IMAT ICSERA 2021

The 12TH International Meeting on Advances in Thermofluids (IMAT)

1ST NOVEMBER 2021

MONDAY | 8.00 A.M - 6.00 P.M

UNIVERSITI TEKNOLOGI MALAYSIA

Airmeet

In collaboration with

UNIVERSITAS INDONESIA

EVERGREEN

MESSAGE FROM THE IMAT 2021 CHAIRMAN

PROF. DR. MAZLAN ABDUL WAHID
School of Mechanical Engineering
Faculty of Engineering
Universiti Teknologi Malaysia

Welcome all participants to the 12th International Meeting on Advances in Thermofluids (IMAT 2021). The School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia is honoured to host this IMAT 2021 in UTM Johor Bahru in collaboration with Universitas Indonesia and Evergreen Journal.

While we regret that the Covid-19 pandemic has prevented us from holding the physical conference here in Malaysia, but at the same time we are very excited about the opportunities to hold the first innovative virtual conference on the 1st of November 2021. With a total of 84 papers that have been accepted from several countries, all papers will be published by American Institute of Physics (AIP) and selected papers will be published in Evergreen Journal and Journal of Thermal Analysis and Calorimetry (JTAC) based on the standard review process. Sincerest appreciation goes to the distinguished plenary and invited speakers who have shared their knowledge and experience.

I am very much thankful for the cooperation and assistance of our academic staffs at School of Mechanical Engineering, Universiti Teknologi Malaysia, working with me as a team, in making this 12th IMAT a success. I am also grateful towards our partner universities, notably Universitas Indonesia, King Abdullah University of Science and Technology, without which there would have been no IMAT to begin with. Thank you to Professor Muhammad Idrus Alhamid and his team, Professor Kim Choon Ng and his team, Professor Kyaw Thu from Kyushu University as well as the Evergreen Journal team in making sure the success of this conference. We wish to provide the most pleasurable time to all participant to exchange ideas and discuss about new researches and applications related to thermal and fluids areas, thus provide an opportunity for the communication and cooperation among researchers.

Thank you

MESSAGE FROM THE CHAIR OF SCHOOL OF MECHANICAL ENGINEERING

PROF. IR. DR. ZAINI AHMAD
School of Mechanical Engineering
Faculty of Engineering
Universiti Teknologi Malaysia

Bismillahirrahmanirrahim & Assalamualaikum wrm.wbt.,

It is my pleasure to welcome all the participant and distinguished speakers to the 12th International Meeting on Advances in Thermofluids (IMAT) and the 7th International Conference on Saving Energy for Refrigeration and Air-Conditioning (ICSERA).

Welcome to School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia.

First of all, I would like to express my appreciation for the efforts of IMAT and ICSERA organizing committee; the Thermofluid Programme's academic staff from School of Mechanical Engineering, UTM on making these virtual conferences a success. IMAT2021 was organized with the collaboration of Universitas Indonesia and Evergreen Journal of Kyushu University. Meanwhile, ICSERA2021 is with the collaboration of Society of Air-conditioning and Refrigerating Engineers of Korea (SAREK). School of Mechanical Engineering, UTM is very honored to have this kind of international collaboration and hope it will stay strong in the future.

IMAT and ICSERA was initially planned as a physical conference, however due to the COVID19 pandemic, the conference mode has been changed to a virtual conference. In spite of the virtual conference mode, IMAT and ICSERA still received a very good response. There are eighty-four (84) technical papers have been accepted for presentation at IMAT and fifty-six (56) technical papers have been accepted for presentations at ICSERA. Both of the conferences have received participation from all over the world. Majority of the participants are from Malaysia, Indonesia and South Korea. In addition, both of the conferences have also received participation from all over the world such as from Japan, United Kingdom, Saudi Arabia, India, Libya, Oman, Tunisia and Nigeria. Therefore, I would like to congratulate IMAT and ICSERA participants for your effort and participation.

I look forward for IMAT and ICSERA to be a platform for the exchange of ideas, discussion and insights throughout all the 11 virtual parallel sessions. I hope all of the participants will leverage on the conference as a valuable platform to share information, discuss new ideas and to initiate collaboration especially in addressing the challenges ahead in the field of thermofluids, air-conditioning and refrigeration. I wish all of you a successful conference and productive day ahead.

Thank you.

IMAT 2021 ORGANIZING COMMITTEES

CHAIRMAN
Prof. Dr. Mazlan Abdul Wahid

DEPUTY CHAIR
Dr. Aminuddin Saat
Prof. Dr. Normah Mohd Ghazali

SECRETARIAT
Dr. Ummikalsom Abidin
Dr. Natrah Kamaruzzaman

TREASURER
Dr. Mohd Ibthisham Ardani
Dr. Muhammad Noor Afiq Witri Muhammad Yazid

FACILITIES COMMITTEE
Ir. Dr. Mohamad Nur Hidayat Mat
Dr. Faizal Hasan
Dr. Fazila Mohd Zawawi
Mr. Mohamad Nor Musa

TECHNICAL COMMITTEE
Dr. Mohd Fairus Mohd Yasin
Prof. Ir. Dr. Syahrullail Samion
Prof. Dr. Kahar Osman
Assoc. Prof. Ir. Dr. Hayati Abdullah
Assoc. Prof. Ts. Dr. Haslinda Mohamed Kamar
Dr. Zulkarnain Abdul Latiff
Dr. Norikhwan Hamzah

PUBLICITY / PROMOTION
Dr. Muhamad Hasbullah Padzillah
Dr. Ken K.Y. Wong
Dr. Md. Mizanur Rahman
Mr. Mohd Zubil Bahak
Mr. Mohd Kamal Arifin

IMAT 2021 INTERNATIONAL ADVISORY COMMITTEES

Prof. Dr. Mazlan Abdul Wahid, Universiti Teknologi Malaysia
Prof. Muhammad Idrus Alhamid, Universitas Indonesia, Indonesia
Prof. Dr. Ir. Bambang Sugiarto, Universitas Indonesia, Indonesia
Prof. Ir. Harinaldi, Universitas Indonesia, Indonesia
Prof. Ir. Yulianto Universitas Indonesia, Indonesia
Prof. Nasruddin, Universitas Indonesia, Indonesia
Prof. Agus Pamitran, Universitas Indonesia, Indonesia
Professor Kim Choon Ng, King Abdullah University of Science and Technology, Saudi Arabia
Prof. Muhammad Wakil Shahzad, Northumbria University, Newcastle, United Kingdom
Prof. Hwataik Han, Kookmin University, Korea
Prof. Kiyoshi Saito, Waseda University, Japan
Prof. Patrice Estellé, IUT Rennes France
Prof. Aya Hagishima, Kyushu University, Japan
Prof. Takahiko Miyazaki, Kyushu University, Japan
Prof. Byung-Koog Jang, Kyushu University, Japan
Prof. Kyaw Thu, Kyushu University, Japan
Prof. Wang Dong, Kyushu University, Japan
Prof. Chong Cheng Tung, Shanghai Jiao Tong University, China

IMAT 2021 PLENARY SPEAKER

PROF. SIMONE HOCHGREB

University of Cambridge

Simone Hochgreb is Professor of Engineering at the University of Cambridge. Her research involves understanding processes in combustion and reacting flows, as relevant to engines and gas turbines. She is known for methods and analysis of reacting flows in autoignition, laminar and turbulent flows, spray, and particle formation. Her current work is in the application of optical diagnostics to understanding turbulent flames, combustion instabilities, and flame and particle synthesis. Prior to Cambridge she held positions at MIT and Sandia National Labs. She has a PhD from Princeton University, and a BSME from the University of São Paulo. She has received the Wolfson Merit Award and the Society of Automotive Engineers Ralph R. Teetor Award, and has been a fellow of the Royal Aeronautical Society.

IMAT 2021 PLENARY SPEAKER

PROF. WANG RUZHU
Shanghai Jiao Tong University, China

Profesor Wang Ruzhu (R. Z. Wang), born in Dec.1964, graduated from Shanghai Jiao Tong University (SJTU) in 1984, 1987 and 1990 for his bachelor, master and PhD degrees. He was promoted as associate professor in 1992, full professor in 1994 at SJTU. He has written 10 Books and about 600 co-authored international journal papers and more than 100 patents, his ISI h index is 72. He is a leading scientist in heat pumps, CCHPs, solar heating and cooling, and green building energy systems. He has won Chinese National Research Awards in 2010 and 2014 respectively. Prof. Wang received the J & E Hall International Gold Medal from the Institute of Refrigeration (UK) in 2013, Asia Refrigeration Academic Award in 2017, the Nukiyama Memorial Award from the Japanese Society of Heat Transfer in 2018, the IIR-Gustav Lorentzen Medal from the International Institute of Refrigeration in 2019. He was honored to be the Clarivate Highly Cited Researcher in 2017 and 2018 respectively. He had been appointed as the director of Institute of Refrigeration and Cryogenics of SJTU since 1993. Currently he is also the Director- Engineering Research Center of Solar Energy, MOE China, Vice dean of SJTU Energy Institute. His research group has awarded as Excellent Innovative Team of Energy Research from MOST China in 2014 and NSFC in 2015. Prof. Wang is currently the Deputy editor-in-chief of Energy, Regional editor-International Journal of Refrigeration.

IMAT 2021 INVITED SPEAKER 1

PROF. AGUS PAMITRAN
Universitas Indonesia

Dr. Agus S. Pamitran is a lecturer and researcher in the Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia. His research has been focusing on heat transfer, pressure drop, and flow pattern of two-phase flow boiling, especially with natural refrigerants, in microchannel since 2002, and sea-water ice slurry generator since 2010. He completed his undergraduate program at the Mechanical Engineering program at the University of Indonesia in 1999. Then he continued his master and doctoral studies at the refrigeration engineering program at Chonnam National University, completed in 2009. After completing his doctoral studies he started his educational career as a lecturer at the Department of Mechanical Engineering. , Faculty of Engineering, University of Indonesia, until now. In 2012-2018 he was asked to lead the Marine Engineering and Naval Architecture Study Program, at the Department of Mechanical Engineering, Faculty of Engineering, University of Indonesia. Then from 2018 until the end of 2021 he was asked to be the secretary of the Mechanical Engineering department. Apart from teaching, he is also active in research, especially on the theme of refrigeration. His research on two-phase flow heat transfer and pressure drop of some refrigerants with microchannel has been started since his master and doctoral studies. At the University of Indonesia he continued his research on this theme, by doing several developments. In 2010 he started an active research with the theme of sea-water ice slurry generator. The main motivation for this ice slurry research is to help improve the quality of refrigerated catch, so as to improve the welfare of fishermen.

IMAT 2021 INVITED SPEAKER 2

PROF. KIM CHOON NG
King Abdullah University of Science & Technology (KAUST),
Saudi Arabia

Professor Ng's research has been focusing on the heat and mass transfer of thermally-driven systems such as solar thermal plants, thermal-hydraulics of nuclear (PWR) plants, cooling machines or chillers, heat pumps. In 2000, he research focus is in the areas of solar engineering, heat pumps, cooling machines and adsorption thermodynamics for solid-vapor interactions where he pioneered the adsorption cycles for seawater desalination and cooling (ADC and MEDAD cycles). The unique features of the thermally-driven cycles are (i) its ability to utilize low temperature waste heat for regeneration of adsorbents such as silica gel, zeolite, activated carbon etc., (ii) its ability to handle high concentration feed in the cycle, (iii) it has almost no major moving parts which makes the cycles robust and low in maintenance (operational cost). The ADC can be synergetically integrated with proven thermal cycles such as the MED/MSF of the desalination industry, boosting the water production by more than 2 folds with the conversion of waste heat for the ADC to produce useful cooling and potable water. The combined or hybridized cycle, called the MEDAD cycle, has one of the lowest specific energy consumption for seawater desalination, less than US$0.5 /m3. His publication in the area of ADC has more than two hundred papers and it forms a major part of his citations in the recent years. His current research H-index (google scholar) is 61 with more than 11894 citations. Dr Ng is associate editors to 3 journals and a professional engineer (PE) in Singapore where he serves as a member of the examination committee of PE Board.

IMAT 2021 INVITED SPEAKER 3

PROF. HASLINDA MOHAMED KAMAR
Universiti Teknologi Malaysia

Ts. Dr. Haslinda Mohamed Kamar is an Associate Professor in the School of Mechanical Engineering, Faculty of Engineering at Universiti Teknologi Malaysia. She has been a faculty member since 1993. Currently, she is the Head of Air-Conditioning Engineering Research Group (ACER) and an Associate Chair (Quality and Strategy) of the School of Mechanical Engineering, UTM. She received her bachelor's degree in Mechanical Engineering from the University of Glasgow, Scotland, in 1993, Master's and Ph.D. from Universiti Teknologi Malaysia in 1997 and 2009, respectively. Her areas of interest include automotive air-conditioning, thermal comfort & energy efficiency in hot climates, indoor air quality (IAQ), natural ventilation as a passive cooling strategy in buildings, and Computational Fluid Dynamics (CFD) modeling and simulation.

UTM Faculty of Engineering
UNIVERSITI TEKNOLOGI MALAYSIA

IMAT2021

The 12th International Meeting on Advances in Thermofluids (IMAT)
7th International Conference on Saving Energy for Refrigeration and Air-Conditioning (ICSERA)
1ST November 2021 | Universiti Teknologi Malaysia

UTM Faculty of Engineering
UNIVERSITI TEKNOLOGI MALAYSIA

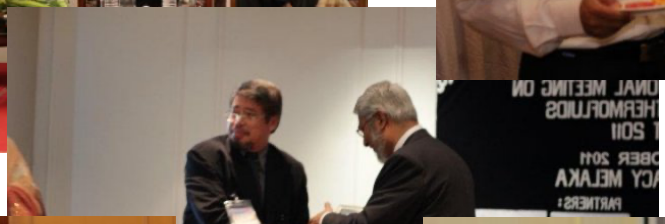
IMAT2021

The 12th International Meeting on Advances in Thermofluids (IMAT)
7th International Conference on Saving Energy for Refrigeration and Air-Conditioning (ICSERA)
1ST November 2021 | Universiti Teknologi Malaysia

UTM Faculty of Engineering
UNIVERSITI TEKNOLOGI MALAYSIA

IMAT2021

The 12th International Meeting on Advances in Thermofluids (IMAT)
7th International Conference on Saving Energy for Refrigeration and Air-Conditioning (ICSERA)
1ST November 2021 | Universiti Teknologi Malaysia



SCAN ME

YOU MAY GET TWO PEAS IN A POD
(Please click on the QR code)

IMAT 2021 PROGRAMME SCHEDULE

TIME	AIRMEET						
8:30 – 8:40	Housekeeping announcement & Welcoming remarks by Emcee						
8:40 – 8:45	Opening Speech by Chair of IMAT - Prof. Mazlan Abdul Wahid						
8:45 – 8:50	Speech by Honorary Chair of IMAT - Prof. Muhammad Idrus Alhamid						
8:50 – 8:55	Speech by Honorary Chair of IMAT - Prof. Kim Choon Ng						
8:55 - 9:00	Address by President of SAREK - Prof. Kim Min Soo						
9:00 – 9:10	Opening Speech by Chair of ICSERA - Prof. Normah Mohd Ghazali						
9:10 – 9:20	Officiating Speech by Chair of School of Mechanical Engineering - Prof. Ir. Zaini Ahmad						
9:20 – 9:25	UTM – montage						
9:30 – 10:20	**PLENARY 1**						
	Speaker	**Prof. Ruzhu Wang**					
	Chairman	Prof. Normah Mohd Ghazali					
	Host	Mr. Jaya					
10:20 – 10:35	TEA BREAK						
10:35 – 11:05		**Invited Speaker IMAT 1**	**Invited Speaker IMAT 2**	**Invited Speaker IMAT 3**			
	Speaker	**Prof. Agus Pamitran**	**Prof. Kim Choon Ng**	**Prof. Haslinda Mohamed Kamar**			
	Chairman	Prof. Mazlan A.Wahid	Prof. Ir. Hayati Abdullah	Dr. M. Noor Afiq Witri			
	Host	Mr. Mohd Amri	Mr. Mohd Hilmi	Mr. Ahmad Mustafa			

TIME	AIRMEET (15 minutes + 5 min Q&A)						
Session	**Session 1A**	**Session 2A**	**Session 3A**	**Session 4A**	**Session 5A**	**Session 6A**	**Session 11A**
Track	**IMAT - Combustion**	**IMAT - Biofuel**	**IMAT – Thermo-fluid**	**IMAT – Thermo-physical Properties**	**IMAT - Heat Transfer**	**IMAT - Energy Conservation**	**ICSERA**
Session Chair	Prof. M. Farid/Dr. Mohd Fairus	Dr. Hasbullah	Prof. Harinaldi/Dr. Hidayat	Prof. Kim Choon Ng/Prof. Mazlan	Prof. Nasrudin/Dr. Kameil	Prof. Ir. Hayati/Dr. Md. Mizanur	Dr. Mohsin/Dr. Ibtisham
Session Host	Mr. Mohd Amri	Mr. Mohd Hilmi	Mr. M. Syafiq	Mr. Ahmad Mustafa	Mr. Rossli	Mr. Ferdaus	Ms. Jannah
Physical Room	Dewan Seminar	Bilik Sangfor	Bilik Ruckus	Bilik Juniper	Bilik HIK Vision	Bilik Aruba	Bilik Mesyuarat Web dan Multimedia
11:10 – 11:30	7	6	2	65	39	1	
11:30 – 11:50	14	13	54	51	50	49	
11:50 – 12:10	11	16	62	25	88	64	
12:10 – 12:30	47	17	46	48	56	74	
12:30 – 12:50	68	27	40	10	45	75	
12:50 – 1:10	8	28	42	30	69	77	
1:10 – 2:30	LUNCH & SOLAT						
2:30 – 3:20	**PLENARY 2**						
	Speaker	**Prof. Simone Hochgreb**					
	Chairman	Prof. Mazlan Abdul Wahid					
	Host	Mr. Jaya					

TIME	AIRMEET (15 minutes + 10 min Q&A)						
Sesion	**Session 1B**	**Session 2B**	**Session 3B**	**Session 4B**	**Session 5B**	**Session 6B**	**Session 11B**
Track	**IMAT - Combustion**	**IMAT - Biofuel**	**IMAT – Thermo-fluids**	**IMAT – Thermo-physical Properties/ Environmen-tal Eng.**	**IMAT - Heat Transfer**	**IMAT - Energy Conservation**	**IMAT – Thermo-fluids**
Session Chair	Prof. Yulianto/ Dr. M. Noor Afiq Witri	Dr. Mohd Fairus/Dr. Hasbullah	Dr. Hidayat	Dr. Ummi/Dr. Norazila	Dr. Aminuddin/ Dr. Natrah	Dr. M. Wakil Shahzad/Dr. Md. Mizanur	Dr. Mohsin/Dr. Zul-hilmi
Session Host	Mr. Mohd Amri	Mr. Mohd Hilmi	Mr. M. Syafiq	Mr. Ahmad Mustafa	Mr. Rossli	Mr. Ferdaus	Ms. Jannah
Physical Room	Dewan Seminar	Bilik Sangfor	Bilik Ruckus	Bilik Juniper	Bilik HIK Vision	Bilik Aruba	Bilik Mesyuarat Web dan Multimedia
3:25 – 3:45	15	29	23	67	36	81	66
3:45 – 4:05	21	19	24	26	63	79	37
4:05 – 4:25	31	83	32	53	82	76	61
4:25 – 4:45	35	80	91	72	73	20	89
4:45 – 5:00	TEA BREAK						
5:00 – 5:20	12	43	5	70	87	86	92
5:20 – 5:40	84	60	57	9	93	85	71
5:40 – 6:00	4	90	41	22	94	55	
6:00 – 6:20	**Closing Remark**						

Session	Physical Room	Location
1	Dewan Seminar	Aras 4 UTMDigital
2	Bilik Sangfor	Aras 2 UTMDigital
3	Bilik Ruckus	Aras 2 UTMDigital
4	Bilik Juniper	Aras 2 UTMDigital
5	Bilik HIK Vision	Aras 2 UTMDigital
6	Bilik Aruba	Aras 2 UTMDigital
11	Bilik Mesyuarat Web dan Multimedia	Aras 4 UTMDigital

IMAT 2021 PLENARY AND INVITED SPEAKER DETAILS

PLENARY 1 **9:30 – 10:20**	Adsorbents for energy conversion and management, and its cross to energy-water-air nexus	Professor Ruzhu Wang Shanghai Jiao Tong University
INVITED SPEAKER IMAT 1 **10:35 – 11:05**	Sea-water Ice Slurry Generator	Prof. Agus Pamitran Universitas Indonesia
INVITED SPEAKER IMAT 2 **10:35 – 11:05**	A Common Energy Platform for Evaluating Efficiency of Desalination Plants.	Prof. Kim Choon Ng King Abdullah University of Science & Technology
INVITED SPEAKER IMAT 3 **10:35 – 11:05**	Covid19: It's Transport in a Confined Space	Prof. Haslinda Mohamed Kamar Universiti Teknologi Malaysia
PLENARY 2 **2:30 – 3:20**	Narrowing the Gap between Experiments and Models in Turbulent Reacting Flows	Prof. Simone Hochgreb University of Cambridge

