1st UAE THERMO-FLUIDS DAY

25 November 2023

Khalifa University, Main Campus, Abu Dhabi, UAE

TABLE OF CONTENTS

1 About UAE Thermo-Fluids Day

UAE Thermo-Fluids Day aims to provide a scientific forum where researchers from educational and research institutions in the UAE come together to discuss the latest research advances in the broad areas of Thermo-Fluid and their applications. In addition, UAE Thermo-Fluid Day allows researchers from UAE institutions to meet, share ideas, and establish collaborative research to advance UAE academic institutions. Thermo-Fluids science is relevant to various fields of engineering and science, including petroleum, renewable energy, climate change, and geophysical flow, to name a few. The UAE Thermo-Fluid Day will encompass the four intersecting fields: Fluids Mechanics, Thermodynamics, Heat Transfer & Combustion. Specifically, the following areas are of interested

- Experimental and Numerical Methods in Fluid Mechanics and Heat/Mass Transfer
- Fundamentals in Fluid Flow and Heat/Mass and Momentum Transfer
- Flow Instability; Turbulent Flows
- Multiphase Fluid Flow and Heat Transfer
- Atomization, Combustion.
- Nano and Micro Fluids Applications
- Natural and Built Environments
- Ocean and Climate Science
- Plasma Physics and Engineering
- Refrigeration, Air Conditioning Systems, and Refrigerants
- Renewable & Sustainable Energy (Wind, Solar, Wave, etc…)

2 Sponsors

The sponsors of 1st Thermo-Fluids Day are gratefully acknowledged.

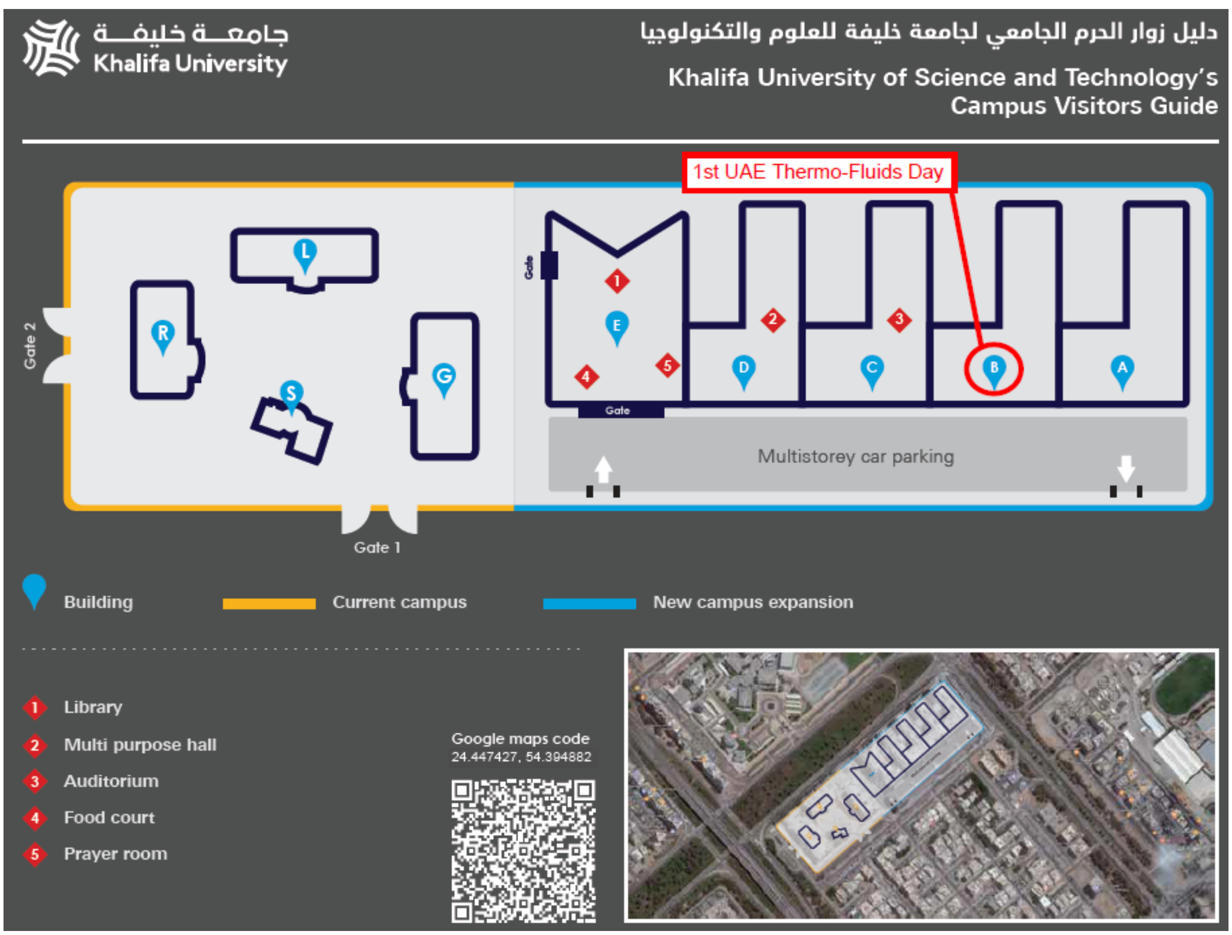
Khalifa University (https://www.ku.ac.ae/)

