN ETHANOL SPRAY IN ITS MIXTURE FORMATION PROCESS UNDER DIESEL ENGINE OPERATING CONDITIONS

*H. Saitoh[1] and H. Tsunoda[2]

[1] Department of Mechanical Engineering, Sojo University, Ikeda 4-22-1, West Ward, Kumamoto City 860-0082, Japan
*saitoh@mec.sojo-u.ac.jp
[2]Former Student of the Graduate School of Mechanical Engineering, Sojo University, Ikeda 4-22-1, West Ward, Kumamoto City 860-0082, Japan

ABSTRACT

This study deals with the development of controlled-ignition technology for high performance CI (Compression Ignition) alcohol engines. Among the alcohol fuels, we focused on ethanol as one of the promising candidates of alternative fuels. The objective of this study is to physically and chemically reveal the mixture formation process up to auto-ignition of an ethanol spray. In our previous experiments and numerical analysis by using a constant volume combustion chamber (CVCC), surrounding gas temperature and pressure conditions before fuel injection for stable auto-ignition of an ethanol spray were revealed. As the next step, we originally designed and built up a rapid compression and expansion machine (RCEM) with variable compression ratio and supply air pre-compression and heating device in order to investigate the effect of gas flow in the combustion chamber after compression stroke on the mixture formation process of an ethanol spray. Prior to experiments, we performed numerical analysis of fluid flow in the intake and compression strokes of the RCEM, and we also performed the mixture formation process of an ethanol spray in the expansion stroke. Instantaneous spatial distributions of excess air ratio, temperature, and activated chemical species inside a spray and their temporal histories from fuel injection were calculated by using the commercial CFD code "CONVERGE". Results showed that enhancement of atomization and evaporation of ethanol due to the fluid flow induced by the compression stroke physically obstructed temperature increase of a spray, and endothermic reactions of H_2O_2 and HO_2 generations chemically obstructed temperature increase. Generations of OH radical and H atom which are recognized as the activated chemical species relating the chain branching reactions as auto-ignition occurred after the H_2O_2 and HO_2 generations. These results allow us to draw one conclusion; Amount of heat supplied into an ethanol spray by the entrainment of high-temperature surrounding gas has to be superior to the above introduced physical and chemical factors of temperature-increase-obstruction for auto-ignition. In this numerical prediction of ethanol spray mixture formation process, sufficient surrounding gas temperature for stable auto-ignition existed between 1100K and 1200K.

REFERENCES

[1] H. Saitoh, K. Uchida, N. Watanabe：Numerical Study on the Required Surrounding Gas Conditions for Stable Auto-Ignition of an Ethanol Spray, Journal of Combustion, Volume 2019-10, Article ID 1329389, (2019), https://doi.org/10.1155/2019/1329389

[2] N.M.Marinov：Detailed Chemical Kinetic Model for High Temperature Ethanol Oxidation, International Journal of Chemical Kinetics, Lawrence Livermore National Laboratory, p.183-220(1999)

MICROSCOPIC IMAGE DIAGNOSTICS WITH VISIBLE EMISSION LIGHT FOR SELF-BURNING SURFACE TEMPERATURE DISTRIBUTION OF CFRP (CARBON FIBER REINFORCED PLASTIC) IN OXYGEN

*Yojiro Ishino[1] and Rion Ito[1]

[1] Nagoya Institute of Technology, Gokiso-cho, Showa-ku, Nagoya 466-8555 Japan

ABSTRACT

Carbon Fiber Reinforced Plastic(CFRP) has been widely used in variety industries because of its high strength-density ratio. Therefore disposal method of waste CFRP should be considered in advance now. We have been proposed a combustion incineration of carbon fiber(CF) residue after recycle use. However, carbon fiber residue is almost impossible to be burned out by even high temperature burnt gas because of extremely high melting point of carbon fiber and a low oxygen concentration in burnt gases[1].

In this study, self-sustained combustion of carbon fiber in oxygen has been reported. In order to obtain information of the self-sustaining mechanism of the combustion, in this paper, a convenient optical technique to acquire the 2D distribution of high surface temperature of glowing solid material was conducted.

In this technique, magnified thermographic view of the high temperature specimen was provided by comparison between RGB brightness information of conventional photograph of the visible light emission of the surface and previously-calculated values from Planck's law and the spectral sensitivity of the camera system.

In this paper, first, accuracy of the method was verified by comparison between the 2D temperature distribution result of a glowing fine (0.5 mm dia.) graphite rod in burnt gas of a propane-air premixed laminar flame and its estimated temperature distribution measured by PR-type thermocouple and compensated by Kaskan's method. Next magnified instantaneous 2D temperature diagnostics of self-burning carbon fiber bundle in oxygen was made. The 2D surface temperature distribution result gives very high temperature over 2900 K on carbon fiber surface in in-oxygen combustion incineration.

Key Words: CFRP, Oxygen Combustion, 2D Measurement of Surface Temperature, Visible Light Emission

REFERENCES

[1] Ishino, Y. and Ishihara, Y., “Temperature Measurement of Self-burning CFRP(Carbon Fiber Reinforced Plastic) in Oxygen (Image Diagnostics of High Surface Temperature with Visible Light Emission)”(in Japanese), Proceedings of Thermal Engineering Conferrence 2021, Japan Society of Mechanical Engineers, No.21-35, (2021), pp. 1-6.

NANOPARTICLES ACCUMULATION IN A PREDETERMINED POSITION UNDER AC ELECTRIC FIELD

*A. Abdelghany[1], Y. Okui[1], Y. Ichikawa[1,2], and M. Motosuke[1,2]

[1] Department of Mechanical Engineering, Tokyo University of Science, 6-3-1 Niijuku, Katsushika, Tokyo 125- 8585, Japan.
*mot@rs.tus.ac.jp
[2]Water Frontier Research Center, Research Institute for Science and Technology, Tokyo University of Science, 1-3 Kagurazaka, Shinjuku, Tokyo 162-8601, Japan.

ABSTRACT

Particles accumulation under AC electric filed around electrodes embedded in a microfluidic device is an efficient technique. However, the accumulation position and performance occasionally depend on the design of the electrode array and applied voltage characteristics. For instance, on symmetric coplanar electrodes at a low-frequency range, typically < 1kHz, the interaction between the electric field and the electric double layer generates a fluidic motion named as AC electroosmosis (ACEO). ACEO flow is always perpendicular to the electrode edge, driving fluid onto the electrode surface, from the gap between the electrodes across the electrode surface [1]. Thus, using single-gap parallel symmetric coplanar electrodes, the particles will accumulate in two different positions far from the electrodes' gap [2]. Furthermore, enhancing the accumulation performance may require a limited design for the electrodes such as the double-gap electrodes [3].

In this study, the use of coplanar symmetric electrodes was also able to induce a flow to the gap between the electrodes in a high-frequency range, namely 50-100 kHz. This flow is used to accumulate polystyrene nanoparticles (PsNPs) in a microfluidic device embedded with symmetric coplanar electrodes. Different from previous research that used the symmetric coplanar electrodes to accumulate particles at two different positions under low frequency [2], the symmetric coplanar electrodes microfluidic device is utilized to accumulate the nanoparticles to a predetermined single spot at high frequency. The concentration performance was systematically studied by using particles with a size of 100 nm. In addition, the relation between the concentration factor (CF) and voltage characteristic was also examined, and as a result, we found that accumulation location and resultant factor were completely changed according to applied frequency and voltages.