SESSION 1 IMAT – COMBUSTION

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 1A IMAT - Combustion				
Session Host: Mr. Mohd Amri			Chairman: Prof. Mohd Farid M. Said /Dr. Mohd Fairus	
11:10 – 11:30	7	Flameless Combustion mode as a Promising Trend: A Review on its Fundamental, role towards Emissions Reduction, Fuel consumption and Performance Enhancement	**Abdelgader Agilah Gheidan**, Mazlan A. Wahid and Anthony Chukwunonso Opia	Universiti Teknologi Malaysia
11:30 – 11:50	14	Effect of Preheating on Combustion Characteristics of a Swirling Flameless Combustor	**Najib Aminu Ismail**, Raid Abid Alwan, Mazlan Abdul Wahid and Aminuddin Saat and Mohammed Bashir Abdulrahman	Universiti Teknologi Malaysia
11:50 – 12:10	11	Numerical Investigation on Air-Fuel Configurations on the Emission Features in a Vortex Flameless Combustion at Various Thermal intensity	**Abdelgader Agilah Gheidan** and Mazlan A. Wahid	Universiti Teknologi Malaysia
12:10 – 12:30	47	A Review of Liquid Fuel Flameless Combustion with Various Flow Configurations	**Mohammed Bashir Abdulrahman**, Mazlan Abdul Wahid, Ali Asmayou, Mohd Fairus Mohd Yasin and Md Mizanur Rahman	Universiti Teknologi Malaysia
12:30 – 12:50	68	Thermal and Emission Characteristics of Liquid Fuel flameless Combustion in a Forward Flow Combustion Chamber	**Mohammed Bashir Abdulrahman**, Mazlan Abdul Wahid, Ali Asmayou, Md. Mizanur Rahman and Mohd Fairus Mohd Yasin	Universiti Teknologi Malaysia
12:50 – 1:10	8	The Effect of Fuel Composition on Asymmetric Flameless Combustion Characteristic	**Abdelgader Agilah Gheidan**, Mazlan A. Wahid, Fudhail A Munir and Muhammad Amri Mazlan Wahid	Universiti Teknologi Malaysia
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 1B IMAT - Combustion				
Session Host: Mr. Mohd Amri			Chairman: Prof. Yulianto/ Dr. M. Noor Afiq Witri	
3:25 – 3:45	15	Experimental of Rotating Detonation Engine with Asymmetric Vortex Combustion Chamber Shape Combustion Chamber	Ahmad Dairobi Ghazali, Mazlan Abdul Wahid, Muhammad Hafiz Azeman and Muhammad Amri Mazlan	Universiti Teknologi Malaysia
3:45 – 4:05	21	Early Assessment of Thermal Analysis Rotating Detonation Engine	**Muhammad Hafizuddin Azeman**, Mazlan Abdul Wahid, Ahmad Dairobi Ghazali and Muhammad Amri Mazlan	Universiti Teknologi Malaysia
4:05 – 4:25	31	Wave Propagation Characteristics in Pre-detonator on Rotating Detonation Engine Initiation	**Muhammad Amri Mazlan**, Mazlan Abdul Wahid, Ahmad Dairobi Ghazali and Hafizuddin Azeman	Universiti Teknologi Malaysia
4:25 – 4:45	35	A Modified Single-Step Chemistry Mechanism for Biogas Detonation Simulations	**Mohammad Nurizat Rahman**, Mazlan Abdul Wahid and Mohd Fairus Mohd Yasin	Universiti Teknologi Malaysia
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	12	Numerical Simulation on a Modified Trapped Vortex Combustor	Dharsannen Sri Selvam, Norwazan Abdul Rahim, Mohd Rashdan Saad, Syahar Syawal, Hasan Mohd Faizal and **Mohd Rosdzimin Abdul Rahman**	Universiti Pertahanan Nasional Malaysia /Universiti Teknologi Malaysia
5:20 - 5:40	84	Influence of Equivalence Ratio on Emissions in Meso-Scale Vortex Combustor	Ali Asmayou, Mazlan Abdul Wahid and Mohammed Bashir Abdulrahman	Universiti Teknologi Malaysia
5:40 – 6:00	4	Prospects and Challenges of Nanofluids as Improved Fuel Diesel and Gasoline Engines: A Critical Review	**Ahmed Sule**, Zulkarnain Abdul Lattif and Mohd Azman Abas	Universiti Teknologi Malaysia
6:00 – 6:20	CLOSING REMARK			

SESSION 2 IMAT – BIOFUEL

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 2A IMAT - Biofuel				
Session Host: Mr. Mohd Hilmi			Chairman: Dr. Hasbullah	
11:10 – 11:30	6	Emission Mechanism and Investigation of Biofuel Combustion Characteristics at Different Techniques Towards NOx Emission Reduction: An Overview	**Abdelgader Agilah Gheidan**, Mazlan A. Wahid and Anthony Chukwunonso Opia	Universiti Teknologi Malaysia
11:30 – 11:50	13	Advance Thermal Configuration Using CFD in Co-firing Coal and Biomass	**Mohd Khairul Hafiz Md Lias** and Mazlan Abdul Wahid	Universiti Teknologi Malaysia
11:50 – 12:10	16	Thermal Characteristics of Biogas Flameless Combustion in Asymmetric Meso-scale Combustor	**Ali Asmayou**, Mazlan Abdul Wahid and Mohammed Bashir Abdulrahman	Universiti Teknologi Malaysia
12:10 – 12:30	17	Combustion Performance of a Single Cylinder Diesel Engine Fueled with Palm Oil Biodiesel Fuel Blends	**Muhammad Qadri Rusli**, Mohd Farid Muhamad Said, Ahmad Mustafa Sulaiman, Mohd Rozi Mohd Perang, Harrison Lau Lik Nang, Nur Sulihatimarsyila Abd Wafti, Mazlan Said and Mohd Kameil Abdul Hamid	Universiti Teknologi Malaysia
12:30 – 12:50	27	The Experimental of Performance Three Variations Gasoline with Bioethanol Blends (E80) on Engine Motorcycle Test Bench with Setting ECM	**Pandega Hariyanto**, Yoshua Calvin, Addarda Irsyad, Muchalis Masuku, Cahyo Wibowo and Bambang Sugiarto	Universitas Indonesia
12:50 – 1:10	28	Performance and Emission of Gasoline RON 88 – Bioethanol High Percentage Blends on Motorcycle Engine Test Bench	**Addarda Irsyad Usman**, Pandega Hariyanto, Yoshua Lian Calvin, Muchalis Masuku, Cahyo S. Wibowo, Muhammad Hanifuddin and Bambang Sugiarto	Universitas Indonesia
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 2B IMAT - Biofuel				
Session Host: Mr. Mohd Hilmi			Chairman: Dr. Mohd Fairus/Dr. Hasbullah	
3:25 – 3:45	29	Effect of Nitrous-oxide on laminar burning velocity and flame stability of biogas combustion	**Shehab Elhawary**, Aminuddin Saat, Mazlan Abdul Wahid and Mohd Zarhamdy Md Zain	Universiti Teknologi Malaysia
3:45 – 4:05	19	Combustion Behaviour of Palm Kernel Shell Torrefied by Mild Pressurization Technique	**Mohd Faizal Hasan**, Mohamad Zulhilmi Mat Ail and Mohd Farid Muhamad Said	Universiti Teknologi Malaysia
4:05 – 4:25	83	Spray Characteristic of Diesel Engine Injector by Using Palm Biodiesel as Fuel	**Siti Nurhidayah Mohd Fauzi**, Zulkarnain Abdul Latiff, Natrah Kamaruzaman and Mohd Farid Muhamad Said	Universiti Teknologi Malaysia
4:25 – 4:45	80	Density, Isothermal Compressibility & Isobaric Expansivity Measurement of Binary Mixture Containing Waste Cooking oil (WCO) &Propan-1-ol at 300.15 K	Sunita Rani, Nisha Sangwan and **Rajesh K Siwach**	Baba Masthnath University and Women Gov. College Gohana - India
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	43	Impact of Biodiesel B30 on Engine Oil Viscosity in Real-World Agricultural Conditions	**Mohd Rozi Mohd Perang**, Mohd Azman Abas, Zulkarnain Abdul Latiff and Mohd Farid Muhamad Said	Universiti Teknologi Malaysia
5:20 – 5:40	60	Nigeria Palm Oil Biodiesel Production Optimization as Post Covid-19 Green Energy and Economic Recovery Strategy	**Ahmed Sule**, Zulkarnain Abdul Lattif and Mohammed Azman	Universiti Teknologi Malaysia
5:40 – 6:00	90	Study on The Optimal Stack Diameter on The Efficiency of Thermoacoustic Engine	**Irna Farikhah**, E. A. Elsharkawy, Harto Nuroso, Mega Novita, Dian Marlina, Kwartriani Rahmatunnisa, Irfan Abd. Rahim, Mohd Zarhamdy Bin Md Zain, Khairunnisa Mohd. Paad and Ahmad Nadhif Masruri	Universitas PGRI Semarang/Kyushu University/SMA Generus Mandiri/Universiti Malaysia Perlis/UTM/Universitas Diponegoro
6:00 – 6:20	CLOSING REMARK			

SESSION 3 IMAT – THERMOFLUIDS

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 3A IMAT - Thermofluids				
Session Host: Mr. Muhammad Syafiq			Chairman: Prof. Harinaldi/Dr. Hidayat	
11:10 – 11:30	2	Influence of Guide Walls on the Aerodynamic Performance of a Vertical-Axis Wind Turbine	**Roaa Ansaf**, Zambri Harun and Mohammed Rasidi	Universiti Kebangsaan Malaysia
11:30 – 11:50	54	A Study on the Use of Brim on a Tidal Turbine with Diffuser Augmentation	**Andika Ikhsan Kamil**, Harinaldi and Ridho Irwansyah	Universitas Indonesia
11:50 – 12:10	62	Development of Wave-Induced Loads Program for Global Ship Hull Strength	**Kurniawan Teguh Waskito** and Yanuar	Universitas Indonesia
12:10 – 12:30	46	Drag Reduction of Float Design of Adaptation of Sailfish Body with Deadrise Angle Variation by CFD	**Antoni**, Gunawan, Allessandro Utomo Allessandro Utomo and Yanuar	Universitas Indonesia
12:30 – 12:50	40	Thermal Null-Offset of Open and Closed Water Tank in Dynamic Environment	**Reski Septiana**, Ibnu Roihan and Raldi Artono Koestoer	Universitas Indonesia
12:50 – 1:10	42	Influence of Operational Speed and Tunnel Length on Piston Wind Characteristics in Jakarta-Bandung High-Speed Railway Project	**Panji Malik Subhani** and Harinaldi	Universitas Indonesia
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 3B IMAT - Thermofluids				
Session Host: Mr. Muhammad Syafiq			Chairman: Dr. Hidayat	
3:25 – 3:45	23	Comparison of Flow Characteristics between Fluid and Coupled Fluid-Structure Interaction of an Automotive Mixed Flow Turbocharger Turbine	**Noor Zafirah Abu Bakar** and Muhamad Hasbullah Padzillah	Universiti Teknologi Malaysia
3:45 – 4:05	24	Secondary Flow Characteristic inside A Mixed Flow Turbocharger Turbine Volute at Different Operating Conditions	**Amiza Azmi** and Muhamad Hasbullah Padzillah	Universiti Teknologi Malaysia
4:05 – 4:25	32	Three- Dimensional Elastic Cantilever Plate Attached to A Solid Block Using FSI Solver in Open FOAM Technology	Maimouna Al Manthari and Perumal Nithiarasu	University of Tech. and Applied Sciences Oman/Swansea University UK
4:25 – 4:45	91	Numerical Investigation into Trapezoid Surface Texture of Journal Bearing and RBD Palm Oil as Lubricant	**Zuraidah Rasep**, Muhammad Noor Afiq Witri Muhammad Yazid, Syahrullail Samion and Nor Azwadi Che Sidik	Universiti Kuala Lumpur (MICET)/UTM
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	5	Flame-front Detection Using High-Speed Chemiluminescence and 2-D Mie-scattering Imaging via Endoscopic Access	Mohd Syahar Mohd Shawal, Suhairil Meon, Juri Saedon and Syahar Shawal	Universiti Teknologi MARA
5:20 – 5:40	57	Development of Cost-efficient Schlieren Flow Visualization using Short Focal-length Spherical Mirror	**Yahya Dimas**, Ridho Irwansyah and Harinaldi	Universitas Indonesia
5:40 – 6:00	41	Entropy in Macroscopic Thermodynamics: How Should be Presented and Understanding?	**Ahmed Bufares**	University of Benghazi, Libya
6:00 – 6:20	CLOSING REMARK			

SESSION 4 IMAT – THERMOPHYSICAL PROPERTIES & ENVIRONMENTAL ENGINEERING

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 4A IMAT – Thermophysical Properties & Environmental Engineering				
Session Host: Mr. Ahmad Mustafa			Chairman: Prof. Kim Choon Ng /Prof. Mazlan	
11:10 – 11:30	65	Characterization of Soy Wax Synthesized with Graphene and MAXene for Building Thermal Management	**Titin Trisnadewi**, Eny Kusrini, Dwi Marta Nurjaya and Nandy Putra	Universitas Indonesia
11:30 – 11:50	51	Modelling of Single Wall Carbon Nanotube Growth in Diffusion Flame	**Muhammad Syafiq Ridhwan** Selamat, Muhammad Thalhah Zainal, Mohd Fairus Mohd Yasin and Norikhwan Hamzah	Universiti Teknologi Malaysia
11:50 – 12:10	25	Carbon Nanomaterials Synthetization of Tire Pyrolysis Oil Using Laser and Spray Pyrolysis Techniques	**Muhammad Mat Junoh**, Aminuddin Saat, Mohd Faizal Hasan and Mohd Zarhamdy Md Zain	Universiti Teknologi Malaysia
12:10 – 12:30	48	Experimental Analysis on the Effect of Height-Above-Burner on Growth Region Characteristics and Morphology of Synthesized CNT in Methane Diffusion Flame	**Muhammad Hilmi Ibrahim**, Norikhwan Hamzah, Mohd Fairus Mohd Yasin and Nurul Adilla Mohd Subha	Universiti Teknologi Malaysia
12:30 – 12:50	10	Briquetting of Mixture of Empty Fruit Bunch (EFB) Fibre and Starch binder: The Effect of Compaction Pressure, Mixing Ratio, Drying Duration and Drying Technique	**Muhammad Ariff Hanaffi Mohd** Fuad, Mohd Faizal Hasan, William Chong Woei Fong and Farid Nasir Ani	Universiti Teknologi Malaysia
12:50 – 1:10	30	Effect of Storage Temperature to Carbon Deposit on Biodiesel Blend	**Aulia Rifai**, Jonathan Kevin, Vito Hutagalung, Jajang Amir Hidayat and Bambang Sugiarto	Universitas Indonesia
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 4B IMAT – Thermophysical Properties & Environmental Engineering				
Session Host: Mr. Ahmad Mustafa			Chairman: Dr. Ummikalsom/Dr. Norazila	
3:25 – 3:45	67	Tribology Characteristic of Cartilage Replacement using Pin on Disc Experiment	Mohamad Mazwan Mahat, Salmiah Kasolang, **Nurul Nadiah Mohd Kamaldin** and Mohd Syahar Mohd Shawal	Universiti Teknologi MARA
3:45 – 4:05	26	The Impact Critical Characteristic of Methanol Contaminant on Lubricant Properties Vehicle SI Engine	Rona Malam Karina, Milda Febria, Catur Y. Respatiningsih, Riesta Anggarani, Lies Aisyah, Setyo Widodo, Rizkia Malik, May Muchar, Dimitri Rulianto, Emi Yuliarita, Sylvia Ayu Bethari and Cahyo Setyo Wibowo	Research and Development Centre for Oil and Gas Technology "Lemigas", Indonesia
4:05 – 4:25	53	Comparation of Active and Passive Controls to Prove the Degradation Time	**Sealtial Mau**, Yanuar and Achmad Riadi	University of Nusa Cendana/Universitas Indonesia
4:25 – 4:45	72	Common Platform for Evaluating Energy Efficiency of Desalination Plants	Kim Choon Ng, Muhammad Burhan, Qian Chen, Doskhan Ybyraiymkul, Faheem Akhtar, Mkum Ja, Raid Alrowais and **Muhammad Wakil Shahzad**	King Abdullah University of Science and Technology (KAUST)/Northumbria University, UK
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	70	The Impact of Operational Flexibility on Electricity Production: A Case Study of a Waste-to-Energy Pilot Project in Bantargebang, Indonesia	Priska Alfatri Hendrayanto, I Putu Angga Kristyawan, Muhammad Hanif and Wiharja	BPPT, Indonesia
5:20 – 5:40	9	Open LCA Life Cycle Assessment Tool to Determine Environmental Life Cycle Hot Spots: Demonstrated to an Aluminium Patrolling Ship	**Md Mizanur Rahman** and Mohd Zamri Mohd Nor	Universiti Teknologi Malaysia
5:40 – 6:00	22	Techno-Economic Readiness and Acceptance Analysis Study of End-of-Life Vehicles (ELV) Implementation in Malaysia	Ahmad Nizam Che Kasim, Nurhidayah Mat Hashim, Noor Azuan Hashim and Zambri Harun	Universiti Kebangsaan Malaysia
6:00 – 6:20	CLOSING REMARK			

SESSION 5 IMAT – HEAT TRANSFER

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 5A IMAT – Heat Transfer				
Session Host: Mr. Rossli			Chairman: Prof. Nasrudin/Dr. Kameil	
11:10 – 11:30	39	Numerical Study of Laminar Convective Heat Transfer Study of Al2O3-H2O:EG Nanofluids	**Ales Daniel** and Ridho Irwansyah	Universitas Indonesia
11:30 – 11:50	50	Design and Analysis of Battery Thermal Management System Employing Air Cooling Method	**Zeluyvenca Avista**, Ubaidillah, Dominicus Danardono Dwi Prija Tjahjana, Eko Prasetya Budiana, Muhammad Nizam and Indri Yaningsih	Universitas Sebelas Maret, Indonesia
11:50 – 12:10	88	Temperature Evaluation of Pouch Lithium-ion Battery Module at Different Cells Arrangement and Thermal Conditions	**Mohd Ibthisham Ardani**, Mat Hussin Ab. Talib, Zul-Hilmi Che Daud, Zainab Asus, Norazila Othman and Mohd Aifaa Mohd Ariff	Universiti Teknologi Malaysia
12:10 – 12:30	56	Experimental on Transient Heating and Cooling of Natural Circulation Flow using A FASSIP-02 Large Scale Experimental Facility	**Mulya Juarsa**, Dedy Haryanto, Ainur Rosidi, Giarno, G. Bambang Heru K. and Adhika Enggar Pamungkas	BATAN, Indonesia
12:30 – 12:50	45	Investigation on Natural Convection in Heterogeneous Porous Enclosures Using Generalized Convection Model	**Abhijit Verma** and Gaurav Tomar	Indian Institute of Science, Bangalore, India
12:50 – 1:10	69	Unsteady Free Convection in Rectangular Enclosures Containing a Darcy-Forchheimer-Brinkmann Medium	**Abhijit Verma** and Gaurav Tomar	Indian Institute of Science, Bangalore, India
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 5B IMAT – Heat Transfer				
Session Host: Mr. Rossli			Chairman: Dr. Aminuddin/Dr. Natrah	
3:25 – 3:45	36	Optimization of Microchannel Heat Sink for Thermal Performance and Pressure Drop using Central Composite Design of Experiment	**Mohamad Nur Hidayat Mat**, Normah Mohd Ghazali and Patrice Estellé	Universiti Teknologi Malaysia
3:45 – 4:05	63	Comparison of 3D Modelling of Single Phase and Two-Phase Flow of Nanofluid through Corrugated Channels.	**Elhadi Abugnah**, Wan Saiful-Islam Wan Salim and Abdulhafid Alfaghi	Universiti Tun Hussein Onn Malaysia
4:05 – 4:25	82	Optimization of a Boron Nitride Nanotubes Nanofluid-Cooled Microchannel Heat Sink at Different Concentrations	**Nur Liyana Nabihah** Yusof, Hielfarith Suffri Shamsuddin, Patrice Estellé, Normah Mohd Ghazali and Ummikalsom Abidin	Universiti Teknologi Malaysia
4:25 – 4:45	73	Interpolated Thermophysical Properties for Minimum Thermal Resistance of a Microchannel Heat Sink	**Hielfarith Suffri** Shamsuddin, Patrice Estellé, Normah Mohd-Ghazali, Maziah Mohamad and Ummikalsom Abidin	Universiti Teknologi Malaysia
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	87	Supercooling Effects on The Drag of a Free-Falling Sphere	Muhammad Sofwan bin Mohamad, Irfan Abdul Rahim, Mohd Zarhamdy Md Zain, Coinneach Mackenzie Dover and Khellil Sefiane	Universiti Malaysia Perlis/Universiti Teknologi Malaysia/ University of Texas at Austin US/ University of Edinburgh UK
5:20 – 5:40	93	Indirect Evaporative Cooling System Improvement: A Review	**Mohd Fahmi Md Salleh** and Mazlan Abdul Wahid	Universiti Teknologi MARA / Universiti Teknologi Malaysia
5:40 – 6:00	94	Experimental and Numerical Investigation of Natural Convection over a Constant Heat Flux Vertical Plate	**Noman Ashraf** and Mazlan Abdul Wahid	Imam Abdulrahman Bin Faisal University/UTM
6:00 – 6:20	CLOSING REMARK			

SESSION 6 IMAT – ENERGY CONVERSION

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 6A IMAT – Energy Conversion				
Session Host: Mr. Ferdaus			Chairman: Prof. Ir. Hayati/Dr. Md. Mizanur	
11:10 – 11:30	1	Low-GWP Refrigerant blends as Replacements of R410A for Domestic Heat Pumps	**Changru Yang**, Nobuo Takata, Kyaw Thu and Takahiko Miyazaki	Kyushu University, Japan
11:30 – 11:50	49	A Review of Recent Advances in Liquid Desiccant Dehumidification and Air-Conditioning	**Oldy Fahlovi**, Nandy Putra and Dinni Agustina	Universitas Indonesia
11:50 – 12:10	64	Investigation on Micro CT Based Method for Performance Estimation of Copper Sintered-Wick Heat Pipe	**Dinni Agustina** and Nandy Putra	Universitas Indonesia
12:10 – 12:30	74	Thermodynamics analysis of revised OTEC Rankine cycle using ammonia refrigerant	**Muhammad Aiman Wafi** Suruji, Nazri Nasir, Norazila Othman, Wan Zaidi Wan Omar and Shabudin Mat	Universiti Teknologi Malaysia
12:30 – 12:50	75	Performance Characteristics of Coolant Additives for Vehicle Cooling System	**Brent Dylan Junus**, Zulkarnain Abdul Latiff and Mohd Rozi Mohd Perang	Universiti Teknologi Malaysia
12:50 – 1:10	77	Study Experimental Dehumidification System Working with Liquid Desiccant	**Wisnu Indrawan**, Muhammad Alhamid and Arnas Lubis	Universitas Indonesia
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 6B IMAT – Energy Conversion				
Session Host: Mr. Ferdaus			Chairman: Dr. Muhammad Wakil Shahzad/Dr. Md Mizanur	
3:25 – 3:45	81	Long Short-Term Memory Neural Network Model for The Control of Temperature in a Multi-Circuit Air Conditioning System	**Ibrahim Oleolo**, Hayati Abdullah, Ismail Mustapha, Maziah Mohamad, Mohammad Nazri Mohd Jaafar, Akmal Baharain and Sapiah Sulaiman	Universiti Teknologi Malaysia
3:45 – 4:05	79	Initial Design Parameters Optimization of Air-Cooled Solar Thermal Ammonia-Water Absorption Chiller	**Desy Agung**, M. Idrus Alhamid, Arnas Lubis and Nasruddin	Universitas Indonesia
4:05 – 4:25	76	Domestic Ejector Air-Conditioning System Performance Using Different Ejector and Refrigerant Type	**Nurul Aida Nabila Abdul Halim**, Zulkarnain Abdul Latiff and Mohd Rozi Mohd Perang	Universiti Teknologi Malaysia
4:25 – 4:45	20	An Advance Air-Conditioning System for Future Sustainability	Muhammad Wakil Shahzad and Kim Choon Ng	King Abdullah University of Science and Technology (KAUST)/Northumbria University, UK
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	86	Experimental Study on the Improvement of Thermal Comfort Inside a Car Cabin	Mohd Kameil Abdul Hamid, Rajwinder Singh, Mohd Zarhamdy Md Zain, Zul Hilmi Che Daud, Izhari Izmi Mazali and Abd Rahim Abu Bakar	Universiti Teknologi Malaysia
5:20 – 5:40	85	Experimental Investigation of Local Thermal Sensation of Vehicle Passengers During Cooldown	**Muhammad Noor Afiq Witri** Muhammad Yazid, Haslinda Mohamed Kamar, Mohd Zukhairi Abd Ghapar, Mohd Aizad Sazrul Sabrudin and Nurfarizal Rasid	Universiti Teknologi Malaysia
5:40 – 6:00	55	Effects of Horizontal Wall-Mounted Air Supply on Particle Distribution in an Operating Room: A Simulation Approach	**Huiyi Tan**, Keng Yinn Wong, Wah Yen Tey, Syie Luing Wong, Bemgba Bevan Nyakuma, Mohd Hafiz Dzarfan Othman, Xinyou Ho, Haslinda Mohamed Kamar, Chew Tin Lee and Muhd Suhaimi Deris	Universiti Teknologi Malaysia
6:00 – 6:20	CLOSING REMARK			