The Art House Cafe from Karammgroup (www.karammgroup.ae)

Business Communications LLC (https://www.bcluae.com/)

3 Venue Maps


Location map
Abu Dhabi Corniche
1st UAE Thermo-Fluids Day
Abu Dhabi Police HQ
Multi story car parking
A
B
C
D
E
G
L
R
S


جامعة خليفة
Khalifa University
دليل زوار الحرم الجامعي لجامعة خليفة للعلوم والتكنولوجيا
Khalifa University of Science and Technology's Campus Visitors Guide
1st UAE Thermo-Fluids Day
Gate 2
Gate 1
Gate
Multistorey car parking
Building
Current campus
New campus expansion
1 Library
2 Multi purpose hall
3 Auditorium
4 Food court
5 Prayer room
Google maps code
24.447427, 54.394882

جامعة خليفة
Khalifa University

دليل زوار الحرم الجامعي لجامعة خليفة للعلوم والتكنولوجيا

Khalifa University of Science and Technology's Campus Visitors Guide

اللباس الواجب ارتداؤه

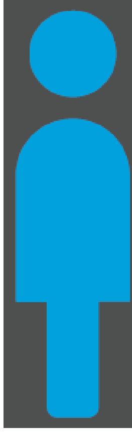
يتوجب على الزوار، بغض النظر عن دينهم أو جنسيتهم ارتداء ملابس لائقة ومناسبة للجو العام للحرم الجامعي. من الضروري أن يحترم كل زائر معايير المجتمع الإماراتي وأن يبتعد عن كل ما قد يسيء إلى الثقافة والعادات والتقاليد المحلية أو ما ينافي الذوق العام.

يجب ملاحظة النقاط التالية فيما يتعلق بالملابس الواجب ارتداؤها أثناء زيارة الحرم الجامعي، مع العلم أنه يجب ارتداء ملابس واقية خاصة للمختبرات عند الضرورة.

Dress Code

Visitors, irrespective of religion or nationality should dress in a modest and appropriate manner on campus. It is required that every visitor respects the norms of UAE society and he/she should not dress in a way that may offend cultural sensitivities and/or may not be within acceptable general taste.

The following points must be observed regarding visitors dress at the University campus. It is noted, however, that special protective clothing for laboratories shall be worn as necessary.



تعليمات ملابس الزوار من الذكور

- يجب على الزوار الذكور ارتداء القميص والبنطلونات الطويلة / الجينز. كما يجب تجنب ارتداء البنطلونات القصيرة (الشورت) أو القمصان بلا أكمام.
- يجب تجنب الملابس التي تعرض كتابات أو رسومات أو صور مسيئة أو مخلة أو تعتبر غير مقبولة.
- يجب تجنب إبراز الوشوم والحلق.

Male Visitors Dress Code

- Male visitors must wear a shirt and long trousers/jeans. Shorts and sleeveless shirts must be avoided.
- Clothes displaying offensive/objectionable writings/drawings/pictures must be avoided.
- Visible tattoos and piercings must be avoided.