REFERENCES

[1] Green, N.G., Ramos, A., González, A., Morgan, H. and Castellanos, A., "Fluid flow induced by nonuniform ac electric fields in electrolytes on microelectrodes. I. Experimental measurements.", Physical review E, Vol. 61, No. 4 (2000), pp. 4011-4018.

[2] Islam, N. and Wu, J., "Microfluidic transport by AC electroosmosis.", Journal of Physics: Conference Series, Vol. 43, No. 1 (2006), pp. 356 -361.

[3] Motosuke, M., Yamasaki, K., Ishida, A., Toki, H. and Honami, S., "Improved particle concentration by cascade AC electroosmotic flow." Microfluidics and Nanofluidics, Vol. 14, No. 6 (2013), pp. 1021-1030.

RESOLUTION ENHANCEMENT BY THE EDSR METHOD FOR BOS ANALYSES

*K. Ota[1] and T. Ukai[1]

[1] Osaka Institute of Technology, 5-16-1 Omiya, Asahi-Ku, Osaka 535-8585 Japan
*m1m21407@st.oit.ac.jp

ABSTRACT

Background Oriented Schlieren (BOS) methods are substituted for the conventional schlieren methods for quantitative density measurements. A schlieren system requires the expensive optical components such as concave mirror and lens, whereas the BOS method has an advantage in that it can take images using only a digital camera, light source, and background pattern, thereby allowing the measurements without the complicated optical components [1]. Additionally, the flow field can be quantitatively visualized using post processing in a BOS analysis. The cross-correlation method, which vector resolution is reduced than an original image resolution, is used in the most of post processing of the BOS analysis. Therefore, the enhancement of the vector resolution is necessary to visualize detailed flow structure. important for visualizing a detailed flow field.

A super-resolution technique based on convolutional neural networks is used for resolution enhancement. An application of the super-resolution technique for the flow field is not too much. One of the applications is that Kong *et al.* used the super-resolution technique for a supersonic flow visualized by the schlieren method [2]. Additionally, Liu *et al.* reproduced the turbulent structure of a two-dimensional flow field using super-resolution technique [3]. In these typical applications, they focus generally on the enhancement of the image resolution without post image processing. The post processing plays a key role in the BOS analysis, and the effectiveness of the super-resolution technique for such a post processing is unknown.

In this study, we applied EDSR (Enhanced Deep Residual Networks for Single Image Super-Resolution) method, which is a representative super-resolution technique, to the BOS analysis. The BOS images and the open source images were used for training in the machine learning. We used the Bicubic interpolation and the EDSR method to evaluate the comparison of their performances. Consequently, the EDSR method provided a better quality of the BOS image than that of the Bicubic interpolation. And then, we will investigate the effects of the different interrogation window sizes on the BOS image which its resolution is enhanced by the EDSR method.

REFERENCES

[1] G. Meier, “Computerized background-oriented schlieren,” Exp. Fluids 33, pp.181-187 (2002).

[2] C. Kong, J. Chang, Z. Wang, Y. Li, W. Bao, “Data-driven super-resolution reconstruction of supersonic flow field by convolutional neural networks,” AIP Adv. 11, 065321 (2021).

[3] B. Liu, J. Tang, H. Huang, X.Y. Lu, “Deep learning methods for super-resolution reconstruction of turbulent flows,” Phys. Fluids 32,025105 (2020).

PSFVIP13-101

Free surface flow of concentrated particle suspension in Hele-Shaw cell

T. Koshiba[1] and T. Yamamoto[2]

[1] National Institute of Technology, Nara College, 22, Yatacho, Yamatokoriyama, Nara 639-1080 Japan
koshiba@mech.nara-k.ac.jp
[2]Osaka Electro-Communication University, 18-8, Hatsucho, Neyagawa, Osaka 572-8530 Japan

ABSTRACT

Particle suspensions dispersed solid particles in liquid are seen in many fields, for example, the manufacturing of composite materials, food industry, the development of pharmaceutical, and civil engineering, etc. In general, it is well known that the flow of particle suspension is affected by various factors relating to dispersions and solvents. Especially, particle concentration is one of important factors and closely relates to complex phenomena in flow. In the case of concentrated suspension, peculiar phenomena like a jamming and a phase transition arise from interaction between particles. The effect of the particle interaction on flow behavior of suspension is interesting in the view of rheology. In this study, the particle behaviors in free surface flow of suspension were examined experimentally. As free surface flow, we used a radial flow pattern flowing in a Hele-Shaw cell which is constructed by two parallel plates. Furthermore, we prepared six kinds of suspension with different concentrations and particle sizes as test fluids and observed temporal change in interface shape in flow. The suspension flow was controlled by adjusting a supply pressure to Hele-Shaw cell. In the observation of free surface, we found that the undulation of liquid surface with time occurred. Additionally, in the inside region of the concave and convex part on free surface, we confirmed the non-uniformity of particle concentration suggesting particle aggregation [1]. The undulation of free surface depends on the particle concentration and the pitch of undulation becomes large with increasing the particle concentration. Furthermore, in experiments using smaller size of particles, we found that the undulation of free surface transited to the viscous fingering seen in flow of two fluids with different viscosity. In the radial flow pattern between parallel plates, since the velocity component of the radial direction decreases with increment in interface radius, the mean shear rate acting on fluids also decreases. Accordingly, it was considered that the non-uniformity of particle concentration seen in the inside of free surface suggested the hydrodynamical interaction between particles accompanying to the change in shear deformation.

REFERENCES

[1] Luo, R., Chen, Y. and Lee S., “Particle-Induced Viscous Fingering: Review and Outolook”, PHYSICAL REVIEW FLUIDS, Vol. 3, (2018), pp. 110502-1-110502-15.

SIMULTANEOUS MEASUREMENT OF GASEOUS FLOW AND SPARK CHANNEL USING INEXPENSIVE INORGANIC FLUORESCENT TRACER

*C. Kondo[1] and M. Yoshioka[1]

[1] Okayama University of Science, 1-1 Ridai-cho, Kita-ku, Okayama 700-0005 Japan
*c-kondo@ous.ac.jp

ABSTRACT

Flow dynamics near the spark-plug gap and its effect on the behavior of spark dicharge and the development of initial flame kernel have become important in promoting more stable lean/dilute combustion in internal combustion engines. One of the experimental techniques for measuring instantaneous gaseous flow near solid objects such as a spark plug is fluorescent particle image velocimetry (f-PIV). However, many of the available fluorescent particles are expensive and/or combustible because they are organic materials. Then, previous our studies have proposed mass production of inexpensive inorganic-based fluorescent particles (f-particles), and we found that such f-particles have also been used in f-PIV measurements of the air flow around a spark-plug gap in automotive applications, and the flow even between the plug gap could be detected under atmospheric conditions [1]. In the present study, we investigated the possibility of simultaneous measurements of the spark channel and the flow behind of it including even between a spark-plug gap using f-PIV with the proposed f-particles, in future in order to validate the corresponding numerical models, such as the behavior of spark channel. As a result, under atmospheric air condition, the fluorescence images of particles was able to be detected, separating from the spectrum of spark discharge even between the gap of spark plug at spark timing, using combinations of the proposed f-particles and adequate optical band-pass filters, then the flow behind of spark channel was analyzed. This measurement technique also possibly leads to validate the models of development and extinction of initial flame, by developing the simultaneous measurement of gaseous temperature, under investigation using f-particles which emit multi-color fluorescence [2].