SESSION 11 IMAT – THERMOFLUIDS

TIME	IMAT	TITLE	AUTHORS	AFFILIATION
Airmeet Session 11A ICSERA				
Session Host: Ms. Jannah			Chairman: Dr. Mohsin/ Dr. Ibtisham	
11:10 – 11:30		ICSERA		
11:30 – 11:50		ICSERA		
11:50 – 12:10		ICSERA		
12:10 – 12:30		ICSERA		
12:30 – 12:50		ICSERA		
12:50 – 1:10		ICSERA		
1:10 - 2:30	LUNCH & SOLAT			
Airmeet Session 11B IMAT - Thermofluids				
Session Host: Ms. Jannah			Chairman: Dr. Mohsin/ Dr.Zul-hilmi	
3:25 – 3:45	66	A Laboratory-scale Study of Cleanroom Design for Ambient Haze Condition	Rizkyandra Wintantomo and Yulianto Sulistyo Nugroho	Universitas Indonesia
3:45 – 4:05	37	Skin and Body Temperature Parameter Calibration of MAX30100 Sensor Module Based on Arduino-Uno	**Juan Karnadi**, Ibnu Roihan and Raldi Artono Koestoer	Universitas Indonesia
4:05 – 4:25	61	Effect of Magnetic Field on the Velocity Distribution of Gallium and Galinstan in Circular Microchannel	Amnani Rosland, **Natrah Kamaruzaman** and Mohsin Mohd Sies	Universiti Teknologi Malaysia
4:25 – 4:45	89	Simplified Approach for Braking Pressure Applied to a Thin Concave Composite Wall Calculation	**Sofiene Helaili**	ISTEUB / LASMAP (EPT), Carthage University, Tunisia
4:45 – 5:00	TEA BREAK			
5:00 – 5:20	92	Investigation on the Effect of Surface Texture Depth into the Performance of Journal Bearing using CFD	**Zuraidah Rasep**, Muhammad Noor Afiq Witri Muhammad Yazid, Syahrullail Samion and Nor Azwadi Che Sidik	Universiti Kuala Lumpur (MICET)/UTM
5:20 – 5:40	71	Inertial Flow Focusing Device Fabricated using 3D-printed Mold and Replica Molding Technique	**Ummikalsom Abidin**, Nur Anis Humaira Mohamad and Fazila Mohd Zawawi	Universiti Teknologi Malaysia
5:40 – 6:00				
6:00 – 6:20	CLOSING REMARK			

SESSION 1A

7. Flameless Combustion Mode as a Promising Trend: A Review on its Fundamental, Role Towards Emissions Reduction, Fuel Consumption and Performance Enhancement

Abdelgader A.S. Gheidan, Mazlan Bin Abdul Wahid, Anthony C. Opia

High-Speed Reacting Flow Laboratory, School of Mechanical Engineering, Universiti Teknologi Malaysia,81310 UTM Skudai, Johor, Malaysia

Abstract

Flameless combustion techniques have created a huge opportunity in the patronizing high number of combustiblefuel materials. This opportunity is connected with its potential in providing acceptable features of the combustion regime,mostly centered on mixing the working medium (reactants) above the temperature of auto-ignition of the fuel. Recent research has provided some potential of flameless combustion towards pollution reduction and energy generation. However, this technology still requires more attention to improve its versatility. The effect of flameless combustion on emissions reduction and its performance enhancement both on biofuels and fossil products is reported in this paper. Flameless combustion follows mixture before oxidizes in the combustion process, thus generates lower NOx emissions with clear and flameless of very little visible radiation. Similar values in the area of pollutant emissions for all studied fuels,especially with bio-fuel showed that flameless combustion yields good results both on conventional and diluted fuels. The review still points that flameless combustion regime modelling might not be capable of predicting intermediate species forreducing the emissions to zero level due to some inherited properties in the operation. Assessing the thermal performanceof flameless combustion, showed similar temperature distribution for all the fuels studied, although bio-fuel temperature observed little below, due to the inert gases of CO2 of large amount towards cooling the reactants. Challenges like wall- flame quenching and residence time were discovered on micro-scale combustors. The report suggests that little modifications on the flameless combustor in the area of wall-flame quenching will contribute to solving the NOx emissionproblems.

Keywords: *Flameless combustion, NOx Emission reduction techniques, Emission mechanism.*

SESSION 1A

14. Effect of Preheating on Combustion Characteristics of Swirling Flameless Combustor

Najib Aminu Ismail[1,2] Raid Abid Alwan[1] Mazlan Abdul Wahid[1] AminuddinSa'at[1] Mohammed Bashir Abdulrahman[1,]

[1]High Speed Reacting Flow Laboratory, School of Mechanical Engineering, Universiti Teknologi Malaysia

[2]Department of Mechanical Mechanical Engineering, Ahmadu Bello University, Zaria

Abstract

Flameless combustion is a state-of-the-art combustion technique that provides uniform temperature distribution while increasing efficiency and reducing emissions. An internally preheated swirling flameless burner (IPSFC) operating in flameless mode has been developed and tested to provide improved combustion characteristics and very low pollutant emissions. The IPSFC burner was operated at a heat input of 7 kW to 15 kW with a preheated combustion temperature. Inthe study, the effects of combustion air with and without preheating on gaseous fuel combustion characteristics are investigated for the six cases: SFR2, SFR4, SFR42, PSFR2, PSFR4 and PSFR42. The results show that the best configuration for a flameless burner is the SFR42 case, a vortex burner with four tangential air inlets, 12 axial air inlets, and 11 coaxial fuel inlets. The preheated air was found to increase the thermal efficiency of the process by about 10% compared to the process without preheating but at the cost of a small increase in NOx emissions. At an equivalence ratio of 0.8, the lowest NOx and CO emissions were found to be 3 ppm and 24 ppm, respectively. Temperature uniformity varied from 0.03 in SFR42 to 0.04 in SFR2 at different equivalence ratios.

Keywords: *Preheat, Emission, Flameless combustion, Swirling*

SESSION 1A

11. Numerical Investigation on Air-Fuel Configurations on the Emission Features in a Vortex Flameless Combustion at Various Thermal Intensity.

Abdelgader A.S. Gheidan, Mazlan Bin Abdul Wahid, Lei Li, Amri M., Anthony C. Opia

High-Speed Reacting Flow Laboratory, School of Mechanical Engineering, Universiti Teknologi Malaysia,81310 UTM Skudai, Johor, Malaysia

Abstract

A numerical study on the effects of air/fuel configuration on NOx emissions control in a vortex flameless combustion at different thermal intensity was conducted in this paper. The investigation employed the structure of small combustors which are influenced by some combustion factors. The layout is produced through diverse air and fuel injection configurations selected due to their potential on distinct combustion characteristics. In reverse configurations mode, the airinjection port is located at the exit end of the combustor; while in the forward configurations, the air injection port is placedat the reversed end of the combustor exit, with a change of fuel position. Our investigation indicated that in both non-premixed and premixed combustion modes, the NOx discharge is extremely low. However, in premixed combustion mode, the forward and reverse flow configuration (FP, RP) premixed produced a significantly low level of NO and CO comparedto the other air configurations in non-premixed at all equivalence ratios Φ. The investigation observed that any changes tothe fuel injection position affect the mixture preparation, resulting in premature combustion at very low emissions, hot spots and increased emissions. The reverse-crossflow configuration (RC1) has more potential to attain lower NO (approximately 1.30E-05 ppm) together with low CO (approximately 223 ppm) emissions when compared to the other flow configuration (RC2 and RC3). More so, a lower combustor volume leads to a very high thermal intensity of 324–393MW/m^3- atm thus resulted in a reduced residence time and gas recirculation, and high CO and NOx emissions.

Keywords: *Vortex Flameless combustion, Non-Premixed, premixed combustion, Air configurations*

Emissions, Thermal intensity

SESSION 1A

47. A Review of Liquid Fuel Flameless Combustion with VariousFlow Configurations

Mohammed Bashir Abdulrahman[1, 2, a], Mazlan Abdul Wahid[1, b], Ali HousseinAsmayou[1, c], Najib Aminu Ismail[1, d], Md Mizanur Rahman[1, e], Mohd Fairus Mohd Yasin[1, f]

[1] High-Speed Reacting Flow Laboratory, Faculty of Engineering, School of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.

[2] Department of Mechanical Engineering, Federal Polytechnic Mubi, Adamawa State, Nigeria

Abstract

Flameless combustion offers more promising results than conventional combustion, such as reduced NOx emissions with enhanced thermal efficiencies. Flameless combustion of various gaseous fuels and furnace configurations have been fully investigated. However, the burning of liquid fuel flameless combustion is still scarce and not fully explored due to its complex processes such as atomization, evaporation, and mixture formation before combustion takes place. Despite the efforts and recent developments in this area, it is still essential to perform extensive studies on liquid fuel flameless combustion using various liquid fuels and flow configurations to improve the versatility of using liquid fuel as a source of energy. Several preliminary studies have reported numerous promising results of using liquid fuel in both small and complex designs. However, research in this area is still in the early stages, and thorough investigations are required, especially in simple and large burners to mimic industrial burners. This review paper aims to provide key findings from the latest development in liquid fuel flameless combustion with a thorough explanation of the combustion characteristics of liquid fuel flameless combustion. The end of this article highlights some possible contributions and research gaps.

Keywords: *flameless combustion, liquid fuel, low emissions, burner configuration, temperature uniformity*

SESSION 1A

68. Thermal and Emissions Characteristics of Liquid Fuel Flameless Combustion in a Forward Flow Combustion Chamber

Mohammed Bashir Abdulrahman[1, 2], Mazlan Abdul Wahid[1], Ali Houssein Asmayou[1], Adam Kasani[1], Mohd Fairus Mohd Yasin[1], Md Mizanur Rahman[1]

[1] High-Speed Reacting Flow Laboratory, Faculty of Engineering, School of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.
2 Department of Mechanical Engineering, Federal Polytechnic Mubi, Adamawa State, Nigeria.

Abstract

This paper presents the experimental results of liquid fuel flameless combustor for various heat inputs of 10, 12, 14, 16, 18, 20, and 21 kW. The visual observations and temperature profile in the reaction zone and the NOx and CO gasses at the outlet are used to evaluate the performance of the flameless combustor. Fuel and air are injected at ambient conditions. The tangential air inlets acted as swirl generators to improve the internal recirculation rates and residence time and increase the dilution of reactants entering the reaction zone, resulting in flameless combustion. Ethanol is injected symmetrically and axially, while air is injected tangentially through 12 nozzles along the entire length of the combustion chamber. The results showed that by distributing the tangential air inlets along the entire combustion chamber length, the overall swirl strength was reduced by distributing the swirl generators evenly along the entire length of the combustion chamber. This resulted in good combustion performance in flameless combustion mode with low peak temperature in the reaction zone. It is assumed that the uniform temperature distribution in the furnace is the cause of the reduction of NOx to 3.6 ppm and CO to 15 ppm at $\Phi = 0.9$.

Keywords: *Liquid fuel, flameless combustion, forward-flow, temperature-uniformity, low emissions*

SESSION 1A

8. The Effet of Fuel Composition on Asymmetric Flameless Combustion Characteristic

Abdelgader A.S. Gheidan[1], Mazlan Bin Abdul Wahid[1], Fudhail A Munir[2],Amri M.[1]

[1]High-Speed Reacting Flow Laboratory, School of Mechanical Engineering, Universiti Teknologi Malaysia,81310 UTM Skudai, Johor, Malaysia

[2]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka (UTeM), 76100 Durian Tunggal Melaka, Malaysia

Abstract

The world's energy demand has increased tremendously due to industrial development and population growth. The increased consumption of key energy sources such as coal, oil and natural gas has had a significant impact on the atmospheric environment. Of all alternative fuels, biogas offers the greatest potential benefit to the energy supply and the environment. Biogas from the anaerobic digestion of biomass and biological waste by microorganisms can be used in calefaction, transportation and power production as a sustainable energy supply. Nevertheless, the Low Calorific Value (LCV) of biogas poses a significant challenge for converting biogas into electrical or thermal energy. A three-dimensional (3D) CFD (Computational Fluid Dynamics) research was conducted to show the different arrangements of biogas flameless combustion compared to standard modes through using a computational methodology that impalements the Rε/k−ε and theeddy dissipation turbulence. Also, studying the effect of added hydrogen to the biogas composition from 2% to 8% by volume on NOx emissions. The findings validated flameless combustion as one of the leading methods for biogas usage. The decline in contaminant production and fuel consumption are the primary source of biogas flameless combustion dominance. Besides, adding only 2% of hydrogen to the biogas component leads to stability and uniform temperature, and adding hydrogen to the biogas component at up to 4% by volume alleviates the production of nitrogen oxides. However, adding hydrogen from 4% to 8% increases the NOx composition rate due to the high peak temperature.

Keywords: *Biogas, Flameless combustion, Low pollutant formation, Hydrogen, CFD, Asymmetric combustor*

SESSION 2A

6. Emission Mechanism and Investigation of Biofuel Combustion Characteristics at Different TechniquesTowards NOx Emission Reduction: An Overview

Abdelgader A.S. Gheidan, Mazlan Bin Abdul Wahid, Anthony C. Opia

High-Speed Reacting Flow Laboratory, School of Mechanical Engineering, Universiti Teknologi Malaysia,81310 UTM Skudai, Johor, Malaysia

Abstract

In combustion, nitrogen oxide is one of the pollutants produced especially under fossil fuels. As a result of high emissions from fossil fuels, alternative fuel sources with less or free from emissions have evolved in response to the energy demand and environmental challenges associated with fossil products. In mitigating combustion emissions, a variety of technologies have been developed, including patronizing bio-fuel and flameless combustion applications. This paper gives an overview of bio-fuel benefits and flameless combustion as a solution to fuel emissions and the concepts that underpin it. The study found that biodiesel combustion produces low NOx emissions with optimal operation conditions using additives, water injection, utilization of DeNOx and application of NOx adsorbed catalyst. However, influence by some parameters such as molecular structure and biodiesel properties, adiabatic flame temperature etc. were discussed. According to the findings on NOx emission mitigation methods, exhaust gas recirculation (EGR) and delayed injection timing are both reliable and low-cost techniques. Among these strategies, EGR reduces NOx emissions in bio fuelled engines by regulating oxygen intensity and combustion maximum temperature while marginally lowering HC and CO pollutants at a 5–25 % EGR rate.

Keywords: *Biofuel, Emission, Combustion, Emission reduction techniques, Emission mechanism*

SESSION 2A

13. Advanced Thermal Configuration using Computational Fluid Dynamic in Co-firing Coal and Biomass

Mohd Khairul Hafiz Md Lias[1], Mazlan Abd Wahid[1]

[1] High-Speed Reacting Flow Laboratory (HIREF),
School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia,
81310 Johor Bahru, Johor, Malaysia.

Abstract

In Malaysia, coal is the largest fossil fuel used to generate power electricity which is 54.05%. By burning coal, greenhouse gas (GHG) emissions will continuously increase and affect global warming. Therefore, one of the alternatives is by promoting the co-firing coal and biomass in the existing coal-fired power plant in Malaysia. The past decade has seen an increase in the use of Computational Fluid Dynamic (CFD) to analyse the thermal performance of the coal-fired power plant. A coal-fired power plant for this study is referring to a 150MW subcritical boiler with tangential burner with the design pressure and temperature are 188 bar and 540 °C. This plant burned coal from Adaro and Hatillo. CFD simulation for co-firing is using 5% of sawdust at first level burners. From this study, several CFD methods were applied including the Eulerian-Langrangian approach to solve the two-phase gas-solid equation, Reynolds Averaged Navier Stokes (RANS) equation combined with k-ε Model to solve the turbulence problem in the gas phase, Discrete Ordinates Method (DO) to solve the radiative transfer equation (RTE) and Weighted Sum of Gray Gases Model (WSGGM) was used to calculate the absorption coefficient of the gas mixture. Based on the contour analysis result, found out that by co-firing coal and biomass, the burnout is reduced from an average of 0.45 kg/s to 0.33 kg/s when compared to pure coal combustion. This is mainly due to high volatile matters (VM) in the biomass. The concentration per unit thermal is also reduced to 42% when co-firing with sawdust. The early devolatilization will increase the gas and local temperature at each burner zone. The CO concentration was higher at the burner zone due to devolatilization.

Keywords: *Computational Fluid Dynamics (CFD), Co-firing, Ash Deposit, Discrete Ordinates Method (DO), Burnout*

SESSION 2A

16. Thermal Characteristics of Biogas Flameless Combustion inAsymmetric Meso-scale Combustor

Ali Houssein Asmayou, Mazlan Abdul Wahid, Mohammed Bashir Abdulrahman

High-Speed Reacting Flow Laboratory, Facultyof Engineering, Schoolof Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.

Abstract

Thermal characteristics of non-premixed conventional flame and flameless mode at fixed fuel mass flow rate of biogas (60% CH4 + 40% CO2) in an asymmetric meso-scale vortex combustor were compared. Axial and tangential oxidizer configurations are investigated in this paper. The three-dimensional form of governing equations including the Realizable-k–e model and the eddy-dissipation have been described using to simulate temperature distribution and combustion stability. The effect of tangential oxidizer configurations on the temperature distribution is also explored. The non-premixed meso-scale flameless combustion is much more stable compared to the non-premixed conventional flame. With a low of 10 % O_2 concentration, the meso-scale flameless mode has higher temperature homogeneity throughout the combustor than conventional flame. The maximum temperature in meso-scale non-premixed conventional flame and flameless combustions were 1692 K and 1160 K respectively.

Keywords: *Meso scale combustion, Flameless mode, Non-Premixed, Biogas, tangential air*

SESSION 2A

17. Combustion Performance of a Single Cylinder Diesel EngineFueled with Palm Oil Biodiesel Fuel Blends

Muhammad Qadri Rusli[1], Mohd Farid Muhamad Said[1],Ahmad Mustafa Sulaiman[1], Mohd Kameil Abdul Hamid[1],Mohd Rozi Mohd Perang[1], Mazlan Said[1], Harrison Lau Lik Nang[2],Nur Sulihatimarsyila Abd Wafti[2]

[1]Automotive Development Centre, Institute for Vehicle Systems and Engineering,Universiti Teknologi Malaysia, 81310 Johor Bahru, Malaysia

[2]Malaysian Palm Oil Board, Persiaran Institusi, Bandar Baru Bangi,43000, Kajang, Malaysia

Abstract

Palm oil biodiesel fuel blends could be used as diesel fuel substitutes in compressed ignition diesel engines for power generation. Over the years, various versions of biodiesel fuel blends had been commercially deployed owing to theirbenefits to reduce the engine emission released while maintaining the cost for consumers. Although its benefits to the environment are well accepted, there is a need to compare the combustion performance of these fuel blends on said engines,since it affects the engine performance and emission released. In this study, three palm oil biodiesel fuel blends – B10, B20, and B30 – were tested in a single cylinder Yanmar L70N diesel engine. The engine was coupled to an eddy current dynamometer at 2500 rpm (medium speed) with loads up to 50% of its maximum. The cylinder pressures generated by the fuel blends were measured using a piezoelectric pressure transducer installed inside the combustion chamber and above the piston head. The crank angle degrees was recorded using a crank angle encoder and synchronized with the cylinder pressure using an NI data acquisition system. From these two data, combustion results such as the heat release rate, ignitiondelay, and combustion duration were determined. The high oxygen content of the B30 palm oil biodiesel fuel blend improved most combustion indicators such as the peak cylinder pressure and cumulative heat release. However, it may notbe as efficient since its combustion duration was much extended. While priorly mentioned capabilities are well established,its efficiency needs improvements before replacing currently conventional diesel fuel, B10.

Keywords: *Power Generation, Biodiesel Fuel Blends, Palm Oil, Single Cylinder*

SESSION 2A

27. The Experimental of Performance Three VariationsGasoline with Bioethanol Blends (E80) on Engine Motorcycle Test Bench with Setting ECM

Pandega Atana Tamma Hariyanto[1], Yoshua Lian Calvin[1], Addarda Irsyad Usman[1], Muchalis Masuku[1], M. Hanifuddin[1], Cahyo S Wibowo[1,2], N.A Fathurrahman[2], Maymuchar[2], R Anggarani[2], Bambang Sugiarto[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok-Indonesia

[2] Research and Development Centre for Oil and Gas Technology (LEMIGAS), Research and Development Agencyfor Energy and Mineral Resources, Ministry of Energy and Mineral Resources, Jakarta-Indonesia

Abstract

Ethanol has been considered as an effort to achieve an alternative of using fossil fuel. It will generate higher performance and cleaner emission. The properties of fuel blends affected the volatility characteristics, RON, distillation, density, RVP, and oxygenate content, decreasing heating value. The objective of this study is to analyze the effect using E80 in three variation gasoline from the market on SI engine. The gasolines (RON88, RON90, RON92) were blended with 80% bioethanol, and each fuel characteristic physical–chemical (ASTM D Methodc) was tested. A performance test was performed using motorcycle engine test bench (AVL Dyno Test) with ECM setting (Ignition Timing was set 20°BTDC 28°BTDC, 30°BTDC, 32°BTDC and injection duration -10% ms). The results of this study using E80 in three variation gasoline on SI engine (150cc engine) increased the performance by 0.0926% with improvement in ECM setting.