تعليمات ملابس الزائرات من الإناث:

- يتوجب على الزائرات ارتداء ثياب محتشمة مع مراعاة ارتداء القمصان/البلوزات ذات أكمام طويلة والتنانير الطويلة أو الفضفاضة أو الفساتين الواسعة والطويلة.
- يجب تجنب ارتداء الشورت والتنانير القصيرة والقمصان بلا أكمام أو المنخفضة.
- يجب تجنب الملابس الضيقة أو الشفافة أو القصيرة أو الفاضحة أو التي تعرض الخصر أو الورك أو الساقين.
- يجب تجنب الملابس التي تعرض كتابات أو رسومات أو صور مسيئة أو مخلة أو تعتبر غير مقبولة.
- يجب تجنب إبراز الوشوم والحلق فيما عد حلق الأذن والحناء.

Female Visitors Dress Code

- Female visitors must dress conservatively wearing long-sleeved shirts/blouses with long and loose fitting skirts/slacks, or a long-sleeve loose fitting long dress.
- Shorts, short skirts and sleeveless/low-cut neckline shirts must be avoided.
- Clothing that is tight, transparent, or short and shows too much skin or exposes the waist, back, or legs must be avoided.
- Clothes displaying offensive/objectionable writings/drawings/pictures must be avoided.
- Visible tattoos and piercings must be avoided. This does not include generally acceptable items such as earrings and henna.

4 Local and National Organizing Committees

Local Organizing Committee
Dr Hamid Abderrahmane, Khalifa University, Abu Dhabi.
hamid.abderrahmane@ku.ac.ae

Dr Afshin Goharzadeh, Khalifa University, Abu Dhabi.
afshin.goharzadeh@ku.ac.ae

Dr Yap Yit Fatt, Khalifa University, Abu Dhabi.
yap.fatt@ku.ac.ae

National Organizing Committee
Dr Hamid Abderrahmane, Khalifa University, Abu Dhabi.
hamid.abderrahmane@ku.ac.ae

Dr Amjad Tuffaha, American University of Sharjah, Sharjah.
atufaha@aus.edu

Dr Bobby Mathew, United Arab Emirates University, Al Ain.
bmathew@uaeu.ac.ae

Dr Mehdi Nazarinia, Heriot-Watt University, Dubai.
m.nazarinia@hw.ac.uk

Dr Mohammad Qasaimeh, New York University, Abu Dhabi.
maq4@nyu.edu

Dr Sharul Sham Bin Dol, Abu Dhabi University, Abu Dhabi.
sharulshambin.dol@adu.ac.ae

Dr Youssef Al Haik, University of Sharjah, Sharjah.
yhaik@sharjah.ac.ae

5 Program Schedule

Time	
8:00 - 8:30	Registration
8:30 - 8:45	Opening Ceremony (Room B00055)
8:45 - 9:15	**Invited Lecture 1: Prof Dimtrios Kyritsis (Room B00055)**
9:15 - 9:30	Coffee Break
9:30 - 12:30	**Parallel Sessions (1A, 1B and 1C)**
12:30 - 13:30	Photo Session & Lunch
13:30 - 14:00	**Invited Lecture 2: Prof Youssef Belhamadia (Room B00055)**
14:00 - 14:30	Coffee Break
14:30 - 16:30	**Parallel Sessions (2A, 2B and 2C)**
16:30 - 16:45	Closing Ceremony

9:30 - 12:30, Parallel Session 1A (Room B00055)

Time	Presentation
9:30-10:00	TFD1: Numerical Solution to a One-Dimensional Nonlinear Problem of Heat Wave Propagation in a Rigid Thermal Conducting Slab *by N H Sweilam, A F Ghaleb, M S Abou-Dina & M M A Hasan**
10:00-10:30	TFD 2: Hydrodynamics of wall-bounded turbulent flows through screens: a numerical study *by Walid Abou-Hweij* & Fouad Azizi*
10:30-11:00	TFD 4: Numerical Simulation Study on Ice Accretion over NACA0012 airfoil Under Varying Airflow Conditions *by Aghyad B. Al Tahhan* & Sharul Sham Dol*
11:00-11:30	TFD 6: Maxwell-Stefan Diffusion Approach for Modeling Water/Ethanol Separation using Sweeping Gas Membrane Distillation *by M .K. Fawzy* & S. Bandini*
11:30-12:00	TFD 7: Comparison of Aerodynamic Models for Aeroelastic Analysis *by Sheharyar Malik* & Ali Khalid*
12:00-12:30	TFD 23: Thermo-acoustically Driven Refrigerator (TADR) *by Isam Janajreh* & Omar Al-Mufti*