REFERENCES

[1] Kondo, C., “Fluorescent PIV measurement of gaseous flow near spark plug gap using inexpensive inorganic fluorescent tracer”, Proc. The 29th International Symposium on Transport Phenomena, No.58 (2018), pp.1-6.

[2] Kondo, C. and Yoshioka, M., “Production of two - color inexpensive inorganic fluorescent tracer as an advancement toward the simultaneous measurement of gaseous flow and temperature in engines”, Proc. The 32nd International Symposium on Transport Phenomena, No.90 (2022), pp.1-7.

EFFECT OF CONTTAINER SIZE ON FLOW STRUCTURE IN A RECTANGULAR CONTAINER INJECTED AIR FROM A SLIT NOZZLE

*T. Ichihara[1], T. Kiwata[2], K. Toyoda[3], H. Uchida[4], Y. kawamura[4] and M. Tsuji[4]

[1] Graduate School of Natural Science and Technology, Kanazawa University
Kakuma-machi, Kanazawa-shi, Ishikawa 920-1192, Japan
*itakuto@stu.kanazawa-u.ac.jp
[2] School of Mechanical Engineering, Kanazawa University
Kakuma-machi, Kanazawa-shi, Ishikawa 920-1192, Japan
[3] Hokkaido University of Science,
7-Jo 15-4-1 Maeda, Teine, Sapporo, Hokkaido 006-8585, Japan
[4] Universal Can Corporation, 1500 Suganuma, Oyama-cho,
Sunto-gun, Shizuoka 410-1392, Japan

ABSTRACT

Plane jets have been used for drying, heating, cooling, dust removal, and many other applications. The jet injected from a nozzle inserted in a container is used to dry and heat on the inside wall of the container. Shakouchi et al. [1] and Mataoui et al. [2] have shown the effects of nozzle length and location and Reynolds number on the flow structure in a container with an inserted rectangular nozzle. However, the effect of jets injected from outside the container has not been examined. The present paper experimentally investigates the effect of container size on the flow structure in a container when air is injected from a slit nozzle installed outside the container.

Air was injected into the container through a slit nozzle of 5 mm in width b_0 and 300 mm in length, which was installed at the top of the container. The Reynolds number of the jet was changed from Re = 2,000 to 4,000 ($Re = U_0 b/\nu$, where U_0 is the velocity at the center of the nozzle exit and ν is the air kinematic viscosity). Three types of containers with widths of D = 50, 60 and 70 mm were tested. The velocity in the container was measured by PIV for aspect ratios of H/D = 1 to 3, where H is the container depth. The nozzle position was also changed horizontally from the center of the container to the container wall.

It was found that the aspect ratio H/D of the container and the Reynolds number of the jet had an influence on the flow structure in the container. When the H/D of the container is small, a single recirculation vortex region is generated in the whole container. When the H/D of the container is large, two recirculation vortex regions are created in the container. Vortex 1 is generated at the upper part of the container, and vortex 2 is generated at the lower part of the container and rotates the opposite direction of vortex 1. The size of vortex 1 does not change as the depth of the container increases. However, the size of vortex 2 expanded as the depth of the container increases.

REFERENCES

[1] Shakouchi, T., Suematsu, Y. and Ito, T., “A Study on Oscillatory Jet in a Cavity”, Bulletin of JSME, Vol. 25, No. 206 (1982), pp.1258-1265.

[2] Mataoui, A., Schiestel, R. and Salem, A., “Flow Regimes of Interaction of a Turbulent Plane Jet into a Rectangular Cavity: Experimental Approach and Numerical Modelling”, Flow, Turbulence and Combustion, Vol 67, No.4 (2001), pp.267-304.

EVALUATION OF PRESSURE DISTRIBUTION AROUND SMALL ROTOR BLADE IN LOW OXYGEN PARTIAL PRESSURE REGION USING PSP

K. Ogawa[1], *Y.R. Li[1], S. Someya[2], T. Inagaki[1], S. Saito[2], S. Baba[2] and N. Takada[2]

[1] Graduate School of Sci. and Eng., Ibaraki Univ., 4-12-1 Nakanarusawa, Hitachi 316-8511
[2] AIST, 1-2-1 Namiki, Tsukuba 305-8564 Japan
*yanrong.li.mech@vc.ibaraki.ac.jp

ABSTRACT

In recent years, pressure distribution measurement with high accuracy by PSP (Pressure Sensitive Paint) has been performed for high-speed moving objects with large pressure changes. However, there remains a problem in measuring pressure around low-speed moving objects such as small rotating machines, because the pressure sensitivity of PSP near atmospheric pressure is too small. In the present study, the effects of oxygen partial pressure on the static properties of PSP were investigated. Sample coupon tests using a vacuum container were conducted for PdTFPP/Poly(IBM), PtOEP/Poly(IBM), $Ru(dpp)_3$/Poly(IBM) and $Ru(dpp)_3$. The oxygen partial pressure was set to be between 2 - 10 kPa (with an interval of 2 kPa) and the total pressure was set to be between 20 - 100 kPa (with an interval of 20 kPa). The intensities of PSP emission, that are normalized by the maximum intensity in oxygen partial pressure = 0 kPa, i.e., the relative intensity of PSP emission, decreases with the increasing of oxygen partial pressure. Total pressure had almost no effect on the relative intensity change of tested PSP. The relationship of the rate of change in relative intensity is $Ru(dpp)_3$/Poly(IBM) > $Ru(dpp)_3$ > PtOEP/Poly(IBM) > PdTFPP/Poly(IBM). $Ru(dpp)_3$/Poly(IBM) shows the highest pressure sensitivity, where PdTFPP/Poly(IBM) shows the lowest pressure sensitivity between the tested PSP.

Acknowledgment

This work was supported by JSPS KAKENHI Grant Number JP20K20978.