***Keyword**: Bioethanol, Performance, E80, SI engine, ECM*

SESSION 2A

28. Performance and Emission of Gasoline RON 88 – Bioethanol with High Percentage Blends on Motorcycle Engine

Addarda Irsyad Usman[1], Pandega Hariyanto[1], Yoshua Lian Calvin[1], MuchalisMasuku[1], Cahyo S. Wibowo[2], Muhammad Hanifuddin[2], Bambang Sugiarto[1]

[1,]*Universitas Indonesia, Depok, Indonesia*

[2,]*LEMIGAS, Jakarta Barat, DKI Jakarta, Indonesia*

Abstract

Bioethanol has been used as a blending agent with gasoline supporting reduction to the usage of fossil fuel in internal combustion engine. The impact for this was influence the volatility characteristics of the fuel blends to be better combustion rate and output. The engine requires variant of adjustment such as ignition timing and injection duration to counterbalance and optimize the characteristics of the fuel. Test subject to the research are gasoline RON 88 with variety of bioethanol blends of E70, E80, and E90. The configuration given to the ignition timing were increasing advance, and also faster injection rate. Advance ignition timing impacting faster ignition and faster opening of the injector impacting bigger volume of fuel that enter the combustion chamber. Research conducted using AVL Engine Dynamometer Testbed to read the result of the test of power, torque and fuel consumption, and also AVL Compact Diagnostic System to read the result of emission. Main results in this test was to proof the use of bioethanol as blending agent to improve the output of power, torque fuel consumption, also better emission, such increasing power 9,9% compared to pure gasoline, and 1,9% improvement for torque compared to pure gasoline.

Keywords: *performance, bioethanol, gasoline RON 88, emission*

SESSION 3A

2 Influence of Guide Walls on the Aerodynamic Performanceof a Vertical-Axis Wind Turbine

Rooa Ansaf[1], Islam Hashem[2], Mohd Rasidi Rasani[1], Zambri Harun[1, a)]

[1]Department of Mechanical and Manufacturing Engineering, Faculty of Engineering and Built Environment,Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Malaysia

[2]Mechanical Power Engineering Department, Faculty of Engineering Mattaria, Helwan University, 11718 Cairo,Egypt

Abstract

Costs, low efficiency and ability for self-starting are typical setbacks in many wind turbine installations. In order to overcome the disadvantages of vertical-axis wind turbine (VAWT) configuration, a simple and cost-effective steering technology is developed by installing a set of fixed guide walls around it. The influence of guide walls on the aerodynamic performance of a VAWT is analyzed numerically in this paper. Three geometrical parameters including inner radius, outer radius, and relative angle of the integrated guide walls are considered. A two-dimensional computational fluid dynamics (CFD) simulation employing the unsteady Reynolds-averaged Navier-Stokes (URANS) approach is carried out on a classical NACA 0021 straight-bladed VAWT. The Shear-Stress-Transport, SST k – ω model is employed to simulate the turbulent flows through the wind turbine. The numerical results are compared with the experimental data, and the comparison proved that the numerical simulation can accurately predict the aerodynamic performance of vertical-axis wind turbines. Three arrangements of flat-type guide walls are chosen for the current investigation. On basis of the 2-D modelling, the maximum power coefficient is increased from 0.31 (without guide walls) to 0.91 at the tip speed ratio, λ =3.3. A wider operating range could be obtained for the appropriate arrangement of guide walls that gives the best flow orientation at the inlet to the vertical-axis wind turbine. Furthermore, the flow-field demonstrated that passages between guide walls could act as a nozzle shape that could augment the flow velocity and consequently increase the massflow rate through the turbine.

Keywords: *Bionic airfoil, Vertical axis wind turbine, Cofficient of power, Computational fluid dynamic(CFD)*

SESSION 3A

54. A Study on the Use of Brim on a Tidal Turbine with Diffuser Augmentation

Andika Ikhsan Kamil[1], Harinaldi[1] and Ridho Irwansyah[1]

[1] Department of Mechanical Engineering, Universitas Indonesia, Kampus UI Depok, 16424, Indonesia

Abstract

Tidal turbine is a horizontal axis turbine utilizing tidal energy to generate electricity while several existing studies related to the tidal turbine have been carried out, especially on the tidal turbine design to increase the turbine performance. The tidal turbine design is improved in various methods, namely augmenting a diffuser, modifying the diffuser angle, and appending a brim at the end of the diffuser as used in wind turbines with the aim of keeping vortex away from the tidal turbine outlet to minimize pressure drop. This study aims to compare the use of the brim by observing the level of effectiveness and the efficiency of the output power. The comparison is made by modelling the tidal turbine at seven diffuser angles and the use of brim. The four diffuser angles were varied started from 10.43° to 35.97° with tidal current velocity around 0.7 m/s. The model is then analysed by applying the computational fluid dynamics approach utilizing ANSYS application. It is then expected to obtain higher level of effectiveness and power efficiency of the tidal turbine by introducing the brim combined with diffuser augmentation.

Keywords: *Tidal turbine, brim, turbine efficiency*

SESSION 3A

62. Development of Wave-Induced Loads Program for Global Ship Hull Strength

Kurniawan T. Waskito[1], Yanuar[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, Indonesia

Abstract

In recent years, the increasing size of merchant ships demands more precise calculations of global ship hull strength. Hence, accurate prediction of the wave-induced loads more significantly the vertical bending moment in violent seas has become inevitable for structural strength design. To develop an accurate prediction method, more detailed wave-induced local pressure distribution on the whole wetted surface of the ship hull needs to be calculated to obtain the integrated value of the vertical bending moment. For that purpose, some ship hull models are used in this study i.e., slender and blunt Wigley, and bulk carrier models. The 3-D linear frequency domain potential flow method is used to compute the wave-induced pressure and motions of a ship with forward speed in waves. We confirm the promising results of the integrated value of vertical bending moment in comparison with the real ships of bulk carrier model either from experiments or computation methods. In conclusion, this program is fast and reliable to compute the global ship hull strength at a preliminary design stage.

Keywords: *Wave-induce loads, vertical bending moment, pressure distribution*

SESSION 3A

46. Drag Reduction of Float Design of Adaptation of Sailfish Body with Deadrise Angle Variation By CFD

Antoni[1], Gunawan[2], Allessandro Utomo[2], and Yanuar[2]

Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Kampus UI Depok,Depok 16424, Indonesia

Abstract

A seaplane is an aircraft equipped with a hull type useful for water landing operations and retractable wheels forland landings. Several innovations to minimize hull resistance have been carried out, such as hydrofoil, polymer paint, water repellent paint, and air injection. Recently, there have been biomimicry innovations that can also reduce hull resistance. The concept of biomimicry is an approach by adopting or imitating innovations that have long existed in nature. This study uses the adaptation of the sailfish body shape with deadrise angle variation to the catamaran hull design to provide benefits such as reduced resistance. This study aimed to identify the effect of biomimicry design and deadrise angle variations on reducing the hull float resistance of seaplanes. Tests werecarried out on the monohull with Froude Number 0.5 – 0.9 with intervals of 0.1 and deadrise angles of 20°,25°, and 30°. The numerical test results show that the design with a 30° deadrise angle has the smallest drag coefficient value. The variation of the deadrise angle affects that increasing the deadrise angle can reduce the drag coefficientvalue. The biomimicry design drag coefficient is smaller than the conventional design in the Froude number rangeof 0.5 – 0.9.

Keywords: *Biomimicry; CFD; Deadrise Angles; Float Catamaran; Total Resistance*

SESSION 3A

40. Thermal Null Offset of Open and Closed Water Tank inDynamic Environment

Reski Septiana[1], Ibnu Roihan[1], Raldi A. Koestoer[1]

[1] Heat Transfer Laboratory, Mechanical Engineering Department, Universitas Indonesia, Depok, Indonesia

Abstract

Ambient condition is a natural phenomenon which holds many unspoken laws or truths, that can be analyzed toenrich human's mind. This surrounding condition seems dynamics or transient for 24hours of observation, but with a bigger perspective, it is indeed a large, complex, and quasi-steady state system. Yet the laws hold not only for the state of system,the property of the fluids available in the nature is also an interesting phenomenon. This paper studies the thermal null offset of half-filled water tank left in ambient environment. The study was performed by analyzing multi-days reading fromvarious sensors employed on the system. The repeated reading pattern produced by each sensor indicated that the thermalnull offset of the system was obviously affected by the dynamic of ambient temperature for the open and closed water tank.However, the sequential quasi steady state is different for open and closed tank. For the open tank, daily thermal equilibriumcondition was achieved with significant temperature difference between the sensors located below the water surface. Yet, insignificant temperature difference was found in closed tank. Hence, it indicates that the lid-closing accelerates the thermal equilibrium inside the tank. The thermal null offset of water tank will be used as a basis for developing new model of heattransfer from hot fluid inside the tank to the ambient air which commonly found in various industries.

***Keywords**: Thermal Null-Offset, Environmental Heat Transfer, Dynamics Condition, Vertical Cylinder Tank, Tropical Ambient Condition*

SESSION 3A

42. Influence of Operational Speed and Tunnel Length on Piston Wind Characteristics in Jakarta-Bandung High-Speed Railway Project

Panji Malik Subhani[1] and Harinaldi[1]

[1]Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia,Depok, Jawa Barat, 16424, Indonesia.

Abstract

Indonesia is developing its first high-speed railway that will be connecting Jakarta and Bandung. Given the nature of the landscape, the railway will have several tunnels along the route, including two tunnels in Halim and Walini. When a high-speed train passes along a tunnel, there is a phenomenon called the piston effect. The piston effect may lead to some concerns, such as aural discomfort among passengers and tunnel boom. One of the piston effect's components is piston wind, the wind gusts around the train when it travels along the tunnel. This study aimed to understand the piston effect phenomenon, the behavior of piston wind and its influencing factors. The study was conducted by simulating the movement of a 1:50 simplified high-speed train model along the tunnel in computational fluid dynamics software. The study found that when the train moves in the tunnel, there are wind gusts at both the front and back of the train. The wind velocity at the area closer to the train is bigger than other areas of the tunnel. The study also showed that operational speed has a more significant influence on the wind velocity behavior in high-speed train tunnels than tunnel length.

Keywords: *High-Speed Train, Piston Effect, Tunnel, Computational Fluid Dynamics, Piston Wind.*

SESSION 4A

65. Characterization of Soy Wax Synthesized with Graphene and MAXene for Building Thermal Management

Titin Trisnadewi[1], Eny Kusrini[2], Dwi Marta Nurjaya[3], Nandy Putra[1]

[1] Applied Heat Transfer Research Group, Department of Mechanical Engineering, Universitas Indonesia, Depok, West Java 16424, Indonesia
[2]Department of Chemical Engineering, Faculty of Engineering, Universitas Indonesia, Depok, West Java 16424, Indonesia
[3]Department of Metallurgical and Material, Faculty of Engineering, Universitas Indonesia, 16424 Depok, West Java, Indonesia

Abstract

PCM as a thermal energy storage has weaknesses, namely low thermal conductivity and poor material stability, especially for organic PCM. This study aims to characterize PCM by improving their properties using the shape stabilization by adding nanoparticles as a support material. The PCM soy wax was synthesized with two nanoparticles: graphene and MAXene Ti3ALC2. MAXene is a new family of 2D metal carbides that have close mechanical properties and thermal conductivity to graphene. The synthesis process consisted of stirring using magnetic stirrer and ultrasonication with various percentages of 0.1, 0.5, and 1 wt%. Based on the results obtained, it is known that the morphology of graphene and Ti3AlC2 is in the form of sheets that are able to bind more soy wax so that the stability of the material can be maintained. The value of the thermal conductivity of nano-PCM synthesis, it was found that there was an increase in the value of the thermal conductivity along with the addition of the percentage of nanoparticles. The soy wax+graphene mixture produced the highest thermal conductivity of 0.89 W/mK and the soy wax+Ti3AlC2 mixture was 0.85 W/m.K. Based on the results, the addition of nanoparticles can improve material stability and thermal conductivity.

Keywords: *TES, CM, soy wax, graphene, MAXene*

SESSION 4A

51. Modelling the Flame Synthesis of Single-wall Carbon NanoTubes in Non-premixed Flames with Aerosol Catalyst

Muhammad Syafiq Ridhwan Selamat, Muhammad Thalhah Zainal, Mohd FairusMohd Yasin, Norikhwan Hamzah

High-Speed Reacting Flow Laboratory (HIREF), School of Mechanical Engineering, Faculty of Engineering,Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia.

Abstract

The use of aerosol catalyst in the flame synthesis of carbon nanotube (CNT) is known to yield single-wallCNT (SWCNT) that is useful for various applications. Modelling works are needed to optimize operating conditionsfor SWCNT growth but are unavailable. Therefore, a baseline model for the aerosol-catalyst system in flames is developed and the effect of oxygen on SWCNT growth is investigated. A baseline flame model for a normal diffusionflame is established via Computational Fluid Dynamics. A dispersed phase model is employed to simulate the entrainment of catalyst particles. The flame model is coupled with a published CNT growth rate model to predict theCNT growth rate at each particle. Inlet oxygen concentration is varied from 19% to 27% to study the effect of oxygenon SWCNT growth. Satisfactory validation of the baseline flame shape and temperature is established. Results showthat increasing oxygen concentration from 19% to 27% results in a 30% decrease in CNT length for representative particle 3 for each inlet condition. Regardless of burner operating conditions, high SWCNT growth is consistently predicted between 120-140 mm HAB, which indicates the existence of an optimum range for SWCNT growth in aerosol-based flame synthesis.

Keywords: *Computational Fluid Dynamics (CFD), Dispersed Phase Model (DPM), Single-Wall Carbon Nanotube (SWCNT), Flame Synthesis, Aerosol-Based Synthesis*

SESSION 4A

25. Carbon Nanomaterials Synthetization of Tire Pyrolysis OilUsing Laser and Spray Pyrolysis Techniques

Muhammad Mat Junoh, Aminuddin Saat, Mohd Faizal Hasan and Mohd Zarhamdy Md Zain

School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, UTM, 81310 Johor Bahru, Johor, Malaysia.

Abstract

Carbon nanomaterials (CNMs) synthesis techniques using laser and spray pyrolysis are presented in this paper. Tire pyrolysis oil (TPO) was used as the carbon source for the synthetization. Parameter such as temperature, gas flow rate, time reaction and peristaltic pump speed was implemented. Elemental analysis for TPO has shown hydrogen, nitrogen and sulphur consist with 86.05%, 9.69%, 1.16% and 1.09% accordingly. Laser pyrolysis technique divided into four methods: brick medium, rotary, vapor and spray. As the results, brick medium shown the maximum yield for CNMs synthesized at 500^{o}C with reaction time of 10 minutes and speed of 3 rpm with 0.0734 grams per cycle. On the other hand, spray pyrolysis experimental has been done using a conventional vertical furnace and the stainless steel 316 (SS316) was selected as the catalytic reactor. The experiment found that implementation of temperature 900^{o}C, 0.5L/min N2 flow rate and 3 rpm peristaltic pump speed has successfully produced CNMs and then collected for FESEM and EDX analysis. Furthermore, yield analysis on the SS316 plate (dimension 7.5cm x 3 cm) has found CNMs yield increased with the increasing peristaltic pump speed (1-4 rpm) up to 753%.

Keywords: *Carbon nanomaterials; laser pyrolysis; spray pyrolysis; tire pyrolysis oil*

SESSION 4A

48. Experimental Analysis on the Effects of Height-Above-Burner on Growth Region Distribution, Characteristics and Morphology of Synthesized Carbon Nanotubes in Methane Diffusion Flame

Muhammad Hilmi Ibrahim[1,2], Norikhwan Hamzah[1,2], Mohd Fairus Mohd Yasin[1,2], and Nurul Adilla Mohd Subha[3]

[1]School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru,Johor, Malaysia

[2]High Speed Reacting Flow Laboratory, Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia

[3]School of Electrical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru,Johor, Malaysia

Abstract

Analysis of growth region characteristics of carbon nanotubes (CNTs) synthesized in a methane diffusion flame at atmospheric conditions was performed. The CNT was synthesized on nickel nanoparticles which were formed through oxidation of pure nickel wire in flame. The pure nickel wire was accurately positioned in the flame using a 2-axis robotic arm at the desired height-above-burner (HAB). The analysis focuses on the effect of HAB on temperature, distribution anddensity of the CNT growth region, including synthesized CNT diameters. Overall, there were apparent differences between CNT synthesized at low HAB (9 mm and below) and the CNT synthesized at high HAB (10 to 14 mm). Experimental results showed that the average CNT growth region temperature at low HAB is 100°C higher than at high HAB. Additionally, the CNT growth region temperatures at low and high HAB were consistent with a relatively narrow range ofless than 40°C. Furthermore, the average CNT growth region at high HAB was 30% wider with higher CNT growth density than at low HAB. The synthesized CNT outer diameters constant to HAB due to synthesizing at growth region. The forest density at low HAB is lower in percentage than at higher HAB possibly due to difference in the catalytic activity rate. The synthesized CNT morphology of the as-grown tubes was examined by Field Emission Scanning Electron Microscopy (FESEM), High-resolution Transmission Electron Microscopy (HRTEM), Energy Dispersive X-ray (EDX), and by Raman Spectroscopy. Whereas the CNT growth regions were analyzed using wire-based macro-image analysis.

Keywords: *Carbon nanotubes, Diffusion flame, Combustion, Electron microscopy, Flame synthesis*

SESSION 4A

10. Briquetting of Mixture of Pulverized Empty Fruit Bunch(EFB) Fiber and Starch Binder

Muhammad Ariff Hanaffi Mohd Fuad[1], Mohd Faizal Hasan[1,2], William ChongWoei Fong[1,2], Farid Nasir Ani[1]

[1]School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM JohorBahru, Johor, Malaysia

[2]Automotive Development Centre (ADC), Institute for Vehicle Systems and Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

Abstract

This study focuses on the physical appearance, surface structure and moisture content of briquette made of mixture of pulverized Empty Fruit Bunch (EFB) fibre and starch binder. The effect of compaction pressure (20-80MPa), mixing ratio of EFB and starch binder (90:10, 80:20, and 70:30), and drying duration (50-150 minutes) were investigated. Moisture content, unit density and physical appearance (i.e cracking surface) were determined. Only unit density increased with increasing compaction pressure, but the increase was not significantly different ($p>0.05$) between one another. With increasing starch binder composition, the moisture content significantly increased ($p<0.05$) and lead to the longer drying duration and expansion of the produced briquette. With increasing drying duration, the moisture content and unit density of briquette decreased significantly ($p<0.05$) and caused a mild to severe deterioration on the physical surface of dried briquette. A suitable briquetting condition obtained at compaction pressure of 80 MPa, mixing ratio of 80:20 and drying duration of 120 minutes provides a less cracking surface, high unit density (766.82±2.45 kg/m^3) and moisture content (~5%). The findings obtained are beneficial for future study such as torrefaction or as fuel feeding.

Keywords: *Empty fruit bunch, Starch binder, Physical appearance, Surface structure.Moisture content*

SESSION 4A

30. Effect of Storage Temperature to Carbon Deposit onBiodiesel Blend

Aulia Rifai[1], Jonathan Kevin Samuel[1], Vito Hutagalung[1], Jajang AmirHidayat[2], and Bambang Sugiarto[3]

[1] Undergraduate School of Mechanical Engineering Universitas Indonesia, Depok 16424 Indonesia
[2] Graduate School of Mechanical Engineering Universitas Indonesia, Depok 16424 Indonesia
[3] Faculty of Engineering, Universitas Indonesia, Depok 16424 Indonesia

Abstract

Energy is the most important aspect in human life. With the population growth, the need of energy will exponentially increase. According to International Energy Agency (IEA) there's a 45% growth or 1.6% growth per yearand 80% of the energy generated by fossil based fuel. This become a problem because the fossil based fuel is non-renewableenergy and the demand of energy is keep on increasing. The world needs renewable based fuel to overcome this problem, the only problem is, the biodiesel prone to degradation of its quality and have combustion effect to the engine. The purposeof this study was to determine the temperature effect of biodiesel storage to determine what's the best condition to store biodiesel to avoid deposit. Based on this test, there's a correlation between temperature and deposit growth on biodiesel, the lower the temperature will create more deposit on biodiesel storage.

Keywords: *Degradation quality; Temperature and humidity effect; Hot Chamber Deposition Test Rig; Deposit on biodieselstorage*

SESSION 5A

39. Numerical Study of Laminar Convective Heat TransferStudy of Al_2O_3-H_2O:EG Nanofluids

Ales Daniel[a)] and Ridho Irwansyah[b)]

Fluid Mechanics Laboratory, Department of Mechanical Engineering, University of Indonesia, Depok 16424, Indonesia

Abstract

Nanofluids generate considerable interest in increasing the heat transfer performance of a system, specifically as a coolant. A study of Aluminum Oxide (Al2O3) nanoparticles in water and ethylene glycol (60:40) mixture is conductedby using computational fluid dynamics (CFD) to investigate its potential by assuming a single-phase and constant thermophysical properties model. This study aims to analyze its heat transfer enhancement by using a theoretical propertiesmodel in several volume concentrations: 0.3%, 0.6%, 1.0%, and 1.5%. A straight tube geometry by neglecting its wall thermal conductivity is used under laminar conditions and constant heat flux and 293 K inlet temperature, compared with various Reynolds numbers. The model is validated using Shah Correlation, resulting 8% maximum deviation. The Einsteintheoretical model fails to predict the viscosity properties resulted in more than a 10% deviation from the current experimental findings. Notwithstanding the theoretical prediction, the simulation observed that a 1.5% volume concentration increased higher heat transfer enhancement around 3%.

Keywords: *Nanofluids, CFD, Laminar, Single-phase, Heat transfer*

SESSION 5A

50. Design and Analysis of Battery Thermal ManagementSystem Employing Air Cooling Method

Zeluyvenca Avista[1,a)], Ubaidillah[1,b)], Dominicus Danardono Dwi Prija Tjahjana[1,c)],Eko Prasetya Budiana[1,d)], Muhammad Nizam[2,e)], and Indri Yaningsih[1,f)]

[1]Department of Mechanical Engineering, Universitas Sebelas Maret, Surakarta 57126, Indonesia

[2]Department of Electrical Engineering, Universitas Sebelas Maret, Surakarta 57126, Indonesia

Abstract

Li-ion batteries are an essential component used in Electric Vehicles and Hybrid Electric Vehicles, where theirtemperature contributes a significant effect on safety and performance. This study design and analysis for a battery thermalmanagement system using air as a coolant. The work was done by employing ANSYS Fluent. The modification of the position of the battery module with 18650 batteries was investigated. The space between the batteries supports the air conditioner flowing in and out. The two sides of the battery module wall were also fully opened. The heat generated duringdischarge was simulated with the help of a user-defined function. The ambient air temperature was set by 22°C, where air velocity was kept constant by 0.1 m/s. A pressure-based, laminar, k-epsilon turbulent, incompressible, transient solver wasused in the simulation. The solver used a SIMPLE algorithm for pressure and velocity coupling. Airflow in the fluid domainwas considered as laminar flow. A three-dimensional arrangement equation solves the computing domain for mass, momentum, and energy eternity during battery discharge conditions. Fluid domains and solid domains are combined to simulate conjugate heat transfer. The simulation shows that the staggered battery module lowers the temperature better than the aligned battery module.

Keywords*: Li-ion, air coolant, temperature,battery thermal management system*

SESSION 5A

88. Temperature Evaluation of Pouch Lithium-Ion Battery Module Different Arrangement and Thermal Conditions

M. I. Ardani[1], V. Punichelvan[2], M. H. Ab Talib[2], Z. H. C. Daud[3], Z. Asus[3], N. Othman[4] and M. A. M. Ariff [5]

[1]Centre for Low Carbon Transport in Cooperation with Imperial College London (LoCARtic), 81310 Skudai, Johor Bahru, Malaysia

[2]School of Mechanical Engineering, Faculty of Engineering Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Malaysia

[3]Automotive Development Centre (ADC), School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Malaysia

[4]Panel Aeronautics, School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Malaysia

[5]Faculty of Electrical and Electronic, Universiti Tun Hussein Onn, Batu Pahat, Malaysia

Abstract

Thermal transport, which occurs in a single cell due to charging and discharging, will affect the cell's electrical performance. The heat generated needs to be appropriately managed via external cooling to ensure optimum electrical and electrochemical performance and whilst minimising any cell degradation. Therefore, modelling of thermal behaviour of the batteries is essential, which will essentially serve as a tool for evaluating cell performance and safety. Additionally, this can be used as a guideline, particularly in the design stage of battery module/pack design. This paper presents a thermal analysis of a lithium-ion battery module with different cell arrangements under various inlet velocities. The average outlet temperature of each module and average battery surface temperature are simulated at different cell gaps. Increasing the air velocity of the inlet cooling air will increase the magnitude of turbulence in the cell casing domain. At low airspeed, tighter cell gaps promote better cooling; however, when the airspeed increases, larger spacing between cells gives pronounce cooling effect. This is manifested by a higher outlet air temperature up to 3 degrees Celsius between arrangement of 5-mm to 10-mm. Moreover, higher cell gaps cause the cell temperature to be relatively uniform at higher air speed.

Keywords: *Lithium-Ion Battery, Heat Transfer, Convection*

SESSION 5A

56. Experimental study on Transient Heating and Cooling of Natural Circulation Flow using A FASSIP-02 Large Scale Experimental Facility

Mulya Juarsa[1], Giarno1, Dedy Haryanto[1], Ainur Rosidi[1], G. Bambang Heru K.[1] and Adhika Enggar Pamungkas[1]

[1] Thermal Hydraulics Experimental Laboratory, Centre for Nuclear Reactor Technology and Safety, National Nuclear Energy Agency of Indonesia, PTKRN Gd.80 Kawasan PUSPIPTEK Setu, Tangerang Selatan 15314

Abstract

The Fukushima Daiichi nuclear plant accident in 2011 has provided important information about the failure of the active cooling system if the installation does not have electrical power. So that research on the phenomenon of natural circulation flow through passive cooling systems is an essential topic in improving the safety management of nuclear power plants in the world. However, research on natural circulation flow experimentally using large-scale facilities has not been widely carried out. This research aims to analyse the natural circulation flow rate phenomenon under transient heating and transient cooling conditions. The research method was conducted experimentally using the FASSIP-02 Test Loop, a large-scale experimental facility with a head height of 9.1 meters. Experiments were carried out by varying the temperature setting in the heat source (WHT), respectively, at 60 °C, 70 °C, and 80 °C. The results showed that the flow regime formed was in a turbulent flow regime, with the Reynolds numbers being 9352, 12634, and 17370, respectively, for variations in temperature settings of 60 °C, 70 °C, and 80 °C.

Keywords: *Nuclear accident, transient, natural circulation, passive cooling, Reynolds number, FASSIP*

SESSION 5A

45. Investigation on Natural Convection in Heterogeneous Porous Enclosures Using Generalized Convection Model

Abhijit Verma and Gaurav Tomar

Department of Mechanical Engineering, IISc Bangalore, Bengaluru-560012, India

Abstract

Natural convection in heterogeneous porous enclosures heated from side is investigated using generalized porous convection model. A second-order accurate finite volume scheme is employed to solve the governing equations. We consider a square enclosure with porosity varying linearly from the wall to the core and a vertical enclosure with porosity varying exponentially near the walls as test cases in the current investigation. It is observed that generalized convection model solved by second-order scheme captures the underlying physics better and is able to reduce the discrepancy between the computational and theoretical/experimental results available in the literature.

Keywords: *Natural convection, Variable porosity medium, Generalized model, Second order scheme*

SESSION 5A

69. Unsteady Free Convection in Rectangular Enclosures Containing a Darcy-Forchheimer-Brinkmann Medium

Abhijit Verma[1] and Gaurav Tomar[2]

[1] Department of Mechanical Engineering, IISc Bangalore, Bengaluru-560012, India

Abstract

Numerical study of unsteady free convection in enclosures containing Darcy-Forchhimer-Brinkmann medium is carried out. For an initial stationary and isothermal fluid in the enclosure, free convection occurs due to sudden heating and cooling of the opposite side walls keeping the horizontal ones in adiabatic condition. Finite volume method with projection algorithm is used to solve the governing equations. Effect of permeability of the medium on unsteady Nusselt number averaged over side wall as well as central vertical section is analysed for a given Rayleigh number in square, tall and shallow enclosures.

Keywords: *Unsteady free convection, Darcy number, Aspect ratio, Average Nusselt number*

SESSION 6A

1. Low-GWP Refrigerant blends as Replacements of R410A for Domestic Heat Pumps

Changru YANG[1],Nobuo TAKATA [1],Kyaw THU [1, 2],TakahikoMIYAZAKI[1, 2]

[1]Department of Energy and Environmental Engineering Interdisciplinary Graduate School of Engineering Sciences, Kyushu University Kasuga-koen, Kasuga-shi, Fukuoka 816-8580, Japan
[2] Thermal Science and Engineering Division, International Institute of Carbon-Neutral Energy Research (I2CNER), Kyushu University744 Motooka, Nishi-ku, Fukuoka 819-0395, Japan

Abstract

Domestic heat pumps occupy a significant share of the global heat pump industry. R410A has been a dominantrefrigerant in most domestic heat pumps. This binary blend of R32 and HFC125 with 50% composition offers excellent performance in terms of the coefficient of performance (COP) and the volumetric capacity with near azeotropic nature. However, R410A significantly impacts climate because of its high global warming potential (GWP) of 2088. R32 exhibits a promising performance among the pure refrigerant alternates, and almost all the domestic heat pumps have been replaced with R32 in Japan. But R32 has a GWP value of 677, which is far higher than the value targeted by regulations such as European F-Gas directives. Limitations of the pure refrigerants for the next-generation heat pumps are widely reported. Thus, the objective of the present article was to investigate refrigerant blends with a low GWP to replace R410A in domestic refrigeration applications. A screening of the binary combinations was carried out among the list of 5 pure fluids from one hydrofluorocarbon (HFC), three hydrofluoroolefin (HFCs), and Carbon Dioxide. The selection was based on the single-stage heat pump cycle model by Python. The target GWPs (lower than 300), the relative COPs, and the relative volumetric capacities were considered as metrics of the evaluation compared to the R410A as a baseline system. The result shows that the mixture of R32 and R1234yf would be the most competitive substitute for R410A. This research might supply a clue for the green, environmental, and safe replacement of R410A.

Keywords: *Low-GWP refrigerants, Refrigerant Blends, Alternatives to R410A*

SESSION 6A

49. A Review of Recent Advances in Liquid DesiccantDehumidification and Air-Conditioning

Oldy Fahlovi[1], Nandy Putra[1] and Dinni Agustina[1]

[1]Faculty of Engineering, Universitas Indonesia, Depok 16424, Indonesia.

Abstract

The use of refrigeration systems in building utility systems has been widely used since the twentieth century. However, the data shows that there are many problems that arise and are dangerous to the environment, such as global warming and high energy consumption rates. Several studies using liquid desiccants have shown significant results in overcoming this problem. Through independent temperature and humidity control, the desiccant air conditioning (DAC) system has multiple functions advantages (for example, efficient moisture control, no ozone-depleting coolant and easy regeneration integration, etc.) Compared with the traditional vapor compression cooling system, it is becoming more and more attractive to research attention. Many studies have been conducted to improve the overall performance of the DAC system, improve system configuration and optimization by integrating various energy technologies system design and control. This article introduces a literature review of the latest research progress on liquid desiccant dehumidification and air conditioning systems.

Keywords: *Dehumidification, Desiccant, System Integration*

SESSION 6A

64. Preliminary Investigation on Micro CT Based Method for Performance Estimation of Sintered-Wick Heat Pipe

Dinni Agustina[1] and Nandy Putra[2]

[1] Post-graduate Student of Department of Mechanical Engineering, Universitas Indonesia, Depok-16424, Indonesia
[2] Applied Heat Transfer Research Group, Department of Mechanical Engineering, Universitas Indonesia, Depok-16424, Indonesia

Abstract

The use of sintered-wick heat pipes in heat exchanger performance-related research has gain massive popularity owing to the simplicity and affordable cost of the heat pipes. The system performance is mostly evaluated based on various arrangement and the numbers of heat pipes applied. The initial performance parameters of those commercial heat pipes are not commonly available from the manufacturer. Series of testing must be conducted to obtain that information. Porosity and permeability of the wick are some of the important parameters in investigating the capillary pumping that will eventually affect the performance of heat pipes. This research aims to investigate the feasibility of X-Ray Micro Tomography (Micro CT) equipment to observe the porosity, pore sizes and permeability of sintered wick heat pipe. Visual data of two (2) samples of heat pipes has been quantified and analysed. Three samples of different porosity were made by additive manufacturing technology and scanned by Micro CT. The Micro CT analysis of these porosity samples will serve as a baseline information on fabricating sintered wick with metal 3D printing. The quantification of porosity and pore sizes values of the sintered-wick heat pipes related to their performances is further investigated.

Keywords: *Sintered wick, heat pipe, porosity, Micro CT, non-destructive test*

SESSION 6A

74. Thermodynamics Analysis of Revised OTEC Rankine Cycle Using Ammonia Refrigerant

Muhammad Aiman Wafi Suruji[1], Nazri Nasir[1], Norazila Othman[1], Wan Zaidi Wan Omar[1] and Shabudin Mat [1]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310, UTM Skudai, Johor, Malaysia

Abstract

Ocean Thermal Energy Conversion (OTEC) is one of the Organic Rankine Cycle (ORC) applications, which uses heat energy to produce mechanical work that rotates generators to produce electricity. OTEC system utilized the low-temperature difference between warm surface seawater and cold deep seawater to heat and cool the refrigerant in the system. Since the system's thermal efficiency was relatively low due to the low-temperature difference of seawater, revised OTEC cycles are the solutions to improve the system. For example, Solar Boosted OTEC (SOTEC) and Ejector Pump OTEC (EP-OTEC) cycle modify the basic OTEC cycle, thus obtain the thermal efficiency of 3.3% to 4%. Both modifications change their rotating turbine parameters differently, but these thermodynamic cycles can be combined to form a revised OTEC cycle with higher thermal efficiency. This study developed an algorithm for revised OTEC using MATLAB and further quantified the performance after the modifications. Compared to the previous OTEC cycle thermal efficiency of 3.1 %, this SEP-OTEC cycle thermal efficiency gives 1.2-fold of improvement.

Keywords: *Ocean, Rankine cycle, solar, ejector, ammonia*

SESSION 6A

75. Performance Characteristics of Coolant Additives for Vehicle Cooling System

Brent Dylan Junus[1], Zulkarnain Abdul Latiff [2], and Mohd Rozi Mohd Perang[2]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia

[2]Automotive Development Centre, Institute for Vehicle Systems and Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia

Abstract

Vehicle cooling system removes heat from the engine to maintain operating temperature. Coolant foaming could decrease the heat transfer of the coolant by clogging the radiator. Coolant additives claims to improve radiator performance and have antifoaming effects. The objective of this study was to investigate the performance of coolant additives for vehicle cooling system. First experiment uses an existing vehicle cooling test rig and the coolant additive effects on radiator effectiveness and temperature were evaluated. Second experiment uses ASTM D1881-17 foam test rig. The highest radiator effectiveness increased from 65.39% to 69.72% for tap water and 72.82% to 74.01% for distilled water when coolant additive was added. The highest value for temperature difference between the inlet and outlet of radiator hoses also increased from 1.93°C to 2.07°C for tap water and from 2.09 °C to 2.17°C for distilled water. The highest volume of foam generated decreased from 340ml to 290ml for tap water and from 460ml to 410ml for distilled water when coolant additive was added by 1% of the foaming agent. The rate of foam breakdown was also more obvious when coolant additive was added to the coolants. Therefore, coolant additives can improve radiator performance and have antifoaming effects.

Keywords: *Vehicle cooling, radiator effectiveness, coolant additive*

SESSION 6A

77. Study Experimental Dehumidification System Working with Liquid Desiccant

Wisnu I, M. Idrus Alhamid, N Nasruddin, A Lubis

Department of Mechanical Engineering, University of Indonesia, Depok 16424, Indonesia

Abstract

Lately, much attention is given to improve Indoor Air Quality (IAQ) to address a large-scale outbreak of the SARS and SBS (Sick Building Syndrome) viruses in a building's air conditioning system. An effective system is required to create a proper air ventilation system, balancing the energy needs and air quality. A liquid dryer air cooling is an alternative for the conventional air dehumidification technology as it can enhance air quality and reduce primary energy consumption. This study investigates a crossflow liquid desiccant dehumidification system. A structured packing consisting of a finned tube cooling coil is utilised in the experiment. Ionic liquid flows vertically through the cooling coil from top to bottom, while the air flows horizontally, making it a cross-flow configuration. Cooling water flows inside the tube. The effect of varying solution inlet condition flow rate and air on the system performance is presented in this paper.

Keywords: *Liquid desiccant, Ionic liquid, Dehumidification, Indoor Air Quality*

SESSION 1B

15. Performance of Rotating Detonation Engine withAsymmetric Vortex Combustion Chamber

A. Dairobi G., M. A. Wahid, M. A. Mazlan, M. H. Azeman

School of Mechanical, Faculty of Engineering, Universiti Teknologi Malaysia, Malaysia

Abstract

Rotating Detonation Engine (RDE) is a promising future engine for propulsion either aviation or ground power generation propose. RDE had high-efficiency and high thrust density without compression stages. Nevertheless, RDE is still at an infancy stage that need further study specifically to discover the effect of shape changeable to make it as a versatileengine. Thus, the aim of this research is to study the performance of RDE when using asymmetric vortex shape for combustion chamber. Variable mixing scheme was tested using methane-oxygen reactant. Initiation strategy by using pre-detonator with acetylene-oxygen mixture through 10bar of detonation pressure. RDE with asymmetric vortex shape for combustion chamber was successfully observed in three modes, failure, unstable and continuous detonation wave. The highest thrust generation is 17.7N at equivalence ratio 1.2.

Keywords:

SESSION 1B

21. Early Assessment of Thermal Analysis Rotating DetonationEngine

M.H. Azeman, M.A. Wahid, A. Dairobi G, M. Amri

School of Mechanical, Faculty of Engineering, Universiti Teknologi Malaysia, Malaysia

Abstract

Rotating Detonation Engine (RDE) has been proven for its high thermal efficiency and performance but with limited time operation because of the heat issues. This paper deals with early assessment of wall heat loads for thermal analysis of RDE. Initial experimental result will be shown for the uncooled rotating detonation engine, this RDE has successfully achieved detonation at various equivalence ratio reactant but with limited time operation because of the heat issues, the temperature produced to the wall of RDE is near to maximum critical temperatures of the mild steel wall of RDE. The wall temperature was observed to be in the range of 1100 °C .From the uncooled result, it will be redesigned to include active cooling system to tackle the heating issue with several parameters such as the mass flow rate of coolant and appropriate volume tank of coolant to be taken into consideration. Thus, the aim of this study is to determine the maximum duration of uncooled and cooled RDE to run at stable detonation condition.

Keywords:

SESSION 1B

31. Wave Propagation Characteristics in Pre-detonator on Rotating Detonation Engine Initiation

Muhammad Amri Mazlan, Mazlan Abdul Wahid, Ahmad Dairobi Ghazali and Hafizuddin Azeman

High Speed Reacting Flow Research Laboratory (HiREF), School of Mechanical Engineering, Faculty ofEngineering, Universiti Teknologi Malaysia (UTM), Johor Bahru, Johor, 81310, Malaysia

Abstract:

One of the important components in Rotating Detonation Engine (RDE) is the ignition system.The use of pre-detonator as an initiator has been presented by few researchers. They claim that the formation of detonation may occur in the annulus chamber of the RDE if not in the pre-detonator. Although formation of detonation may occur in the RDE, there are some concerns about the effectiveness of maintaining rotating detonation wave in RDE. The purposeof this research is to study the effect of detonation event in the pre-detonator to RDE initiation.Various parameters are investigated including type of fuel, equivalence ratio, ignition energy,and length of Shchelkin spiral. CEA program is used to determine suitable fuel and range of equivalence ratio to be use in the pre-detonator. Experiments were carried out to determine the effect of equivalence ratio, ignition energy, and Shchelkin spiral in pre-detonator on characteristic of the propagating wave. The formation of detonation wave near open end of tube tend to create pointed shape exit wave which contribute to the success rate of rotating detonation wave initiation in annulus chamber of RDE.

Keywords: *Rotating detonation engine, Pre-detonator, Ignition, Initiation*

SESSION 1B

35. A Modified Single-Step Chemistry Mechanism for Biogas Detonation Simulations

Mohammad Nurizat Rahman[1], Mazlan Abdul Wahid[1] and Mohd Fairus Mohd Yasin[1]

[1]High Speed Reacting Flow Research Laboratory (HiREF), School of Mechanical Engineering, Faculty ofEngineering, Universiti Teknologi Malaysia (UTM), Johor Bahru, Johor, 81310, Malaysia

Abstract

Due to the low computer resources needed, combining a simplified one-step global chemical reaction with a sophisticated computational fluid dynamics (CFD) solver is extremely desired for the engineering design of detonation engines. While many published research in detonation modelling have employed the well-known one-step chemistry model rather than the detailed chemistry, the majority of these studies focused on hydrogen-fuelled detonation engines and hence used a hydrogen-air/oxygen chemistry model. Many combustions engine research has begun to broaden the fuel flexibility capabilities by experimenting with renewable-based fuels such as biogas. Biogas has the potential to be one of the alternative fuels for combustion engines due to its renewable source. Therefore, there is the option of using biogas to power a detonation engine, which should be further researched. As a result, coupling a CFD solver to a simplified one-step global chemical reaction of biogas detonation is very desirable for the development of a functional biogas-fuelled detonation engine. Hence, the modified one-step chemistry for biogas detonation was established in this study by optimizing the temperature exponent value, *n*, to fit the reaction rate to actual biogas detonation experimental data. The results of biogas detonation simulations via the modified one-step mechanism agreed well with the experimental findings and the simulation using a detailed GRI Mech 3.0 mechanism, with 15.75% and 8.29% differences in detonation velocity, respectively. Thus, the ability of the modified one-step chemistry reaction to represent biogas detonation has been demonstrated, as the mechanism was able to predict detonation characteristics reliably and efficiently. All in all, the model would surely contribute to a more realistic situation during the design of a detonation-based engine, where a shorter simulation duration is preferred. When compared to experiment approaches thatare restricted to high-resolution research, the proposed mechanism allows for a more comprehensive examination of detonation features. Therefore, the established chemistry mechanism in this study has the potential to be integrated with a digital twin system for a future biogas-fueled detonation engine system to monitor the detonation performance as operating conditions were tweaked.

Keywords:

SESSION 1B

12. Numerical Simulation on A Modified Trapped Vortex Combustor

Dharsannen Sri Selvam[1], Norwazan Abdul Rahim[1], Mohd Rashdan Saad[1], Syahar Syawal[2], Hasan Mohd Faizal[3] and Mohd Rosdzimin Abdul Rahman[1, a)]

[1]Department of Mechanical Engineering, Faculty of Engineering, Universiti Pertahanan Nasional Malaysia, KemSg Besi, Kuala Lumpur, Malaysia

[2]School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, Malaysia

[3]Automotive Development Centre, School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310, Johor Baharu, Malaysia

Abstract

This study investigates combustion characteristics inside a modified trapped vortex combustor. The modified trapped vortex combustor is characterized by a variable guide vane ratio. The effect of the incoming velocity is analysed. The Arrhenius rate of the one-step methane/air reaction mechanism is used in this study. Numerical results demonstrated the significant changes of the vortex inside the cavity by introducing the guide vane. Moreover, variable guide vane ratio also has a significant effect on temperature distribution, NOx emission and combustion efficiency. Therefore, this study has improved the understanding effect of the guide vane towards the development of the high-efficiency low NOx trapped vortex combustor.

Keywords:

SESSION 1B

84. Influence of Equivalence Ratio on Emissions in Meso-Scale Vortex Combustor

Ali Houssein Asmayou[1], Mazlan Abdul Wahid[1], Mohammed Bashir Abdulrahman[1]

[1] High-Speed Reacting Flow Laboratory, Faculty of Engineering, School of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.