REFERENCES

[1] Taku NONOMURA, Keisuke ASAI, “Simple estimation of Frequency Response of Two-layer Pressure sensitive pint Model”, *Trans. Japan Soc. Aero. Space Sci.*, Vol. 62, No. 2, pp. 112–115, 2019

NOISE SUPPRESSION METHOD FOR PSP DATA BASED ON REDUCED-ORDER MODELING

*T. Inoue[1], Y. Matsuda[1,2], T. Ikami[3,4], T. Nonomura[3], Y. Egami[5] and H. Nagai[4]

[1] Department of Modern Mechanical Engineering, Waseda University, 3-4-1 Ookubo, Shinjuku-ku, Tokyo, 169-8555, Japan
* deseiko@ruri.waseda.jp
[2] Japan Science and Technology Agency, PRESTO, Saitama 332-0012, Japan
[3] Department of Aerospace Engineering, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi-prefecture 980-8577, Japan
[4] Institute of Fluid Science, Tohoku University, 2-1-1 Katahira, Aoba-ku, Sendai, Miyagi-prefecture 980-8577, Japan
[5] Department of Mechanical Engineering, Aichi Institute of Technology, 1247 Yachigusa, Yakusa-Cho, Toyota, Aichi-prefecture 470-0392, Japan

ABSTRACT

PSP (Pressure-Sensitive Paint) is one of the effective pressure measurement techniques. The PSP method is based on oxygen quenching of the phosphorescence emitted from the PSP coating, and pressure can be measured by detecting the variation of the phosphorescence using a camera. Since the PSP images contain noticeable noise such as the camera shot noise and the readout noise, it is important to remove the noise to detect small fluctuation of the pressure.

One of the most effective ways to remove noise is to average images taken under the same conditions. But this method cannot be used for unsteady pressure field.

Therefore, we propose a method to remove noise from unsteady pressure field data. based on the proper orthogonal decomposition (POD) calculated from time-series PSP data and the LASSO (Least Absolute Shrinkage and Selection Operator) regression. The noise is suppressed by truncating the lower modes of POD. To determine the coefficients for each POD mode, we use the sensor selection algorithm and LASSO regression. We applied our method to the pressure field induced by the Kármán vortex street behind a square cylinder. As a result, the error between the processed PSP data and the pressure tap data is greatly reduced, indicating that the proposed method is effective in removing the noise.

REFERENCES

[1] Inoue, T., Matsuda, Y., Ikami, T., Nonomura, T., Egami, Y., Nagai, H., “Data-Driven Approach for Noise Reduction in Pressure-Sensitive Paint Data Based on Modal Expansion and Time-Series Data at Optimally Placed Points”, Physics of Fluids, Vol. 33 (2021), 077105.

[2] Suzuki, K., Oshima, A., Hong, C. and Mochizuki, M., “Subcooled Flow Boiling in a Minichannel”, Heat Transfer Engineering, Vol. 32, No. 7-8 (2011), pp. 667-672.

[3] Zienkiewicz, O.C. and Taylor, R.C., The finite element method, 4th Edition, Vol. I (1989), McGraw Hill.

SCHLIEREN VISUALIZATION OF EXTINGUISHING PROCESS OF DIFFUSION FLAMES IN AIRBURST BLAST EXTINGUISHMENT WITH MICRO EXPLOSIVES

*A. Yuma Yoshida and B. Hiroyuki Torikai

Hirosaki University, Bunkyo Town 1, Hirosaki, Aomori 036-8560 Japan
*A. h22ms543@hirosaki-u.ac.jp and B.torikai@hirosaki-u.ac.jp

ABSTRACT

A blast extinguishment is one of methods to extinguish a fire, which basically consists of a diffusion flame, using a blast waves produced by detonation of explosives and a flow of gaseous detonation products. Using the method, we can put out intense fires instantaneously and it controls the extinguishing effective range by adjusting the amount of input energy to the explosion. Therefore, it can be said that it is used as emergency mitigation measures for large forest fires and post-earthquake fires. The authors have conducted experimental studies of airburst blast extinguishment on a laboratory scale [1-3]. In airburst, the blast wave is reflected on the ground. When the reflected blast wave merges with the incident shock wave to form a single wave, which is known as the Mach Stem. Few studies have focused on elucidation of relationship between the extinguishing effectiveness of the airburst blast extinguishment and the Mach stem.

In this study, in order to clarify the blowoff characteristics of airburst blast extinguishment, we have conducted laboratory-scale experiments to blast off a methane-air jet diffusion flame using 10 mg silver azide pellets. In addition, to reveal the extinguishing effect of the Mach stem, the extinguishing processes have been investigated with a schlieren visualization method.

From the experimental results, we show the existence of the optimal height of burst for extinguishing the jet diffusion flame. At the optimal height of burst, the Mach stem is formed just before the incident blast wave interacts with the diffusion flame. Thus, the formation of the Mach stem is considered to play an important role in airburst blast extinguishment.

REFERENCES

[1] Torikai, H., "Basic research on blast extinguishment", Journal of JAFSE "KASAI", Vol. 67, No. 6 (2017), pp. 14-19 (in Japanese).

[2] Torikai, H., Soga, Y., and Ito, A., "Schlieren visualization of blast extinguishment with laser-induced breakdown", Proceedings of the Combustion Institute, Vol. 36, No. 2 (2017), pp. 3297-3304.

[3] Sekikawa, R. and Torikai, H., "Blast extinguishment of a methane-air jet diffusion flame using a silver azide pellet", Science and Technology of Energetic Materials, Vol. 80, No. 1 (2019), pp. 12-14.

A CRYOGENIC TEMPERATURE SENSITIVE PAINT MEASUREMENT USING METAL COMPLEX MATERIALS AND PHOSPHORS ~PLELIMINARY EXPERIMENTS~

S. Yasuzawa[1,2], *S. Someya[2], Y.R. Li[1] , S. Saito[2], S. Baba[2] and N. Takada[2]

[1] Graduate School of Sci. and Eng., Ibaraki Univ., 4-12-1 Nakanarusawa, Hitachi 316-8511
[2] AIST, 1-2-1 Namiki, Tsukuba 305-8564 Japan
* s.someya@aist.go.jp

ABSTRACT

Thermal control, an effective utilization of unused heat, is important for ensuring the efficiency of sophisticated mechanical systems. The measurement of temperature and heat flux is indispensable for realizing advanced thermal control. An effective use and a control of a cold thermal energy of the liquid natural gas or the liquid hydrogen is essential for reducing greenhouse gas emissions and for achievement of carbon neutral society. However, temperature measurement methods applicable to such cryogenic system are limited, e.g., using an infrared camera in a low temperature condition is difficult because of the low IR emission. A commercial IR camera can be used at temperature over -40 degrees.

We investigated temperature sensitivity of many kinds of organic / inorganic luminescent materials and clarified several temperature sensitive materials. In this presentation, we show a demonstration of transient temperature measurement of a low temperature copper surface heated by a pulsed electro-heater. The copper plate was cooled by using liquid nitrogen and its temperature was kept at -90 degrees before the heating. Then, a part of the copper plate was heated during 10ms using a nichrome wire heater.

Acknowledgment

This work was supported by JSPS KAKENHI Grant Number JP20H02073.

REFERENCES

[1] Someya, S., “Particle-based temperature measurement coupled with velocity measurement”, Meas. Sci. Technol. 32 (2021) 042001.

[2] Kim, D., Kim, M. and Kim, K. C., “Effect of frost on phosphorescence for thermographic phosphor thermometry”, Meas. Sci. Technol. 28 (2017) 125202.

[3] Egami, Y., Fey, U., Klein, C., Quest, J., Ondrus, V. and Beifuss, U., “Development of new two-component temperature-sensitive paint (TSP) for cryogenic testing”, Meas. Sci. Technol. 23 (2012) 115301.