Abstract

This paper presents biogas numerical flame characteristics (60% CH4 + 40% CO2) in an asymmetric mesoscale vortex combustor. Lean limit is phi = 0.51 for all Re, the stable range is large at low Re (0.51<phi<1.86 at Re =2500), decreasing with increasing Re until (0.51<phi<0.56) at Re =7840. A strong vortex forms a stable zone for the flame and demonstrates the high stability of biogas flame for the mesoscale combustor. Flame stability in mesoscale vortex combustion is due to two important effects; the strong tangential vortex controls the flame field and vortex influence. The rich equivalence ratio, Ø=1.86, recorded the highest peak temperature at 1870 K. It produced the highest value for CO and NOx gases emissions, at 5.65 e-04 ppm and 10.06 ppm, respectively. Meanwhile, the lean equivalence ratio, Ø=0.56, recorded the lowest peak temperature at 1533 K but interestingly managed to produce the lowest value for CO and NOx gas emissions, at 1.43 e-07 ppm and 1.2132 ppm, respectively. Thus, it can be concluded that the higher ratio of air in the fuel, the lower the temperature and emissions.

Keywords: *Meso scale combustion, Flame combustion, Non-Premixed, Biogas*

SESSION 1B

4. Prospects and Challenges of Nanofluids as Improved Fuel for Diesel and Gasoline Engines: A Critical Review

Ahmed Sule [1, 2] Zulkarnain Abdul Latiff, [1] and Mohammed Azman Abbas [1]

[1)]Automotive Development Centre, Schoolof Mechanical Engineering, Faculty of Engineering, Universiti TeknologiMalaysia, 81310 Johor Bahru, Malaysia

[2)]Mechanical Technology Education Section, Technical Education Department, Kogi State College of Education,1033 Ankpa, Nigeria

Abstract

Climate change has been a global challenge which researchers have been trying to curtail over the years using different scientific approach. A major contributor to climate change is the combustion of fossil fuel which has influenced the search for alternative fuel sources among which is biodiesel. However, Biodiesel is associated with its limitations including higher nitrogen oxide (NOx) emission. This paper therefore present various recent findings and impacts of blending nanoparticles with biodiesel and conventional diesel, the merits and demerits in terms of power output, fuel consumption and emissions as well as the possibility of using nanofluids as future alternative fuel for diesel and gasoline engines.

Keywords: *Fossil fuels, Biodiesel, Nanofluids, Emission, Power output*

SESSION 2B

29. Effect of Nitrous-oxide on laminar burning velocity and flame stability of biogas combustion

Shehab Elhawary, Aminuddin Saat, Mazlan Abdul Wahid, Mohd Zarhamdy Md Zain

High-Speed Reacting Flow Laboratory, School of Mechanical Engineering, Faculty of Engineering, UniversitiTeknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.

Abstract

Biogas is a potential alternative energy source with low environmental impact. However, the practical applications of biogas are relatively limited due to the existence of CO2 which acts as a dilutant that decreases the burning rate of biogas. Nitrous oxide (N2O) is known to be a powerful oxidizing agent for propulsion applications which can enhance the combustion rate of biogas. In the present paper, the laminar burning velocity (LBV), and flame instability of Biogas/N2O oxide were experimentally studied at different equivalence ratios. The spherical propagating premixed flames for various mixtures of biogas-N2O were determined using the constant volume combustion vessel at 303 K and atmospheric pressure. Two mechanisms were used in CHEMKIN-PRO software in order to estimate the simulated LBVs of Biogas-N2O mixtures. The results indicate that the decline in LBVs was prominent in the fuel-rich mixtures than in the fuel-lean mixtures with CO2 dilution. It is found that the influence of curvature on the flame front is weakened at the fuel lean-to-stoichiometric mixture due to the decrease in the flame thickness, therefore, flame instability tends to increase at the lean-to-stoichiometric region. Sensitivity analyses were applied on Biogas-N2O mixtures, it was clear that the reaction of N2O decomposition is the dominant reaction in Biogas/Nitrous oxide combustion particularly with fuel-lean mixtures mixture ($\varphi = 0.6$).

Keywords: *Laminar flame speed, Nitrous oxide, Biogas, flame instability, Markstein length, Alternative fuels.*

SESSION 2B

19. Combustion Behaviour of Palm Kernel Shell Torrefied by Mild Pressurization Technique

Mohd Faizal Hasan[1, 2], Mohamad Zulhilmi Mat Ail[2] and Mohd Farid Muhamad Said[1]

[1] Institute for Vehicle Systems and Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor,Malaysia

[2] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

Abstract

Continuous supply of the costly inert gas such as nitrogen during torrefaction is undesirable due to cost burden in terms of operational cost. Therefore, mild pressurization technique is introduced to eliminate completely the dependence on inert gas for producing solid biofuel for power generation. In the present study, the combustion behavior of torrefied palm kernel shell (PKS) was investigated by using Thermogravimetric Analyzer (TGA) under temperature range of 30 to 900°C, heating rate of 10 and 30 °C/min, and oxygen flow rate of 20 ml/min. The used PKS has been torrefied by the mild pressurization technique for various residence times of 30 to 90 minutes and temperature of 300°C. Based on the thermogravimetry (TG) and derivative thermogravimetry (DTG) curves, it was found that the ignition temperature of the torrefied samples ranges from 276 to 285°C while the burnout temperature ranges from 476 to 482°C. For the PKS torrefied with longer residence time, the total percentage of mass loss during combustion stage becomes less. Finally, when the heating rate for TGA is changed from 10 to 30°C/min, almost same TG and DTG curves were obtained.

Keywords:

SESSION 2B

83. Spray Characteristic of Diesel Engine Injector by Using Palm Biodiesel as Fuel

Siti Nurhidayah Mohd Fauzi[1], Zulkarnain Abdul Latiff[1], Natrah Kamaruzaman[2], Mohd Farid Muhamad Said[2]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia.
[2]Automotive Development Centre, Institute for Vehicle Systems and Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia.

Abstract

In Malaysia, palm biodiesel with blend B7 and B10 has been widely used as a commercial fuel. Nevertheless, there are not many studies found on the performance of palm biodiesel with higher blending. This research is conducted to study the spray characteristics of different palm biodiesel blends by using existing diesel engine injectors. Two type of fuel injectors are used which are single hole and multi-hole injector together with three palm biodiesel blends which are B10, B20 and B30. Each biodiesel blend is tested with the approach of open space-type injection by using four varying injection pressures starting from 200 bar, 500 bar, 800 bar and 1,000 bar. In this case, palm biodiesel spray characteristics are analysed in terms of spray tip penetration length, spray cone angle and spray area through Computational Fluid Dynamics (CFD) involving three main stages which are pre-processing, simulation and post-processing. From the result of single hole injector, the range of spray tip penetration length starts from 1.58 cm to 4.47 cm, for spray cone angle is from 19.32° to 19.80° meanwhile for spray area is from 0.41 cm^2 to 3.38 cm^2. On the other hand, for multi-hole injector nozzle, the range of spray tip penetration length starts from 10.42 cm to 11.94 cm, for spray cone angle is from 76.6° to 78.1° and for spray area is from 52.70 cm^2 to 68.82 cm^2. It can be concluded that, multi-hole injector has better fuel spray formation compared to single hole injector. The spray tip penetration length, spray cone angle and spray area increase as the injection pressure increases. The overall best palm biodiesel blend performance is indicated through the widest fuel spray emitted which is from palm biodiesel blend B30 that is injected through multi-hole injector nozzle at the highest injection pressure of 1,000 bar.

Keywords: *Injector nozzle, biodiesel blend, spray tip penetration length, spray cone angle, spray area*

SESSION 2B

80. Density, Isothermal Compressibility & Isobaric Expansivity Measurement of Binary Mixture Containing Waste Cooking oil (WCO) &Propanol at 300.15 K

Sunita Rani[1], Nisha Sangwan[1], Rajesh K.Siwach[2]

[1]Baba Mastnath University, Rohtak, Haryana, India
[2]Department of Chemistry, Government College for women, Gohana, Haryana, India

Abstract

Continuous use of conventional fuel is harming the environment and in view of the limited number of resources, researchers are motivated to look for alternative sources of energy. Waste cooking oil can be a good alternative for conventional fuel as it can help in reducing the cost of machinery while protecting the environment. However, the thermal performance of waste cooking oil is not meeting the requirements. This study investigates the effect of thermophysical properties such as density, isothermal compressibility and isobaric relative expansivity on the mixing of propanol and WCO. Density of the mixture was measured by using specific gravity bottle over the mole fraction range of 0.1 to 0.9 at 300.15 K. This density data is then used to measure isothermal compressibility and isobaric expansivity. In this paper, the experimental density data is correlated with the Tait equation of state. PC-SAFT equation is used for the calculation of parameters and the prediction of densities of the WCO.

Keywords: *Waste cooking oil, Propanol, PC-SAFT, Tait- equation of state, Isobaric Thermal Expansivity, Isothermal Expansivity*

SESSION 2B

43. Impact of Biodiesel B30 on Engine Oil Viscosity in Real-World Agricultural Conditions

Mohd Rozi Mohd Perang[1], Mohd Azman Abas[1], Zulkarnain Abdul Latiff[1] and Mohd Farid Muhamad Said[1]

[1]Automotive Development Centre, Institute for Vehicle Systems and Engineering, Universiti Teknologi Malaysia,81310 Johor Bahru, Johor

Abstract

Biodiesel is a non-toxic and biodegradable fuel in which the combustion produces lower particulate matter, hydrocarbon emissions and smog. Due to its benefit to the environment, biodiesel has been adopted as a fuel additive in blends with petroleum diesel in compression ignition engines. In Malaysia, the existing diesel available at pump stations has biodiesel B7 and B10 fuel blends but plans to implement B20 soon, albeit with concern among the vehicle manufacturers. With the Malaysian government's plan to further increase the palm oil content in biodiesel, there is a need to study the impact of higher blends on the engine, components and system. This study focuses its investigation on the impact of biodiesel with B30 fuel blend on the engine oil viscosity in tractors under real-world agricultural conditions. Engine oil samples from a tractor were taken before and after 150 hours of operation for oil viscosity analysis following to ASTM D445 standard. Under similar agricultural conditions, the result shows that degradation of oil viscosity from B30 fuel blend was higher by 9.7% at 100°C and 11.7% at 40°C compared to B7 fuel blend after 150 hours of similar engine operations.

Keywords:

SESSION 2B

60. Nigeria Palm Oil Biodiesel Production Optimization as Post Covid-19 Green Energy and Economic Recovery Strategy

Ahmed Sule[1, 2], Zulkarnain Abdul Latiff [1]and Mohammed Azman Abbas[1]

[1] Automotive Development Centre, School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Malaysia
[2]Mechanical Technology Education Section, Technical Education Department, Kogi State College of Education, 1033 Ankpa, Nigeria

Abstract

In recent times, climate change has significantly affected the world by creating devastating challenges like flooding, desert encroachment and unpredictable ecosystem in the globe and African continent like others have experience these issues. Fossil fuel combustion is proven to contribute to climate change however, biodiesel as a renewable and green energy identified as a factor in mitigating the problem has not been well developed by many African countries including Nigeria despite abundant availability of raw materials unlike US, China, Malaysia, and Indonesia that have developed and improved biodiesel production industries. The emergence of Covid-19 pandemic has caused economic recessions globally therefore, this paper presents major issues affecting the development of Nigeria's palm oil industry that significantly affects palm oil biodiesel product ion, suggest possible solutions to the industry stagnation and finally, provide palm oil biodiesel production optimization strategies to cushion the negative economic impact of Covid-19 and simultaneously contributing to world's green energy initiative.

Keywords: *Ecosystem, Green Energy, Covid-19, Nigeria, Biodiesel*

SESSION 2B

90. Study on The Optimal Stack Diameter on The Efficiency of Thermoacoustic Engine

Irna Farikhah[1], E. A. Elsharkawy[2], Harto Nuroso[3], Mega Novita[4], Dian Marlina[5], Kwartriani Rahmatunnisa[6], Irfan Abd. Rahim[7], Mohd Zarhamdy bin Md Zain[8], Khairunnisa Mohd Paad[9], Ahmad Nadhif Masruri[10]

[1] Department of Mechanical Engineering, Faculty of Engineering and Informatics, Universitas PGRI Semarang. Sidodadi Timur No.24, Central Java 50232, Indonesia
[2]Department of Interdisiplinary Graduate School of Engineering Sciences, Kyushu University 744 Motooka Nishi-ku Fukuoka 819-039, Japan.
[3]Department of Physics Education, Faculty of Natural Sciences and Informatics, Universitas PGRI Semarang
[4]Department of Electrical Engineering, Faculty of Engineering and Informatics, Universitas PGRI Semarang, Jl. Sidodadi Timur No.24, Kec. Semarang Tim., Semarang, Central Java 50232, Indonesia.
[5]Department of Pharmacy, Faculty of Pharmacy, Universitas Setia Budi, Jl. Letjen Sutoyo, Mojosongo, Kec. Jebres, Surakarta, Central Java 57127, Indonesia
[6]Senior High School (SMA) Generus Mandiri, Bogor, Indonesia
[7]Faculty of Mechanical EngineeringTechnology, Universiti Malaysia Perlis, Kampus Tetap Pauh Putra, 02600 Arau, Perlis, Malaysia
[8]School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Johor Malaysia
[9]Department of Chemical and Environmental Engineering, MJIIT, Universiti Teknologi Malaysia Jl. Sultan Yahya Petra, Kampung Datuk Keramat, 54100 Kuala Lumpur, Malaysia
[10]Department of Mechanical Engineering, Faculty of Engineering, Universitas Diponegoro Jl. Prof. Sudarto No.13, Tembalang, Semarang, Central Java 50275, Indonesia

Abstract

The scarcity of energy resources has been attracting some researchers to find some renewable energy alternatives. On the other hand, there are huge waste heat from industries which are harmful for our environment. The waste heat can be converted into useful energy using thermoacoustic technology such as the thermoacoustic engine. There are parameters that have an impact on engine efficiency such as the stack diameter of the thermoacoustic engine. Therefore, the aim of this research is to find the optimal stack diameter. The investigation was conducted numerically. It was found that the optimal stack diameter is 2 cm and the efficiency of the thermoacoustic engine is 36 %.

Keywords: *Thermoacoustic Engine, stack diameter, efficiency*

SESSION 3B

23. Comparison between Computational Fluid Dynamics and Fluid-Structure Interaction Models of an Automotive Mixed Flow Turbocharger Turbine

Noor Zafirah Abu Bakar[1,2] and M.H. Padzillah[1]

[1]UTM Centre for Low Carbon Transport in Cooperation with Imperial College London, Faculty of Mechanical Engineering,Universiti Teknologi Malaysia, Johor Bahru, Malaysia

[2]School of Computer Science and Engineering, Taylor's University, 1, Jalan Taylors, 47500 Subang Jaya,Selangor¸Malaysia

Abstract

Engine downsizing with turbocharger is one of the technologies being adopted to reduce carbon dioxide emission. Nevertheless, turbocharger is yet at its fullest efficacy thus drives researchers to study on its system, for example the turbine rotor. One of the key areas of the study is the inlet flow characteristics of the turbine rotor as there have direct influence on the turbine efficiency. There have been multiple studies on the inlet flow but all of them considered only the fluid domain whereby in actual condition, there are interactions between fluid and solid domains, which in this case are turbine rotor blades. The objectives of this study are to compare two numerical models, a non-coupled (NC) and two-waycoupled (2-WC) fluid-structure interaction simulation models on turbine rotor inlet flow characteristics and to assess theirsignificance. Experimentally validated simulation models were developed using ANSYS CFX, Mechanical Static Structure and System Coupling which function as an interface between fluid and structure domains at 50% of rotor speed. Flow characteristics in terms of flow angles and velocity components were analyzed at four different locations of rotor inlet circumference and at 10% spanwise plane. Results shows that the difference of absolute and incidence angle between NC and 2-WC models are more apparent at tongue which has non-uniform flow. At 10% streamwise plane, the flow characteristics and velocity components display significant variations close to the shroud because of a localized low- pressure region at suction surface in NC model. Consistent differences between NC and 2-WC models at suction and pressure surfaces of the blade are caused by Arbitrary Lagrangian-Eulerian formulation in 2-WC model. Blade deformationbetween the two simulation models is negligible but shows significant difference in equivalent stress value which 2-WC model recorded higher value. It is concluded that if the study related to overall performance of turbine, a non-coupled simulation model is sufficient as the outcome will have minimal difference compared to coupled simulation. In addition, coupled simulation model is computationally expensive. However, if the study is specifically related to tongue area or structural performance, thus fully coupled simulation model should be considered for better accuracy.

Keywords:

SESSION 3B

24. Secondary Flow Characteristic inside a Mixed Flow Turbocharger Turbine Volute at Different Load Conditions

A.Azmi and M.H. Padzillah

UTM Centre for Low Carbon Transport in Cooperation with Imperial College London, School of Mechanical Engineering,Universiti Teknologi Malaysia, Johor Bahru, Malaysia

Abstract

This paper presents an experimentally validated numerical analysis on the influence of different loads toward secondary flow characteristics inside a mixed-flow turbine volute. Two volute designs with constant A/R (ratio of volute area to centroid radius) but with different volute aspect ratios (VAR) (b/h) namely VAR05 and VAR15, have been considered for this analysis. The flow characteristics are analysed on this rectangular volute plane cross-section at 0°, 90 °, 180°, and 270° positions under steady-state conditions. Secondary flow characteristics are identified based on pressure contours and velocity streamlines at these positions. The results show significant differences in the secondary flow characteristics at low and high loads. As the load increases, the pressure increases that cause the deflection of primary streamwise flow to become stronger and at certain positions, flow separations occurred. It is observed that a weak local re-circulation flow at the centre lower partof 90 ̊plane position creating a strong secondary couple vortex flow structure. This paper shows that the secondary flow structures that exist in the volute are highly influenced by different loads, thus affecting the turbine performance.

Keywords:

SESSION 3B

32. Three-Dimensional Elastic Cantilever Plate Attached to a Solid Block Using FSI Solver in Open FOAM Technology

M. S. Al Manthari[1] and P. Nithiarasu[2]

[1]Department of General Requirements, University of Technology and Applied Sciences, Ibri 516, Oman.

[2]College of Engineering, Swansea University, Bay Campus, SA18EN, Swansea, UK.

Abstract

This test model of three-dimensional (3D) numerical approach follows the two-dimensional (2D) example model presented in many literatures. There is an unstable manner could be expected by this system like the sharp angles of the solid blockwhich lead to high vorticity in the fluid flow through the slender plate. The work done in this paper is focused on the use of foam-extend project. Along with this project, the advanced fluid-solid interaction (FSI) was also used. The FSI problems show a strong coupling between both fluid and structure due to the forces acting from fluid to solid and later causing the structural geometry deformation. In the 3D flow for FSI problems, it is not easy to select the relevant results. Since the foam-extend 4.0 version is usedfor FSI solvers, there are some problems were faced during run that case. It could be noted that the results do not show full agreement between the case model presentations. It is also important to note that the behaviour of the flexible plate in this work and in other studies was tested with transitional displacement and no rotation in the solid. However, the results obtained in other literature present the transitional and torsional rotation. As for the overall disagreement in the results between this work and the other studies, this may very likely be related to what was diagnosed in the 2D plate simulations. Specifically, it was assessed howclosely tied is the mesh refinement/resolution to the response time of the plate and that even though the fluid flow is laminar (Reynolds numbers well below 1000), the presence of vortices indicates that a turbulence model should still have been used for properly modelling their presence without the need to increase drastically the mesh refinement/resolution. However, there were some situations where the modelling limitations revealed that even if the flow pattern is considered a laminar flow (due to the smallReynolds number and/or due to the clear shaped vortices), turbulence modelling or fairly more refined meshes should be used in order to properly capture the generation and release of vortices. A full 3D simulation does not necessarily imply that it will always reveal non-symmetrical flow profiles, as also revealed in the elastic cantilever plate attached to the solid block, even though it should possibly have revealed that torsion should have occurred.

Keywords:

ESSION 3B

91. Numerical Investigation into Trapezoid Surface Texture of Journal Bearing and RBD Palm Oil as Lubricant

Zuraidah Rasep[1,2], Muhammad Noor Afiq Witri Muhammad Yazid[1] , Syahrullail Samion[1] and Nor Azwadi Che Sidik[3]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.
[2]Process Engineering Technology Section, Malaysian, Institute of Chemical and Bioengineering Technology (MICET), Universiti Kuala Lumpur, 78000 Alor Gajah, Melaka, Malaysia.
[3]Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Malaysia.

Abstract

Surface texture modification of journal bearings has sparked the interest of researchers in this field because it has the potential to improve journal bearing performance. Although many geometries have been studied, an introduction of surface texture in the geometry shape of trapezoid still has not been explored by researchers and scientists. This case study also explores the potential of RBD palm oil in the application of modification of surface texture in a journal bearing. RBD palm oil acts as a renewable source of lubricant which has a high potential for replacing conventional oil (engine oil) in the future. In this case study, the method of analysis chosen was a CFD ANSYS FLUENT. Three different geometries were introduced at the location of divergent A and rotational speed of 1000 rpm and eccentricity ratio 0.7. As a result, geometry 1 at the location of divergent A enhances journal bearing effectiveness at 800 and 1000 rpm, while geometry 2 (trapezoid shape) increases journal bearing performance at 800 rpm. Both cases were assessed with a constant eccentricity ratio of 0.7. Moreover, by modifying the chemical structure of RBD palm oil, it has a high ability to replace engine oil in the future.

Keywords: *Palm oil, Journal bearing, CFD, ANSYS FLUENT, Tribology, Surface texture*

SESSION 3B

5. Flame-front Detection Using High-Speed Chemiluminescence and 2D Mie-scattering Imaging via Endoscopic Aspect

Syahar Shawal[1], Mohd Rosdzimin Abdul Rahman[2], Suhairil Meon[1], JuriSaedon[1], Mazwan Mahat[1]

[1]School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, Malaysia

[2]Department of Mechanical Engineering, Faculty of Engineering, Universiti Pertahanan Nasional Malaysia, Malaysia

Abstract.

This study aims to contribute knowledge of diagnostic techniques for visualizing and detecting premixed flame propagation in spark-ignited engines through chemiluminescence and laser-based techniques. In the present work, combined measurements of high-speed broadband chemiluminescence and phase-locked 2D Mie scattering imaging wereconducted to image the flame-front in a production engine via endoscopic access. An in-house detection algorithm was developed to post-process the raw images in order to extract quantitative information such as flame contour, flame area, and flame propagation speed. Together with the corresponding cycles' pressure traces, a wealth of detailed information on combustion can be acquired in little time. However, CMOS cameras that are designed for these very high frame rates exhibit much more (and in some cases less repeatable) read-out noise than cameras based on CCDs or noise-optimized "slow" CMOS detectors. Similarly, image intensifiers for kHz repetition rates degrade image resolution, dynamic range, and photometric accuracy even more than their "slow" counterparts for applications with typical video rates of up to 100 Hz. In view of these limitations that high-speed camera systems currently have, the present work explores how well details of the spark are discernible here. The results showed that the broadband chemiluminescence high-speed imaging provides details of the spark and the ignition event, therefore this could be a significant advantage over Mie-scatter imaging. The lack of an intensifier-induced "halo" means that the flame kernel can be distinguished from the spark earlier. However, without the analog amplification of the intensifier upstream of the CMOS detector the signal is much closer to the read-out noise of the detector. The results also showed that chemiluminescence image always yields a larger burnt area than Mie-scatter image due to the line-of-sight projected signals. The flame growth derived from chemiluminescence images shows a good correlation with pressure-derived heat release with a correlation coefficient of 0.72. However, the Mie-scattering images provide additional information on in-cylinder turbulent flow-field, bulk flow velocity, and the local wrinkled turbulent flame propagation.

Keywords: *Flame-front, High-speed imaging, Chemiluminescence, Mie-scattering, Endoscopy*

SESSION 3B

57. Development of Cost-Efficient Schlieren Flow Visualization Using Short Focal-Length Spherical Mirror

Yahya Dimas Z[1], Harinaldi[1], and Ridho Irwansyah[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia
Kampus Baru UI, Depok, West Java, 16424, Indonesia

Abstract

Optical flow visualization is a method that utilizes refractive index gradient for seeing the disturbance of air in transparent media. The technique that uses this method is the schlieren technique, the simplest optical arrangement in this technique is single mirror off-axis schlieren system. However, until recently the usage of cost-efficient spherical mirror made by acrylic in this optical arrangement is still lacking. This research will focus on a single mirror off-axis schlieren system using a spherical mirror with 600 mm diameter and 584.2 mm focal length. The observed object will be varied based on the density difference between the air around it and the atmospheric air. In this study, the effective field-of-view obtained will be observed as well as the resulting image contrast and its relationship with the camera's ISO. The resulting effective field-of-view depends on the geometry and surface mirror quality. The highest image contrast is obtained from the Schlieren effect which caused by mixing different densities gases, especially for butane gas, while the lowest image contrast is obtained from the Schlieren effect caused by temperature changes in the air. The recommended camera's ISO depends on the type of phenomenon being observed.

Keywords: *Flow Visualization, Optical Flow, Visualization, Schlieren Technique, Mirror-type Schlieren System*

SESSION 3B

41. Entropy in Macroscopic Thermodynamics: How Should It BePresented and Understood?

Ahmed M. Bufares

Mechanical Engineering Department, Faculty of Engineering, University of Benghazi

Abstract

Entropy is one of the important concepts in physics and thermodynamics and it is one of the quantities surrounded by amount of mystery. The ambiguity of this concept is due to several reasons, including the mathematical and physical interpretation of the concept as presented by Clausius. This ambiguity is also due in part to the widening of the use of the term entropy to represent several physical concepts that are not related to the basic concept of entropy as introduced by Clausius himself. In this paper, the mathematical and physical modeling of this term will be reviewed. Furthermore, this paper will show how does this term has been used to describe some physical concepts which are not related to the original use of it. And finally, in order to remove the confusion and ambiguity surrounded this term, the paper will show how should the entropy explained and understanding from the macroscopic thermodynamics point of view.

Keywords:

SESSION 4B

67. Tribology Characteristic of Cartilage Replacement using Pin on Disc Experiment

M Mazwan Mahat[1], S Kasolang[1], Nurul Nadiah Mohd Kamaldin[1] and Mohd Syahar Mohd Syawal[1]

[1] School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA
40450 Shah Alam, Selangor, Malaysia

Abstract

This study focuses on the performance and mechanical response of thermoplastic polyurethane (TPU), polylactic acid (PLA) and acrylonitrile butadiene styrene (ABS) as a tibial insert for cartilage replacement in total knee replacement (TKR) experimentally. As known, cartilage replacement has experienced a high successful procedure but there is still wear and fatigue problems of the materials used for the implants. Thus, this study aims to investigate the behaviour and response of these materials disc under different type of human activities within the same load conditions. The discs were made by using the 3D printing method for both PLA and ABS while for TPU is cut from a 500 mm x 500 mm x 4 mm. The analysis was conducted using the Pin on Disc Wear and Friction test rig by inserting the pin apply to 4mm thickness disc model design to mimic the mechanical principles of compression load on the knee prosthesis. The experiment was conducted to study the morphology of the disc materials to observe the effect of wear rate on the proposed materials. It is found that the wear rate is strongly correlated with different types of human activities.

Keywords: *Cartilage, Biomaterials, Pin-on Disc method, Tribology*

SESSION 4B

26. The Impact Critical Characteristic of Ethanol Contaminant on Lubricant Properties Vehicle SI Engine

Rona Malam Karina[1], Milda Fibria[1], Catur Y Respatiningsih[1], M.Hanifuddin[1,2], Riesta Anggarani[1], LiesAisyah[1], Setyo Widodo[1], Rizkia Malik[1], May Muchar[1], Dimitri Rulianto[1], Cahyo Setyo Wibowo[1]

[1]Department of Product Application Technology, Research and Development Center for Oil and Gas Technology "LEMIGAS",Jakarta, Indonesia.

[2]Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok, Indonesia.

Abstract

Methanol has been widely used as an alternative fuel. Several studies reporting the usage of methanol as an alternative fuel that likely will influences the performances of lubricating oil. The presence of contaminants in lubricating oil was suspected to be the cause of lubrication failure. The key properties of lubricants were analyzed to observed these phenomena. This study aims to determine the lubricant contaminant, and the lubricating efficiency of lubricating oil diluted with various methanol-gasoline mixtures. The results show that the properties of lubricants change as fuel diluted into the lubricating oil,namely kinematic viscosity, density, flash point, total acid number, total base number, water content, and wear tendency. The kinematic viscosity changes about 17.5% from the initial value, while the density, flash point, and TBN change were changed 0.5%, 70%, and less than 1%, respectively. The relatively small decrease of kinematic viscosity indicates the ability of lubricating oil to maintain the lubrication required. This condition was observed in the measured wear scar diameter which is still below the allowable value of 0.5 mm. The acidity (TAN) increases significantly, however, the value was still below 2 mg.KOH/gr. The water content of lubricating oil raises as the presence of water in fuel was increases and scar diameter. This is aligned with the increase in the acidity value of lubricating oil. According to those results, the usage of methanol as a fuel blend is still possible. However, it should be noted that the increase in water content, in the long term, is suspected to cause lubrication failure. This is because the presence of water can trigger an increase in the acidity and corrosiveness of the lubricant.

***Keywords**: methanol, fuel, lubricant*

SESSION 4B

53. Comparison of Active and Passive Controls to Prove the Degradation Time

Sealtial Mau[1], Yanuar[2], Achmad Riadi[2],

[1] Department of Mechanical Engineering Education, Faculty of Teacher Training and Education, Universitas Nusa Cendana, Kupang 85001, East Nusa Tenggara, Indonesia
[2]Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, Indonesia

Abstract

Flow control is an important topic because of its contribution in energy saving in piping system. The choice of control technique of flow is no less important to be applied in the right case. One of the impacts of this application is the efficiency of degradation time which can occur at any time. Therefore, the purpose of this study is to reveal the degradation time of two types of flow control. The research applied a piping system with a horizontal position. For the working fluid, it uses a mixture of nata de coco fibres for active control and spiral pipes for passive control with the slurries as the working fluid. Both pipes used were the three lobes spiral pipe and the pentagon spiral pipe with the average size P/Di = 7. The results revealed that the active control had a variation of the drag reduction value with the increasing time indicating that degradation occurred while the passive control had a constant value. The results of this study are very important to determine the type of flow control that will be used according to the case handled and the results to be obtained.

Keywords: *Non-Newtonian, Flow control technique, Pressure drop, Spiral pipe, Drag reduction*

SESSION 4B

72. A Common Platform for Evaluating Energy Efficiency of Desalination Plants

Kim Choon Ng[1], Muhammad Burhan[1], Qian Chen[1], Doskhan Ybyraiymkul[1], Faheem Akhtar[1], M Kum ja[1], Raid AlRowais[2], Muhammad Wakil Shahzad[3]

[1] Water Desalination & Reuse Center, King Abdullah University of Science & Technology (KAUST), Saudi Arabia (KSA)
[2]Dept. of Civil Engineering, Jouf University, Saudi Arabia (KSA)
[3]Dept. of Mechanical & Construction Engineering, Northumbria University, Newcastle, UK.

Abstract

The energy efficiency (EE) of seawater desalination processes is usually expressed in terms of the consumption of derived energy, either kWh electricity or low-grade heat per m3 of water produced. Nevertheless, the conventional figure of merit(FOM) of EE is not a law of Thermodynamics but is merely a ratio of useful output to energy input. These FOMs have unfortunately omitted the embedded quality of derived energy (DE) input, underlying their generation methods. To avoid a thermodynamic misconception, it is important that both quantity and quality (Q&Q) of DE input are equally recognized. An inadequate efficacy analysis, based on unmerited quantitative apportionment, may result in an unjust comparison of energy efficacy across the desalination methods. This article clarifies the misconception of seeming parity between the quantitative units of electricity and thermal heat sources. A common energy platform of standard primary energy of heat supply is proposed, where the DE consumed in the EE definition are converted to the causative common platform of heat supply, giving a rigorous thermodynamic framework of heat and reverse heat engines. We examined the specific energy efficiency from more than 60 plants to demonstrate the novel concept of standard primary energy approach.

Keywords: *Desalination, Energy efficiency, Thermodynamic Platform, Standard Primary energy.*

SESSION 4B

70. The Impact of Operational Flexibility on Electricity Production: A Case Study of a Waste-to-Energy Pilot Project in Bantargebang, Indonesia

Priska A Hendrayanto[1], I Putu A. Kristyawan[1], Muhammad Hanif[1], Wiharja[1]

[1] BPPT Center of Technology for the Environment, Agency for the Assessment and Application of Technology (BPPT), Geostech Building 108, Kawasan Puspiptek, Serpong, Tangerang Selatan, 15314, Indonesia

Abstract

A 100 ton/day Waste-to-energy (WTE) in Bantargebang, Indonesia is a national pilot project under collaboration between BPPT and the Province of DKI Jakarta. The WTE process using incinerator system produces electricity using a 750 kW of steam turbine generator. Currently, electricity production is around 350 kW because it follows the demand, only for internal needs. Properties of MSW as a feedstock is crucial to maintain the stable operation of the WTE plant. In order to meet with the properties MSW in Indonesia, the system of WTE plant requires significant flexibility in operation. This work evaluates the impact of operational flexibility in Bantargebang's WTE to optimal electricity production. Data used in this work were obtained daily during operation from January to June 2021. Some changing in MSW properties, especially in moisture content were evaluated to find the best practices for maintaining the stable operation. The results show the MSW has high moisture content, and the electricity production was more than 105 kWh/ton. Operational flexibility during the altering of properties of MSW feedstock was examined in order to understand its impact on the electricity production. Best practices from this work are potential to become a routine operational procedure of WTE in commercial scale.

Keywords: *Electricity, incineration, moisture content, operational flexibility, waste to energy*

SESSION 4B

9. Open LCA Life Cycle Assessment Tool to Determine Environmental Life Cycle Hot Spots: Demonstrated to an Aluminum Patrolling Ship

Mohd Zamri Mohd Nor [1] and Md Mizanur Rahman [1]

[1]School of Mechanical Engineering, Faculty of EngineeringUniversiti Teknologi Malaysia (UTM)
81310 Johor Bahru, Johor, Malaysia

Abstract

In this study, the life cycle assessment methodology has been used to assess the environmental impact of the life cycle ofan aluminium patrol vessel of 25 meters in Malaysia. Three main groups in the life cycle of this ship have been identified namely construction, operation, and disposal. In this analysis, this ship is considered to use the principle of cradle to cradle, means that thisship is built from raw materials and will be recycled after the ship is decommissioned. This ship was built in Perlis Marine Sdn. Bhd. with a 25m long frame with a displacement weighing 25 tons, the main material of this ship is Aluminium and has 2 main engines. The life cycle of this ship for operational purposes is estimated at 25 years. This ship uses diesel fuel as the main fuel to generate mechanical energy to move the ship. Life cycle analysis is performed with the help of Open LCA software and ReCiPe method is used for the life cycle impact assessment phase. The results of the analysis show that the operating part has given the highest impact value with readings climate change (GWP100) 1.36048×10^{8} kg CO2-Eq compared to the construction part with readings climate change (GWP100) 1.70506×10^{6} kg CO2-Eq. A comparative analysis was also conducted by comparing the originalengine to the new model to be identified as having a low impact on the environment when compared to the original engine.

Keywords:

SESSION 4B

22. Techno-Economic Readiness and Acceptance Analysis Studyof End-of-Life Vehicles (ELV) Implementation in Malaysia

Ahmad Nizam Che Kasim[1], Nurhidayah Mat Hashim[2], Noor Azuan Hashim[1],Zambri Harun[2]

[1]*Centre of Value Creation and Human Well-beingFaculty of Economics and Management*

[2]*Department of Mechanical and Manufacturing EngineeringFaculty of Engineering and Built Environment 43600 UKM Bangi, Malaysia*

Abstract

End-of-Life Vehicles (ELV), raised as a debatable and concerning urgent matters among experts and public alike in Malaysia recently. A comprehensive and efficient ELV related policies implementation cannot be successful unless a majority of various groups ofpeople, together with significant Government interventions, support these policies. This article is aimed to quantify and explore urgency of issues arising from the implementation process and the significant contributions of these policies, through analysis of factors that affect public acceptability and the impact of business ecosystem using techno-economic approach. Methods adopted in this study comprised of a thorough literature review and primary data analysis, through survey conducted among the stakeholders. From the initial data collection, significant number of respondents are still unsure on the ELV term (26.7%),proper process of ELV (50%) and 80% of industry players surveyed, agreed that the cost ofthe ELV process should be borne by owner of the vehicles.

Keywords: *End of Life Vehicles (ELV); Remanufacturing; Techno-economic; social-psychological*

SESSION 5B

36. Optimization of Microchannel Heat Sink for Thermal Performance and Pressure Drop using Central Composite Design of Experiment

Mohamad Nur Hidayat Mat[1], Normah Mohd-Ghazali[1,], Hielfarith Suffri Shamsuddin[1] and Patrice Patrice Estellé[2]

[1]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai 81310 UTM Johor Bahru, Malaysia

[2]Univ Rennes, LGCGM, F-35000 Rennes, France

Abstract

This research focuses on analytical optimization approach in maximizing thermal performance and minimizingpressure drop across a rectangular microchannel heat sink using central composite design of experiment with a constant volume flow rate of 4.6 cm^3/s water as coolant. A single and symmetrical microchannel heat sink was created and discretized for numerical computational fluid dynamic (CFD) analysis. An optimization scheme was applied using the response surface method (RSM) to generate 2500 design points against multiple objectives. The selection of the best candidate points fulfilling design criteria was made using an elitist non-dominated sorting genetic algorithm (NSGA2). The model was validated against a previous literature's experiment, and the results were aligned with the present validation study. The outcome of this research results in improving thermal performance for up to 17.27% and minimizing pressure drop by 53.29% from the based design across the channel with optimization value of channel aspect ratio and wall width ratio of 1.11 and 0.0014, respectively. This significant increment of temperature difference due to the highest surface area contact between fluid and silicone consists of 143 channels more than the previous design model,which was 100 channels. The present outcome reveals the novelty approach in improving an existing application of a microchannel heatsink design model water-based coolant by saving pumping power consumption and reducing time consumption to dissipate large heat from a small area.

Keywords:

SESSION 5B

63. Comparison of 3D Modelling of Single Phase and Two-Phase Flow of Nanofluid through Corrugated Channels

Elhadi Kh. Abugnah[1], Wan Saiful-Islam Wan Salim[1] and Abdulhafid M Elfaghi[1]

[1] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, Malaysia

Abstract

A comparison between single phase and multi-phase modelling for the turbulent forced convection flow of nanofluid through semi-circle corrugation channel is implemented numerically to investigate the difference between both models. Ansys Fluent software is used to analyse the effect of suspension of nano particles of SiO2 with two volume fraction 0.02 and 0.08 into water as a base fluid and to determine the difference between the two models regarding to the display of the behaviour of thermohydraulic characteristics of the convective heat transfer flow through the channels. One single-phase model (SPM) besides two three multi-phase models, Eulerian Eulerian Model (EEM), Eulerian Mixture Model (EMM), and Volume of Fluid (VOF) are conducted for the same geometry, same meshing generations and same boundary conditions at Reynold's numbers range from 10000 to 30000. Single-phase and two-phase models predict nearly identical hydrodynamic and thermal fields at 0.02 volume fraction but at volume fraction 0.08 a slight difference at low Reynolds Number and the difference increase with the increase of Reynolds Number according to the findings. The three two-phase models make essentially the same predictions. The two-phase models predict the convective heat transfer coefficient closer to the experimental data than the single-phase model for the problem at hand; however, the two-phase models over-predict the enhancement of the convective heat transfer coefficient caused by an increase in the silica volume fraction.

Keywords: *Corrugation, nanofluid, single phase, multiphase, turbulent flow, heat transfer analysis*

SESSION 5B

82. Optimization of a Boron Nitride Nanotubes Nanofluid-Cooled Microchannel Heat Sink at Different Concentrations

Nur Liyana Nabihah Yusof [1], Hielfarith Suffri Shamsuddin[2], Patrice Estellé[3], Normah Mohd-Ghazali[1], Ummikalsom Abidin[1]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia
81310 UTM Johor Bahru, Johor, Malaysia
[2]Univ Rennes, LGCGM, F-35000 Rennes, France

Abstract

Development of Internet of Things (IoT) requires more data storage and data processing for the digital revolution to take place as automation and digitalization require information to be continuously stored and processed. Consequently, heat dissipation from the microchips - the microprocessor and microcontroller - are getting higher with the sizes getting smaller. The microchannel heat sink (MCHS) has become the most relevant micro-heat exchanger for a small area of high heat removal system. An effective coolant is needed to address the increasing heat load from the microchip and nanofluid, nanosized particles dispersed in a base fluid, is among those explored. This paper reports the outcomes of an optimization approach using multi-objective genetic algorithm (MOGA) to investigate the thermal and hydrodynamic performance of a boron nitride nanotube (BNN) nanofluid-cooled MCHS at concentrations of 0.001%, 0.003%, 0.005%, 0.01% and 0.03%. As the concentration of BNN nanofluid increases, the total thermal resistance of the MCHS decreases, lower by 5.34% at 0.01 concentration, compared to that of water at the same temperature. The lowest thermal resistance for BNN nanofluid-cooled MCHS investigated is 0.071124°C/W at 0.01% of volume concentration. The experimental thermophysical properties obtained at 50°C provides reliability to the optimization modelling outcomes in identifying the best BNN concentration for cooling of a MCHS.

Keywords: *Microchannel heat sink, multi-objective genetic algorithm, optimization, boron nitride nanotube nanofluid*

SESSION 5B

73. Interpolated Thermophysical Properties for Minimum Thermal Resistance of a Microchannel Heat Sink

Hielfarith Suffri Shamsuddin[1], Patrice Estellé [1], Normah Mohd-Ghazali[1], Ummikalsom Abidin[1], Maziah Mohamad[1].

[1]School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia
81310 UTM Johor Bahru, Johor, Malaysia

Abstract

Nanofluids have been at the forefront of research into coolants for the thermal management of micro-electronic systems. The increased heat removal capabilities with the addition of high thermal conductivity nano-sized particles into the base fluid have been established to cause this improved property. Numerous simulations of nanofluid-cooled microchannel heat sink (MCHS) have been completed with and without experimental data, the former better at providing reliability to the models. Experimental data, however, may not be available at any concentration. This study reports an analysis of the thermal resistance of a boron nitride nanotube (BNN) nanofluid-cooled MCHS at concentrations interpolated to be where maximum thermal properties - conductivity and heat capacity - are found. Interpolations were done with five experimental data points and 98%-99% of data points curve-fitted. Results showed that the thermal resistance of the BNN nanofluid-cooled MCHS at 50°C was minimum at 0.0022% concentration, 0.1043 W/K. The outcomes of the study indicate the possible attainment of a better thermal performance of a nanofluid-cooled MCHS with adequate experimental data to be curve-fitted, consequently identifying the best concentration for specific applications.

Keywords: *Microchannel heat sink, multi-objective genetic algorithm, optimization, boron nitride nanotube nanofluid*

SESSION 5B

87. Supercooling Effects on The Drag of a Free-Falling Sphere

M. S. Mohamad[1], A. R. Irfan[1,2] and M. Z. Md Zain[3], C. M. Mackenzie Dover[4], K. Sefiane[5]

[1] Faculty of Mechanical Engineering Technology, Univeriti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.
[2]Green Design and Manufacture Research Group, Centre of Excellence Geopolymer and Green Technology (CEGeoGTech), Universiti Malaysia Perlis, 01000 Kangar, Perlis, Malaysia
[3]School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor Bahru, Johor, Malaysia
[4]Department of Biomedical Engineering, The University of Texas at Austin, Austin, TX, USA
[5]School of Engineering, University of Edinburgh, King's Buildings, Edinburgh EH9 3BF, UK

Abstract

In recent years, various techniques have been researched to form gaseous plastron on solid surfaces. Particularly, on a submerged moving object, plastron layer has been widely investigated for drag reduction purpose. In the present study, the free-falling of supercooled stainless-steel spheres is experimentally investigated. The spheres were supercooled by immersing them in liquid nitrogen in an attempt to produce plastron layer for drag reduction effect, but the drag was increased by this treatment. This was due to the presence of air bubbles trapped in the wake of the spheres which promoted instabilities. Consequently, the wake lost its symmetry, and the drag was increased.

Keywords: *Plastron, Drag reduction, Free-falling, Sphere*

SESSION 5B

93. Indirect Evaporative Cooling System Improvement: A Review

Mohd Fahmi Md Salleh[1], Mazlan Abdul Wahid[2]

[1] School of Mechanical Engineering, College of Engineering, Universiti Teknologi MARA, Malaysia
[2]School of Mechanical Engineering, Universiti Teknologi Malaysia, Johor Bahru, Johor, Malaysia

Abstract

Evaporative cooling has caught the attention of many researchers due to its environmentally friendly characteristic since it uses water evaporation during its process. One of the promising evaporative cooling systems is indirect evaporative cooling (IEC). Various studies have been done by previous researchers to understand the characteristics and improvements towards the performance of IEC. There are many factors influencing the performance of an IEC such as the channel length, channel gap, air temperature, air humidity and air velocity. However, it is noticeable that IEC might face insufficient evaporation that affects the cooling performance. This study provides a review of the research progress for solving this issue especially on the internal enhancement strategy and methods to improve the poor wettability on the wet channel surface. It is expected that IEC will make more contributions especially to further reduce the energy consumptions in buildings with the recent developments and probable future opportunities to cope with this problem.