A CRYOGENIC TEMPERATURE SENSITIVE PAINT MEASUREMENT USING METAL COMPLEX MATERIALS AND PHOSPHORS ~TEMPERATURE SENSITIVITY OF OPTICAL PROPERTIES ~

Y.R. Li[1] , S. Yasuzawa[1,2], *S. Someya[2], S. Saito[2], S. Baba[2] and N. Takada[2]

[1] Graduate School of Sci. and Eng., Ibaraki Univ., 4-12-1 Nakanarusawa, Hitachi 316-8511
[2] AIST, 1-2-1 Namiki, Tsukuba 305-8564 Japan
* s.someya@aist.go.jp

ABSTRACT

An effective use of a wasted cold thermal energy of the liquid natural gas or the liquid hydrogen is essential for reducing greenhouse gas emissions and for achievement of Net-zero CO_2 society. The measurement of temperature and heat flux is indispensable for realizing advanced thermal control. However, there is no way to measure the temperature two-dimensionally at the low temperature. Using an infrared camera in a low temperature condition is difficult because of the low IR emission. A commercial IR camera can be used at temperature over -40 degrees.

One of the other approaches of low temperature measurement is the temperature sensitive paint method, however, the information of temperature sensitive luminophore is quite limited. Therefore, in this paper, we investigated the optical properties of various organic / inorganic luminescent materials at low temperature conditions. The luminescent intensity, the shape of emission spectra (color) and the luminescent lifetime at various cryogenic conditions from -190 degrees to the room temperature were evaluated by using a photomultiplier tube and a spectorometer. The temperature sensitivity of optical properties were summarized.

Acknowledgment

This work was supported by JSPS KAKENHI Grant Number JP20H02073.

REFERENCES

[1] Someya, S., “Particle-based temperature measurement coupled with velocity measurement”, Meas. Sci. Technol. 32 (2021) 042001.

[2] Kim, D., Kim, M. and Kim, K. C., “Effect of frost on phosphorescence for thermographic phosphor thermometry”, Meas. Sci. Technol. 28 (2017) 125202.

[3] Egami, Y., Fey, U., Klein, C., Quest, J., Ondrus, V. and Beifuss, U., “Development of new two-component temperature-sensitive paint (TSP) for cryogenic testing”, Meas. Sci. Technol. 23 (2012) 115301.

IMPROVEMENT OF SPATIAL RESOLUTION OF HIGH-SPEED-BACKGROUND-ORIENTED SCHLIEEN THROUGH ATOMOSPHERIC TURBULENCE

*T. Mizukaki[1], M. Kondo[1], and T. Minezaki[2]

[1] Tokai University, 4-1-1 Kitakaname, Hiratsuka, Kanagawa 259-1292 Japan
*mzkk@tsc.u-tokai.ac.jp
[2]Tokyo University, 2-21-1, Osawa, Mitaka, Tokyo 181-0015 Japan

ABSTRACT

Background-oriented schlieren (BOS) method has been applied for flow visualization for both in-door and out-door experiment on aerodynamic phenomena to obtain quantitative density distribution around objects since Raffel[1] proposed in 2000. The BOS provides spatially integrated information on density changes around objects as same as ordinally schlieren method. On the other hand, one of advantages of the BOS against ordinally schlieren is less limitation of visualization area by optics because schlieren mirrors are not always needed. Therefore, the BOS has advantage for flow visualization on real-size phenomena such as flow field around a real aircraft.

For application of the BOS for flow visualization around a real aircraft, one big problem be solved is improvement of distortion of images by atmospheric turbulence. Even one uses large magnification image system, the atmospheric turbulence would prohibit high spatial resolution in out-door because atmospheric turbulence must be strong near ground level.

In this paper, the authors applied adaptive optics[2] for improvement of spatial resolution of visualized images by the BOS. The adaptive optics includes a telescope, a Shack-Hartman wavefront censer, and a MEMS-type deformable mirror. Laser-induced shock waves were visualized by the BOS though turbulence generated by a heated steel plate. Obtained images were determined its displacement from reference one with cross-correlation function. Numerical analysis was also involved to evaluate the experimental results.

The experimental results showed that adaptive optics can provide improvement of spatial resolution through atmospheric turbulence more than twice better than the case without it.

REFERENCES

[1] Raffel, M., Richard, H., and Meier, G. E. A. "On the applicability of background oriented optical tomography for large scale aerodynamic investigations," *Experiments in Fluids* Vol. 28, No. 5, 2000, pp. 477-481.
[2] Hardy, J. W. "Active optics: A new technology for the control of light," *Proceedings of the IEEE* Vol. 66, No. 6, 1978, pp. 651-697.

VISUALIZATION STUDY ON DROPLET ELECTROHYDRODYNAMIC DEFORMATION IN COMBINED DC ELECTRIC FIELD AND SHEAR FLOW

Guanqiu Hao[1], Xiangdong Liu[1,2] and *Yongping Chen[1,2,3]

[1] Key Laboratory of Energy Thermal Conversion and Control of Ministry of Education, School of Energy and Environment, Southeast University, Nanjing, Jiangsu 210096, PR China
*ypchen@seu.edu.cn

[2] School of Electrical, Energy and Power Engineering, Yangzhou University, Yangzhou, Jiangsu 225009, PR China

[3] Jiangsu Key Laboratory of Micro and Nano Heat Fluid Flow Technology and Energy Application, School of Environmental Science and Engineering, Suzhou University of Science and Technology, Suzhou, Jiangsu 215009, P. R. China

ABSTRACT

A visualization experiment is conducted to investigate electrohydrodynamic deformation of droplet in combined DC electric field and shear flow. The detailed experimental data about both transient and steady droplet deformation parameter (D) and orientation (ϕ_d) are provided at $R>S$ and $R<S$ (R: conductivity ratio; S: permittivity ratio) under different combinations of electric field and shear flow. The internal flow characteristics of the deformed droplet are also examined via the DPIV method. It is indicated that, owing to the competition of the extensional component (EC) and the rotational component (RC) on the droplet from these two fields, the response of ϕ_d is faster than that of D when the electric field is combined with the shear flow. Also under different competition of the EC and RC under $R>S$ and $R<S$, the steady D and ϕ_d show distinct variation. Especially, surface charge convection plays a non-ignorable role in enhancing and reducing droplet deformation for $R>S$ and $R<S$, respectively. In addition, an asymmetric vortex occurs inside the deformed droplet in the combined two filed, and its velocity is reduced under $R>S$ and enhanced under $R<S$ when compared with those in the pure shear flow. The available prediction models are demonstrated by the experimental data to predict D, and a modified prediction model is proposed to improve the prediction accuracy of ϕ_d.

DEVELOPMENT OF TEMPERATURE MEASUREMENT SYSTEM OF AIRFLOW USING ULTRA-FINE THERMOSENSITIVE LUMISIS PHOSPHOR WIRE

*Yu Morito[1], Shumpei Funatani[1]

[1] University of Yamanashi, 4-3-11 Takeda, Kofu, Yamanashi 400-8511, Japan
*g21tm033@yamanashi.ac.jp

ABSTRACT

The measurement of atmospheric temperature of gases is difficult due to the measurement process, and new measurement methods are required by the industry. For the measurement of hot water in a tank, a temperature measurement method using the two-color LIF method is known [1].

The authors have proposed a method for simultaneous measurement of temperature and velocity fields using the two-color LIF method and the PIV method with a single color camera by optimizing the type and concentration ratio of fluorescent dyes [2].
The LIF method is a method to observe the fluorescence of a specific fluorophore by using a laser beam. Using this method, ultra-fine fluorescent wires coated with a temperature-dependent phosphor are placed in the measurement area and visualized by a camera to measure the temperature distribution of gases. The measurement system is relatively inexpensive because the optical system is maintenance-free and the imaging system is a single CCD camera of a common RGB type. The ultra-thin wire enables non-contact measurement without affecting the flow field.

In the previous study, it was found that the two-wavelength spectroscopy using Rhodamine B has a large measurement error due to the slight temperature dependence of the center wavelength of Rhodamine B. In order to solve this problem, the temperature dependence of Rhodamine B was investigated using a luminescent material. Therefore, we evaluate the temperature dependence of Lumisis phosphors and propose a two-color LIF airflow temperature distribution measurement method using a wire with two types of Lumisis phosphors, and evaluate the accuracy of the temperature measurement.

The validation results confirmed the usefulness of the temperature distribution measurement using temperature-dependent phosphors, and the measurement error was significantly reduced by the temperature distribution measurement method using lumisis phosphors.

REFERENCES

[1] Sakakibara,J. and Adrian,R.J.: Whole field measurement of temperature in water using two-colour laser induced fluorescence, Exp.Fluids, Vol.26, (1999) pp.7-15.

[2] Funatani, S., Fujisawa, N. and Ikeda,H.: Simultaneous measurement of temperature and velocity using two-colour LIF combined with PIV with a colour CCD camera and its appliccation to the turbulent buoyant plume, Meas. Sci. and Tech., Vol.15, (2004) pp.983-990.

VISUALIZATION OF MICROBIAL TRANSPORT IN THE GUT OF A ZEBRAFISH LARVA

*Y. KIKUNAGA[1], K. KIKUCHI[1,2], K. NUMAYAMA-TSURUTA[1] and T. ISHIKAWA[1,2]

[1]Department of Biomedical Engineering, Tohoku University
6-6-01 Aoba, Aramaki, Aoba-ku 980-8579 Japan
*kikunaga.yuta.r5@dc.tohoku.ac.jp
[2] Department of Engineering, Tohoku University
6-6-01 Aoba, Aramaki, Aoba-ku 980-8579 Japan

ABSTRACT

There are more than 100 trillion gut bacteria in our guts, which are involved in the breakdown, digestion, and absorption of food. Also, they have a significant impact on various diseases. However, it is not clear how gut bacteria affect these diseases spatially and temporally, and no remedial measure has been found for diseases in which gut bacteria is the main cause. Zebrafish have recently been used as an animal model for human disease because of high genetic homology to humans. A particular advantage of zebrafish is that zebrafish larvae have transparent body, making *in vivo* imaging easy. Thus, they are a good model for intestinal studies. In this study, therefore, to elucidate the relationship between intestinal flow and distribution of gut bacteria under intestinal peristalsis, we used zebrafish larvae, and we performed visualization measurements under chemically controlled intestinal peristalsis. To control intestinal peristalsis, Zebrafish larvae were immersed in water dissolved with a drug that is either a peristaltic promoter or inhibitor. Then, the fluorescent dye was orally injected into the zebrafish larva, and the specimen was observed under a fluorescence microscope. Obtained fluorescence images were analyzed by a software Fiji. The frequencies of peristalsis in the posterior and anterior intestines were measured because the temporal mechanical functions differ between the posterior and anterior intestines. The frequency of peristalsis was promoted by acetylcholine and inhibited by atropine in both the posterior and anterior intestines. However, we observed that the effective duration of the promoters and inhibitors differed between the posterior and anterior intestines. This may affect the intestinal peristalsis and the distribution of the gut bacteria in a zebrafish larva. This study could be essential for future studies on the interaction between gut bacteria and intestinal peristalsis.

REFERENCES

[1] P.J. Turnbaugh, R.E. Ley, M.A. Mahowald, V. Magrini, E.R. Mardis and J.I. Gordon, “An obesity-associated gut microbiome with increased capacity for energy harvest”. *Nature.* **444**, (2006), pp. 1027-1031.

[2] Q. He, L. Wang, F. Wang, and Q. Li, “Role of Gut Microbiota in a Zebrafish Model with Chemically Induced Enterocolitis Involving Toll-Like Receptor Signaling Pathways”. *Zebrafish,* Vol. 11 (2014), pp. 255-264.

[3] J. L. Cocchiaroand and J. F. Rawls, “Microgavage of Zebrafish Larvae”. *J. Vis. Exp.* **72**, e4434, (2013), pp. 1-12.

FUNDAMENTAL RESEARCH ON COMBINATION OF SOLID-LIQUID LEVEL MEASUREMENT AND ELEMENTAL COMPOSITION ANALYSIS USING ULTRASOUND AND LASER

* Y. CHEN[1], N. SHOJI[2], Z. ZHANG[1], T. V. TRAN[2], H. TAKAHASHI[2] and H. KIKURA[2]

[1] Tokyo Institute of Technology, School of Environment and Social Science, Department of Transdisciplinary Science and Engineering, Nuclear Engineering Major, 2-12-1 Ookayama, Meguro-Ku, Tokyo, Japan 152-8550
* chen.y.bm@m.titech.ac.jp
[2] Tokyo Institute of Technology, Institute of Innovative Research, Zero Carbon Energy Laboratory, 2-12-1 Ookayama, Meguro-Ku, Tokyo, Japan 152-8550

ABSTRACT

Since the accident at the Fukushima Daiichi Nuclear Power Plant (1F NPP) on March 11, 2011, a decommissioning of the 1F NPP is underway. Data on fuel debris distribution and chemical properties of fission products (FPs) in primary containment vessels (PCVs) will provide valuable information for planning fuel debris removal and accident analysis. Conventionally, small samples are collected by a robot and transported to an analysis center for elemental analysis. However, it takes a long time and is impractical to analyze the entire PCVs. Meanwhile, a remote survey has been conducted using robots with optical cameras. However, the images have a narrow field of view due to the dark environment in PCVs. [1]

Therefore, this study focused on a laser induced breakdown spectroscopy (LIBS), which enables in-situ elemental analysis, and constructed a remote LIBS system using a microchip laser [2] to mount the system on a robot. However, the accuracy of LIBS measurement decreases when the object's surface is far from the laser irradiation focus, it is necessary to adjust the distance between the focus position and the object. In addition, In the removal of fuel debris, the process of adjusting the water level of the cooling water accumulated by fuel debris and removing the exposed fuel debris with a mechanical arm is planned. According to the height of fuel fragments, the best manipulator is different. Therefore, it is required to develop a simultaneous measurement method for the location of fuel debris and water level.