Keywords: *Indirect evaporative cooling, Evaporative cooler, Internal enhancement*

SESSION 5B

94. Natural Convection over a vertical flat plate: A numerical and experimental investigation

Noman Ashraf[1] and Mazlan Abdul Wahid[2]

[1] Department of Building Engineering, College of Architecture and Planning, Imam Abdulrahman Bin Faisal University, 31451Al-Dammam, Kingdom of Saudi Arabia
[2] High-Speed Reacting Flow Research Laboratory (HiREF), Universiti Teknologi Malaysia, 81310, Johor Bahru, Johor, Malaysia.

Abstract

Numerical analysis was carried out for 2D steady-state laminar boundary layer during natural convection over a vertical flat plate subjected to constant heat flux and verified with experimental results. The experimental test section consists of a vertical flat plate having dimensions of 355 mm height and 177 mm width made from brass sheet. Temperatures of the heated plate wall and the adjacent heated air forming a boundary layer were recorded at various locations under the heat flux of 250, 500, 750 and 1000 W/ m^2 maintained on the plate surface. Numerical analysis for the constant wall temperature conditions had also been carried out for the purpose of validation. The governing boundary layer equations were transformed into non-dimensional forms of continuity, momentum, and energy equations. The resulting non-linear system of partial differential equations were linearised and solved numerically by finite difference method together with the Thomas Algorithms. The results of velocity contour, velocity flow lines, temperature contours, isotherms, velocity vector, and streamlines are presented for constant heat flux and constant wall temperature conditions.

Keywords: *Convection, Numerical Analysis, Experimentation, Heat Flux, Thomas Algorithms, Temperature contours, streamlines*

SESSION 6B

81. Long Short-Term Memory Neural Network Model for The Control of Temperature in a Multi-Circuit Air Conditioning System

Ibrahim Oleolo[1], Hayati Abdullah[1], Ismail Mustapha[2], Maziah Mohamad[1], Mohammad Nazri Mohd Jaafar[1], Akmal Baharain[1], Sapiah Sulaiman[3]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[2]School of Computing, Computer Science Department, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia
[3]Smart Digital Community Research Alliance Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Johor, Malaysia

Abstract

Temperature control is important in energy management of buildings. Air conditioning contributes a high percentage of the total energy consumption. In air conditioning systems, the compressor utilises up to 90% of the total energy being consumed. This can be reduced by varying the speed of the compressor with respect to the required indoor temperature, as such, reducing the overall energy usage of the air conditioning system. The combination of a well-tuned controller and variable frequency drive can be used to achieve this. It is important to develop a good model which can be used to design the controller. Although there are published research works in the development of models for the control of air conditioning systems, there seems to be a lack of study in the area of multi-circuit air conditioning system. In this research study, two models were developed using Long Short Term Memory Neural Network and Recurrent Neural Network, utilising compressor speed and indoor air temperature of a multi-circuit water cooled packaged unit as input and output respectively. Comparing the two models, the results showed that the Long Short-Term Memory Neural Network model performed better across evaluation metrics such as R-squared, Mean Squared Error and Mean Absolute Error, with value of 0.9485, 0.00694, and 0.02268 respectively.

Keywords: *Multi-circuit, Energy management, Temperature control*

SESSION 6B

79. Initial Design Parameters Optimization of Air-Cooled Solar Thermal Ammonia-Water Absorption Chiller

Desy Agung[1], M. Idrus Alhamid[1], Arnas Lubis[1], Nasruddin[1]

[1] Mechanical Engineering Department, Universitas Indonesia, Kampus UI Depok, 16424, Indonesia

Abstract

Air conditioning technology for cooling purposes is now directed more to the use of conventional cooling technology with the support of energy sources from nature. One of them is absorption chiller technology utilising solar energy to support the work of generator components. Absorption chiller system with ammonia-water solution has the advantage of being able to work on pressurised pipes and this solution itself basically has a distinctive aroma so that it also benefits both technicians and users in identifying leaks in the system. In this paper, a small capacity solar ammonia-water absorption chiller with air-cooled condenser and absorber is modelled based on the ambient temperature characteristic in the tropical climate of Indonesia. A multi-objective optimisation is carried out in this work to obtain the appropriate value of the parameters specified for the design, to get the optimum value of system performance and generator outlet solution temperature. The results of this study find that the most appropriate model for the absorption chiller with the limitation for tropical climate use reaching a coefficient of performance at 0.313 is by the condensing temperature in the condenser and absorber at 38°C, evaporating temperature at 6°C, which cause the generator solution outlet temperature to reach approximately 84.4 °C.

Keywords: *Absorption chiller, Ammonia-water, Solar Cooling, Air-cooled, Tropical climate*

SESSION 6B

76. Domestic Ejector Air-Conditioning System Performance Using Different Ejector and Refrigerant Type

Nurul Aida Nabila Abdul Halim[1], Zulkarnain Abdul Latiff [2] and Mohd Rozi Mohd Perang[2]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Johor, Malaysia
[2]Automotive Development Centre, Institute for Vehicle Systems and Engineering, Universiti Teknologi Malaysia, 81310 UTM Johor Bahru, Malaysia

Abstract

Improvement in refrigeration system performance could be achieved by reducing compressor workload through the use of an ejector. Hydrochlorofluorocarbon (HCFCs) refrigerant (R22) which contributes to global warming and ozone depletion needs to be replaced with eco-friendlier hydrocarbon (HC) refrigerant (R290). System performance analysis were carried out using two ejector designs, constant area mixing (CAM) and constant pressure mixing (CPM) utilising both R22 and R290 refrigerant. Several air temperatures were set ranging from 18°C to 29°C. CAM resulted in better performance than CPM with R22 achieving 20% higher cooling compared to R290 at 23°C setting. The low requirement of mass flow rate for R290 reduced the ejector performance in the secondary inlet and lowered cooling capacity with high compressor load. Thus, the entrainment ratio of the CAM with R290 is significantly lowered by 12.70%, 13.35%, 16.39%, 14.74%, and 7.93% compared to R22 for the air temperature settings. The optimum performance was obtained at refrigerant temperature of 12.67°C with air temperature setting at 23°C. The coefficient of performance (COP) demonstrated the same trend accordingly. Higher COP is required because it equates to better efficiency and reduced compressor energy. This is due to the saturated vapour entering the compressor which increased cooling capacity in the evaporator.

Keywords: *Ejector refrigeration system, Energy efficiency, HCFC and HC refrigerant*

ESSION 6B

20. An Advance Air-Conditioning System for Future Sustainability

Muhammad Wakil Shahzad[1*], Kim Choon Ng[2], Muhammad Burhan[2], M Kumja[2] and Qian Chen[2]

[1]Northumbria University, Newcastle Upon Tyne NE1 8ST, United Kingdom

[2]King Abdullah University of Science and Technology, Saudi Arabia

Abstract

The electricity consumed by the ACs will grow up to 3-fold by 2050 and it will be second largest electricity consumer after industrial sector. Over 95% of current cooling market is covered by conventional chillers because of advantages such as reliability and high performance. On the otherhand, they have many disadvantages such as high maintenance cost, chemical refrigerants utilization, noise and vibration issues. Montreal Protocol and Europe Council Directive (3093/94)forced to stop the production of chemical-based refrigerants and planned to eliminate completely by 2030. In order to meet the cooling demand for human comfort and specially growing data center requirement, out-of-box solutions are required urgently. We proposed a disruptive cooling technology called water droplet interceded indirect evaporative cooler (WD-IEC). It devoid the use of mechanical vapor compressors (MVC), chemical-based (chlorofluorocarbons) refrigerants,cooling towers, chilled and cooling water pipes, that eliminates more than 75% of conventional infrastructure of mechanical or thermally driven chillers. The proposed system utilizes only cleanwater for heat removal through evaporative potential of air. The initial results show that the overallspecific energy consumption can be reduced to 0.55±0.05 kWelectric per Refrigeration ton (Rton) from the conventional level of 0.85±0.05 kW/Rton. The other advantages include (i) zero globalwarming potential, (ii) 1/3 water consumption, (iii) maintenance cost is only 10% as compared toconventional chillers and (iv)simple and reliable operation. We also designed and fabricated 1 Rton commercial unit that is under testing and producing excellent results.

Keywords: *Air-conditioning, non-MVC chiller, IEC, sustainable cooling.*

SESSION 6B

86. Experimental Study on The Improvement of Thermal Comfort Inside a Car Cabin

Mohd Kameil, A.H.[1,2,3], Rajwinder, S.[1] , Mohd Zahedi, M.Z.[1], Zul Hilmi, C.D.[1,2,3], Izhari Izmi, M.[1,3] , Abd Rahim, A.B[1,3]

[1]School of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai, Johor
[2]Automotive Development Centre (ADC), Universiti Teknologi Malaysia, Skudai, Johor
[3]Vehicle Engineering Research Group, Universiti Teknologi Malaysia, Skudai, Johor

Abstract

The hot and humid climate throughout the year in Malaysia, which is located near the equator, affects the thermal comfort level inside parked cars. In a closed car cabin environment, the accumulated trapped heat causes the temperature to increase up to an average of 70 °C and this can pose threat to human and pets that are left inside. The objective of this paper is to identify an effective way to improve thermal comfort inside a sedan passenger car cabin that is parked under the hot sun. In this work, six cooling fans are used and placed at certain locations. To further investigate improvement in the thermal comfort, the experiments were also conducted with windows open, the use of cardboards and blankets to cover the windscreen and dashboard, respectively. The final result shows that the cooling fans with windows open, windscreens covered by cardboards and dashboard covered by thick blankets were able to reduce and stabilise the car cabin temperature, hence improved the thermal comfort.

Keywords: *Cabin Temperature, Thermal Comfort, Cooling Fan*

SESSION 6B

85. Experimental Investigation of Local Thermal Sensation of Vehicle Passengers During Cooldown

Muhammad Noor Afiq Witri Muhammad Yazid[1], Haslinda Mohamed Kamar[1], Mohd Zukhairi Abd Ghapar[2], Mohd Aizad Sazrul Sabrudin[2], Nurfarizal Rasid[2]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.
[2]Perusahaan Otomobil Nasional Sdn. Bhd., Persiaran Kuala Selangor, Seksyen 26, 40400 Shah Alam, Selangor, Malaysia

Abstract

Thermal behaviour in a car cabin and how it affects human thermal sensation is the most important element in the strategy for developing an effective car air conditioning system. However, the thermal environment inside a car cabin is complex, characterized by inhomogeneous temperature and air velocity fields due to complex interaction between air conditioning system of the car and the solar radiation. This paper aims to investigate the thermal sensations of vehicle passengers at local body parts during cooling period in a stationary car cabin. Air temperatures near the face, chest and feet as well as local thermal sensation votes that is based on nine-point thermal sensation scales have been collected under clear sky and consistent solar irradiation. Based on the data acquired from the experiments, probit regression analysis was applied for predicting localised air temperature variation ranges corresponding to local thermal sensation of vehicle passengers. The results of this study show that the maximum limit of air temperature recommended to achieve a thermal neutrality is approximately 34°C, 30°C and 29°C for face, chest and feet, respectively. It was also found that facial thermal sensation had a stronger relationship with local air temperature while feet thermal sensation was the least.

Keywords: *Vehicle thermal sensation, Vehicle cooldown, Field experiment, Probit regression*

SESSION 6B

55. Effects of Horizontal Wall-Mounted Air Supply on Particle Distribution in an Operating Room: A Simulation Approach

Huiyi Tan[1], Keng Yinn Wong[2], Wah Yen Tey[3], Bemgba Bevan Nyakuma[4], Syie Luing Wong[5], Mohd Hafiz Dzarfan[6], Xinyou Ho[1], Haslinda Mohamed Kamar[2], Chew Tin Lee[1] and Muhd Suhaimi Deris[7]

[1] School of Chemical & Energy Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia.
[2]School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
[3]Faculty of Engineering, Technology & Built Environment, UCSI University, 56000 Cheras, Kuala Lumpur, Malaysia
[4]Department of Chemistry, Faculty of Sciences, Benue State University, P.M.B 102119 Makurdi, Benue State, Nigeria
[5]Dpto. Matemática Aplicada, Ciencia e Ingeniería de Materiales y Tecnología Electrónica Universidad Rey Juan Carlos, C/ Tulipán s/n, Móstoles Madrid, Spain
[6]Advanced Membrane Technology Research Centre (AMTEC), Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
[7]Bumimaju MTE Engineering Sdn. Bhd., Taman Pinggiran Putra 43300 Seri Kembangan, Selangor, Malaysia

Abstract

Airflow pattern is critical in controlling the movements of bacteria-carrying particles (BCPs) in an operating room (OR). So far, studies on the horizontal wall-mounted air supply diffuser is limited albeit with controversial findings. Therefore, this article examines the efficiency of both vertical and horizontal air supply diffusers in reducing the BCPs at the surgical zone. The airflow in the OR was simulated using a renormalization group (RNG) k-ε turbulent flow model, while the movement of BCPs was simulated using a discrete phase model (DPM). A total of 5 cases were included in the present study, i.e., vertical air supply diffuser (baseline case), two-sided horizontal air supply diffuser on an x-x plane (case 1), two-sided horizontal air supply diffuser on a z-z plane (case 2), single-sided horizontal air supply diffuser on an x-x plane (case 3), and single-sided horizontal air supply diffuser on a z-z plane (case 4). The results showed that the airflow supplied by all the horizontal air supply diffusers (cases 1 – 4) did not reduce the BCPs concentration at the surgical zone. The particle settlement on the patient, however, increased by 8.5-times, 2-times, 13-times, and 15.5-times in cases 1, 2, 3 and 4, respectively.

Keywords: *Operating room, horizontal air supply, bacteria-carrying particles (BCPs), computational fluid dynamics (CFD), unidirectional airflow*

SESSION 11B

61. Effect of Magnetic Field on Different Type of Fluid to the Velocity Distribution in Circular Microchannel

Amnani Rosland[1], Natrah Kamaruzaman[1] and Mohsin Mohd Sies[1]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia
81310 UTM Johor Bahru, Johor, Malaysia

Abstract

Many technologies have been developed to efficiently deliver the required amount of medicines to suitable target areas and preserve desired drug levels in the drug delivery system. Controlling the flow of drug-delivery systems offer numerous advantages. The use of pumps and valves is however necessary to manage the fluid, and a mechanical pump or valve is challenging to incorporate in drug delivery process. Magnetohydrodynamics (MHD) is therefore an effective option with magnetic medicines that ensures that medicines are delivered to the damaged area quickly while keeping the treatment process intact across the entire organism in other areas. MHD makes the perfect, programmable method for fluid pumping and fluid control in microfluidic equipment without mechanical pumps or valves required. In this paper, the effect of magnetic field on velocity distribution is investigated with using different type of fluid. The working fluid used are Gallium and Galinstan. Numerical simulations are performed using Ansys Fluent and the Magnetic induction values are 0T,1T and 1.5T. It is found that velocity decreases with an increase of magnetic field and Hartmann Number.

Keywords: *Liquid Metal, MHD model, Magnetic Field*

SESSION 11B

37. Skin and Body Temperature Parameter Calibration of Max30100 Sensor Module Based on Arduino-Uno

Juan Karnadi[1], Ibnu Roihan[2] and Raldi Artono Koestoer[2]

[1]Faculty of Computer Science, Universitas Indonesia, Depok, Indonesia

[2]Heat Transfer Laboratory, Mechanical Engineering Department, Universitas Indonesia, Indonesia

Abstract

The infrared light emission on MAX30100 sensor module does enable several parameters' digital reading. Referring to datasheet, MAX30100 can only read heartrate and oxygen saturation parameter. The aim of this research isto process the temperature parameter's Arduino digital output and convert it into skin and body temperature -in which they are not on MAX30100- through calibration effort. The foregoing measure is to compare the characteristic of temperature change's aspect with DS18B20 temperature sensor in skin temperature reading. The calibration effort's process has four leading stages. First is averaging to enhance sensor's reading stability. The latter phase is filtering to enhance the sensor's reading accuracy. Next is to initiate fitting phase to better sensor's reading precision. The fitting itself occurs twice, namely nearing the skin temperature's reading result between MAX30100 and DS18B20, and then enhancing the body temperature result using body thermometer as the calibrator. Filtering was performed on ±0.15°C ratebeing obtained from the difference of MAX30100's current averaging result and average of its recent averaging results. The fitting result on each occurrence respectively has ammount of 6.02°C for skin temperature and 1.34°C for body temperature. According to these following results, the error rate is ±0.27% towards body thermometer.

Keywords:

SESSION 11B

89. Simplified Approach for Braking Pressure Applied to a Thin Concave Composite Wall Calculation

Sofiene Helaili[1,2]

[1] Tunisia Polytechnic School, Carthage University, LASMAP (LR03ES06), Rue El-Khawarizmi, BP 743, 2078, La Marsa, Tunisia
[2]Carthage University, ISTEUB, Rue de l'Artisanat Charguia 2, 2035, Tunis, Tunisia

Abstract

Road safety is of extreme importance, especially for mobile tanks transporting dangerous fluids. The designer of a portable tanker must calculate the braking forces that could overturn the tanker or destabilise it. The force to be managed during braking results from the pressure of the sloshing fluid applied to the inner walls and to the outer shell of the tank. The shape of the inner walls could improve the stability of the tank. To control the force amplitude, the internal walls can be made of composites. In this paper, determining the forces and pressure value are developed while composite walls are studied in another further work. The objective is to determine the value of the maximum pressure in the tank and to calculate the transverse and vertical forces. The approach is to take the worst moment of the braking as reference to develop the calculations. The effect of the tank length, the number of internal walls, the filling level, the braking deceleration, and the density of the transported fluid are studied. The results show that the braking force varies from 737 kg to 18,627 tons. If the length value is doubled, the vertical effort is multiplied by a factor of 12.1. If the fluid level is increased by 5% from 93% to 98%, the braking force increase just by 6.6%. The breaking force increase 10% per 100 kg/m^3 of density. For braking deceleration, extreme braking deceleration can increase the fluid pressure by 56.8%. Based on the mobile cistern volume and length, design parameters can be considered by the simplified approach explained I n the paper for mobile cisterns design.

Keywords: *Tank truck dynamics, fluid sloshing, fluid–tank–vehicle interaction, computational fluid dynamics (CFD), composites baffles*

SESSION 11B

92. Investigation on the Effect of Surface Texture Depth into the Performance of Journal Bearing using CFD

Zuraidah Rasep[1,2], Muhammad Noor Afiq Witri Muhammad Yazid[1] , Syahrullail Samion[1] and Nor Azwadi Che Sidik[3]

[1] School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 UTM Skudai, Johor, Malaysia.
[2]Process Engineering Technology Section, Malaysian, Institute of Chemical and Bioengineering Technology (MICET), Universiti Kuala Lumpur, 78000 Alor Gajah, Melaka, Malaysia.
[3]Department of Mechanical Precision Engineering, Malaysia-Japan International Institute of Technology, Universiti Teknologi Malaysia, 54100 Kuala Lumpur, Malaysia.

Abstract

Texture depth of textured journal bearing plays an important role to predict the overall performance of journal bearing efficiently. In this case study, an investigation was carried out on the effect of surface texture depth on the performance of journal bearing using CFD ANSYS FLUET. This case study also innovated the potential of RBD palm oil as a renewable source that can replace engine oil which is used in the application of journal bearing. The depth of texture was introduced at 0.049 mm, 0.050 mm, 0.051 mm, 0.052 mm and 0.053 mm. The study was conducted at different rotational speeds of the journal bearing starting from 200 rpm, 400 rpm, 600 rpm, 800 rpm, and 1000 rpm and at a constant eccentricity ratio of 0.7. As a result, depth texture of 0.050 mm at a rotational speed of 800 rpm and depth texture at 0.050 mm, 0.051 mm, and 0.052 mm at a rotational speed of 1000 rpm showed a high value of maximum fluid film pressure as compared to plain journal bearing. The potential of RBD palm oil was fully studied during the simulation study and proved that these types of bio-lubricants can replace conventional oil in the future specifically in journal bearing applications.

Keywords: *Palm oil, Journal bearing, CFD, ANSYS FLUENT, Tribology, Surface texture*

SESSION 11B

71. Inertial Flow Focusing Device Fabricated using 3D-printed Mold and Replica Molding Technique

Nur Anis Humaira Mohamad[1], Fazila Mohd Zawawi[1] and Ummikalsom Abidin[1]

[1] School of Mechanical Engineering, Universiti Teknologi Malaysia, 81300 Johor Bahru, Malaysia.

Abstract

Inertial flow focusing in a spiral microchannel has the potential to perform high throughput cell separation for clinical diagnostics and biological studies. In this research, the potential of 3D printing technique in fabrication of spiral microchannel mold and demonstration of inertial flow focusing are investigated. The polydimethylsiloxane (PDMS) microchannel replication from the 3D printed mold was successfully fabricated. The percentage difference obtained from the 3D printed mold and the original designed width was 1.29 %. In addition, 1.06 % percentage difference between the replicated PDMS microchannel and the original designed width was measured. Qualitative testing using 20-27 µm sized fluorescent microbeads with flowrate ranges from 400 to 1000 µL/min was completed and hydrodynamic flow focusing functionality was successfully demonstrated. Qualitative results show that microbeads are focused at the outer wall of the spiral microchannel of higher flowrate. This result agreed with the confinement ratio prediction as value lower than 0.07, the Dean drag effect forces the particles to move towards the outer wall of the spiral microchannel. In conclusion, spiral microchannel fabricated from the 3D printed mold demonstrated a good functionality as an inertial microfluidics flow focusing device.

Keywords: *Microfluidic, spiral microchannel, 3D printed mold, inertial flow focusing*

THANK YOU TO OUR SPONSOR

THANK YOU TO OUR STRATEGIC PARTNERS

HIGH SPEED REACTING FLOW RESEARCH LABORATORY
H REF

CFM
COMPUTATIONAL FLUID MECHANICS
SCHOOL OF MECHANICAL ENGINEERING
UNIVERSITI TEKNOLOGI MALAYSIA

ACER
AIR-CONDITIONING
ENGINEERING RESEARCH

Tuai padi antara masak
Esok jangan layu-layuan
Intai kami antara nampak
Esok jangan rindu-rinduan

Kalau ada sumur di ladang
Izinkan kami menumpang mandi
Kalau ada umur yang panjang
The 13th IMAT berjumpa lagi