Therefore, this study aims to construct an elemental mapping system that can measure the solid-liquid level while performing elemental analysis. In this paper, the constructed system is described, and an integration experiment was conducted. Through the experiment, the element distribution is successfully mapped based on the shape reconstruction result. The experiment results also show that the integration between LIBS and ultrasonic measurement could improve the focus of the LIBS system and could make the ultrasonic shape measurement more accurate.

REFERENCES

[1] Tokyo Electric Power Company Holdings, Inc .:About the investigation inside the Unit 1 containment vessel, Decommissioning / Contaminated Water Countermeasure Team Meeting 44th Secretariat Meeting, Tokyo (2017) pp.1-17.
[2] Koji, T., Ryuzo, N., Hironori, O., Takunori, T., Ikuo. W.: Recovery of the laser-induced breakdown spectroscopy system using a ceramic microchip deteriorated by radiation for the remote elemental analysis, Journal of Nuclear Science and Technology, Vol.57, No.10 (2022) pp.1189-1198.

REMOTE LEAKAGE FLOW VISUALIZATION WITH ARM-TYPE ROBOT AND THREE-DIMENSIONAL VECTOR UVP

*N. Shoji[1], T. Moriya[1], Y. Chen[1], H. Takahashi[1] and H. Kikura[1]

[1] Laboratory for Zero-Carbon Energy, Tokyo Institute of Technology, 1-12-1 Ookayama, Meguro-ku, Tokyo 152-8550 Japan
*shoji@us.nr.titech.ac.jp

ABSTRACT

On March 11th, 2011, the severe accident of the Fukushima Dai-ichi nuclear power plant (1F NPP) was happened by the earth quack and the massive tsunami in Tohoku area, Japan. And then, fuel debris was generated in the primary containment vessels (PCVs) of units 1, 2, and 3, respectively. Fuel debris removal is necessary for the accident restoration, but contaminated coolant water leakage is an issue, and identification of its location is required.

Ultrasonic measurement techniques are promising as a method that can overcome the environment within the PCVs (darkness, high-radioactive, muddy water). Ultrasonic sound can be used in opaque liquids and ultrasonic transducers are generally suited to high radiation levels, as used in the decommissioning of Three Mile Island NPP (TMI-2) [1]. In our previous work, we have suggested an idea to identify the leak locations by monitoring the coolant water flow conditions (velocity and direction of the flow) [2] and focused on an ultrasonic velocity profiler (UVP) [3] as a flow measurement method that can be adapted to the environment in PCVs.

The UVP can measure the instantaneous velocity profile along on the ultrasonic propagation path, and the original UVP is basically one-dimensional velocity component measurement. However, real flow forms three-dimensional (3-D) flow and expected to be complex flow structure due to interferences of objects inside the PCVs. Therefore, a measurement system is required to extend to 3-D flow measurement. In addition, the environmental within PCVs are under high radiation. Thus, humans cannot access to directory, and a remote measurement by a robot is necessary.

The purpose of this study is development of the remote 3-D flow vector measurement system combined with the robot. To extend the UVP to 3-D vector measurement, ultrasonic transducer array was constructed by using four transducers, and signal processing method of vector reconstruction was developed. As the robot, we constructed an arm-type robot, and the ultrasonic transducer array was mounted on the arm tip. Moreover, this system applied to a simulated leakage flow to confirm the validity for identification of leakage point of 1F NPP.

REFERENCES

[1] Beller, L. S., “Design and operation of the core topography data acquisition system for TMI-2”, Three Mile Island Reports, GEND-INF-012 (1984).

[2] Nishiwaki, R., Hamdani, A., Takahashi, H., Endo, G., Kikura, H., “Development of a Remote Water Leakage Localization System Combined with Phased Array UVP and Robot”, Proceedings of 11th International Symposium on Ultrasonic Doppler Methods for Fluid Mechanics and Fluid Engineering (ISUD11), (2018), pp. 88-91.

[3] Takeda, Y., “Development of an ultrasound velocity profile monitor”, Nuclear Engineering and Design, Vol. 126 (1991), pp. 277-284.

VISUALIZATION OF FUEL DEBRIS SHAPE AND MEASUREMENT OF WATER LEVEL USING PARAMETRIC ULTRASOUND

*T.V. Tran[1], H. Tanabe[1], H. Takahashi[1] and H. Kikura[1]

[1] Tokyo Institute of Technology, 2-12-1, N1-7 Ookayama, Meguro-ku, Tokyo, 152-8550 Japan
*tran.t.ae@m.titech.ac.jp

ABSTRACT

Estimation of the position and shape of fuel debris is one of the most important missions in decommissioning the Fukushima Daiichi Nuclear Power Plant. Investigations using ultrasound have been considered recently due to its advantages to working in cloudy, dark, and high radiation environments of nuclear reactors. However, a limitation of the ultrasound method is that the ultrasonic wave intensity is attenuated by the distance to the reflected objects. To overcome this limitation, in this study, the parametric sound was applied to measure the distance and the shape behavior of a simplified sample that simulated nuclear fuel debris. The parametric sound is characterized by higher directivity and lower attenuation; thus, it is expected to measure the shape of debris from longer distances. In this paper, the shape of fuel debris was visualized by reconstructing the distance measurement data using parametric ultrasound signals. In addition, a new algorithm was developed to discriminate between the solid/liquid states and measure the level of the water surrounding the debris.

REFERENCES

[1] Endo G., Takahashi, H. & Kikura, H., "Challenge to investigation of fuel debris in RPV by an advanced Super Dragon articulated robot arm (2) design and prototyping of a lightweight super long reach articulated manipulator". 2020 International Conference on Nuclear Engineering collocated with the ASME 2020 Power Conference, (2020).

[2] Takahashi, H., Shoji, N., Ito, A. and Kikura, H., "Fundamental Study on Ultrasound Sensing Technology using Parametric Sound", WIT Transactions on Engineering Sciences, Vol. 12, (2020), pp. 103-111.

RANGE EXTENSION ON ULTRASONIC MEASUREMENT OF BUBBLY FLOW VELOCITY PROFILE

*W. Wongsaroj[1], H. Takahashi[2], N. Thong-un[1], W. Treenuson[3] and H. Kikura[2]

[1] King Mongkut's University of Technology North Bangkok, 1518 Pracharat 1 Road, Wongsawang, Bangsue, Bangkok 10800 Thailand
*wongsakorn.w@eng.kmutnb.ac.th
[2] Tokyo Institute of Technology, 2-12-1 Ookayama, Meguro, Tokyo 152-8550, Japan
[3] Ministry of Higher Education, Science, Research and Innovation, 16 Vibhavadi Rangsit Road Bangkok 10900, Thailand

ABSTRACT

Two-phase bubbly flow occurs in many industrial sectors, especially in the boiling water reactor (BWR). It influences the operation of the reactor. The reactor can be operated safely if the behavior of bubbly flow is understood clearly. Experimental work is required to clarify flow behavior. Generally, the experiment has been executed on the channel or pipe. The bubbly flow velocity of bubble and liquid phases is an important parameter affecting reactor safety. Hence, this parameter is necessary to be obtained accurately. The Ultrasonic Velocity Profiler (UVP) method [1] is applicable to obtain the velocity distribution in fluid flow. It has been applied in several applications [2-3]. Then, the UVP has been proposed for measuring bubble and liquid velocity on the bubbly flow [4]. However, in the normal operation of BWR, the velocity value of both phases on the reactor core is about 2-3 m/s. This range is higher than the maximum velocity (Nyquist velocity) of the UVP in case of the incident angle at 45° degrees, in which the transmission ratio is high, has maximum sensitivity, and has no two-wave mode. Therefore, the extension of the measurement range is a necessity. This paper presents the range extension of the ultrasonic velocity profiler (UVP) to measure the flow velocity profile of bubbly flow beyond Nyquist velocity level. The single transducer is utilized for the measurement. The phase separation algorithm based on the integration of time-frequency analysis (TFA) and Doppler amplitude classification (DAC) collaborates with the cross-correlation function and is applied to achieve the target. In order to confirm the ability of the proposed technique, the experiment at bubbly flow velocity beyond Nyquist level in vertical pipe flow apparatus was conducted. The measurement result obtained was able to guarantee the applicability of the proposed technique to extend the measurement range.

REFERENCES

[1] Takeda, Y., "Velocity Profile Measurement by Ultrasonic Doppler Shift Method," International Journal of Heat and Fluid Flow, Vol. 7 (1986), pp. 313-318.

[2] Kikura, H., Takeda, Y.and Durst, F., " Velocity Profile Measurement of the Taylor Vortex Flow of a Magnetic Fluid using the Ultrasonic Doppler Method" Experiments in Fluids, Vol. 26 (1999), pp. 208-214.

[3] Wada, S., Kikura, H., Aritomi, M., Mori, M. and Takeda, Y., " Development of Pulsed Ultrasonic Doppler Method for Flow Rate Measurement in Power Plant ; Multilines Flow Rate Measurement on Metal Pipe" Journal of Nuclear Science and Technology, Vol. 41 (2004), pp. 339-346.

[4] Wongsaroj, W., Hamdani, A., Thong-Un, N., Takahashi, H. and Kikura, H., "Extended Short-Time Fourier Transform for Ultrasonic Velocity Profiler on Two-Phase Bubbly Flow Using a Single Resonant Frequency". Applied Sciences, Vol. 9 (2019), 5

MEASUREMENT OF MULTIPHASE FLOW IN ROTATING FIELD BY ULTRASONIC VELOCITY PROFILER

*Takumi KONNO[1], Yoshihiko OISHI[1], Hideki KAWAI[1], Sho OSANAI[1]

[1] Muroran Institute of Technology, 27-1 Mizumoto-cho, Muroran, Hokkaido 050-8585 Japan
*oishi@mmm.muroran-it.ac.jp

ABSTRACT

Taylor-Couette flow (TCF) has the most important and interesting characteristics in fluid physics and engineering. TCF is generated from spinning inner cylinder in gap between inner and outer cylinders of concentric circle. TCF is more reliable as a mixing system than the previous mixing methods such as a magnetic stirrer and a mechanical stirrer. In a culture technique using a bioreactor, the commercialized stirrer is not applicable to the bioreactor because the microorganisms are weak to the shear stress. The bioreactor by TCF is expected to be performed in low shear flow in order to prevent physical disruption of microorganic cells. Concentration and location of the microorganisms are required to measure for control of the cultivation accurately. This study shows the experimental results by an ultrasonic measurement in multiphase flows with the TCF with small aspect ratio [1]. In particular, we investigated three flow fields: isolated mixing phenomenon in particle-dispersed solid-liquid two-phase flow, W/O emulsion in droplet-dispersed liquid-liquid flow and microbial dispersion-based solid-liquid two-phase flow.

The microorganisms behave like particles of poly-dispersion in TCF. Therefore, an isolated mixing region (IMR) exists during low-Reynolds-number TCF in solid-liquid two-phase flow [2]. The flow in IMR causes a program of inhomogeneous particle-location while stirring. As a fundamental study, we investigated the detection of concentration of particle in low-Reynolds-number TCF using the echo variation caused by attenuation of ultrasonic energy. In the droplet-dispersed liquid-liquid flow, we evaluated W/O emulsified fuels using ultrasonic measurement, produced 10 %, 20 %, 30 % water-vegetable oil emulsion at 300-800 rpm with a Taylor-Couette reactor [3]. We measured velocity profiles and attenuation of echo at every 100 rpm utilizing the ultrasonic Doppler method and echo intensity measurement. As a result, transitions of the velocity profiles and the echo attenuation were different in each water content ratio. Finally, we performed UVP measurement on the flow of photosynthetic microorganisms and found the possibility of measurement.

REFERENCES

[1] Fujii, T., Oishi, Y., Kawai, H., Kikura, H., Situmorang, R.S. and Ambarita, H., “Frequency Analysis of Chaotic Flow in Transition to Turbulence in Taylor-Couette System with Small Aspect Ratio”, Journal of Physics: Conf. Series, Vol.801 (2017) 012063.

[2] Kato, Y., Oishi, Y., Kawai, H. and Kikura, H., “Measurement of particle concentration and location in Taylor-Couette flow by ultrasound echo intensity”, Proceedings of EPAM2016, (2016), pp. 174-175.

[3] Oishi, Y. Kawai, H., Sasayama, D. and Kikura, H., “Frequency Analysis of Water-Vegetable Oil Emulsion Produced by Taylor-Couette System with Small Aspect Ratio using Ultrasonic Doppler and Attenuation Method”, Proc. the 13th Int. Symp. Ultr. Doppler Meth. Fluid Mech. Fluid Eng., (2021), pp. 90-93.

EVOLVING ULTRASONIC VELOCITY PROFILER and eUVP

Y. TAKEDA[1], E. WINDAHB[2] and H. KIKURA[3]

[1] Faculty of Engineering, Hokkaido University, N13W8, Sapporo, Hokkaido, 060-8628, Japan

[2] Laboratory for Food Process Engineering, Dep. Health Science and Technology, ETH Zürich, Switzerland

[3] Laboratory for Advanced Nuclear Energy, Tokyo Institute of Technology, 2-12-1 Ookayama, Meguro, Tokyo 152-8550, Japan

ABSTRACT

UVP (Ultrasonic Velocity Profiler) has been established as a tool in experimental fluid mechanics as wll as in food engineering as UVP-PD method. Since it has expanded a flow measurement capability from a simple velocity profiling to obtaining flow related characteristics of flowing media, the method is still evolving to a wider range of flow measurement to plant controlling components. This evolution is introduced. One of the further implications of evolution is to change the basic knowledge-gathering methodology from point measurement in space or time to linear or areal measurement of time-dependent velocity distribution. This is effectively identical to obtaining a solution of basic equation such as NS equation – without solving the equation. The mind for such basic approach to the problem solution is yet oldish and classical in general, so that it is expected that new idea must be given to new-comers or young students and researchers. A development of eUVP (educational UVP) is an example of such effort at ETHZ with TIT, and experience of introduction to the class lecture will be presented.

REFERENCES

[1] U. Frisch & S. A. Orszag, “Turbulence: Challenges for theory and experiment”, Physics Today, January 1990, p24.

PSFVIP13

secretary@psfvip13.org
https://psfvip13.org