9:30 - 12:30, Parallel Session 1B (Room B00107)

Time	Presentation
9:30-10:00	TFD 8: Ionic Current Magnetic Fields in Charged Nanopores and Nanoslits by Abraham Mansouri*
10:00-10:30	TFD 11: Numerical Investigation of Tesla Valves for Insulin Drug Delivery *by Aayush Karna*, Seoyoung Kim & Sunil Kumar*
10:30-11:00	TFD 12: Rational Design of Low-GWP Refrigerants as Drop-in Replacements through a 4E Analysis using Robust Thermodynamic Models *by Carlos G. Albà*, Ismail I. I. Alkhatib, Fèlix Llovell & Lourdes F. Vega**
11:00-11:30	TFD 14: Efficiency Assessment of Solar Thermal Radiation Utilizing Advanced Nanofluid on Stretching Sheet *by Saleem Nasir* & Abdullalh Berrouk*
11:30-12:00	TFD 15: A Machine Learning Approach to Solving Inverse Problems in Radiative Transfer in Scattering-Absorbing Medium *by Wenjun Jiang* & Sunil Kumar*
12:00-12:30	TFD 16: Numerical Study on Influence of Electromagnetic Force on Suppression and Growth of Convection Heat and Mass Transfer in an Asymmetric Enclosure *by Ranjit J. Singh*, Sanjairaj Vijayavenkataraman & Sunil Kumar*

9:30 - 12:30, Parallel Session 1C (Room B01048)

Time	Presentation
9:30-10:00	TFD 20: Experimental Characterization of the Wake Fluxes Behind a Two-Dimensional Bluff Body at Varying Reynolds Number *by V. Parezanović**
10:00-10:30	TFD 9: New Mini Portable AC using Peltier Modules *by Kamel Adref, Alyazya Abdulla Murad Karam Alblooshi*, Khalid Adel Khaleefa Naseeb Alshamsi & Mohamed Yousef Jassim Mohamed Al Ali*
10:30-11:00	TFD 10: Characterization and Testing of the 3D-Printed Microfluidic Mixing Probe *by Dima Samer Ali*, Ayoub Glia, Muhammedin Deliorman & Mohammad A. Qasaimeh*
11:00-11:30	TFD 13: Design and Development of Indirect Evaporative Cooler with Enhanced Heat Transfer and Performance *by Hassan Ali*, Sabir Rasheed, Muzaffar Ali & Nadeem Ahmed Sheikh*
11:30-12:00	TFD 18: A Novel Process to Fabricate Paper Based Microfluidic Devices Using Laminating Plastic Sheets *by Shafeek A. Samad, Nityanand Kumawat, Soja Soman & Sunil Kumar*
12:00-12:30	TFD 25: Microfluidic Origami Platform for Edible Oil Quality *by Pavithra Sukumar, Muhammedin Deliorman, Fernando Castaño, Leen Sameer Ali, Dana H. Abujalban, Muhammad Alu Datt & Mohammad A. Qasaimeh**

14:30 - 16:30, Parallel Session 2A (Room B00055)

Time	Presentation
14:30-15:00	TFD 21: Flow and Heat Transfer from Cylinder under FIV: Effect of Spacing and Stagger Angle *by Md. Islam* & Ussama Ali*
15:00-15:30	TFD 26: Spatial Prediction of Dust Distribution in Solar Fields using Two Phase CFD Approach *by Hamza Fiaz*, Ali Al Masabai, Maryam R. AlShehhi & TieJun Zhang*
15:30-16:00	TFD 24: Herringbone Microfluidic Probe for Multiplexed Affinity-Capture of Prostate Circulating Tumor Cells *by Ayoub Glia*, Muhammedin Deliorman, Pavithra Sukumar, Farhad K. Janahi, Bisan Samara, Ayoola T. Brimmo & Mohammad A. Qasaimeh*
16:00-16:30	TFD 27: Experimental Study of Ferrofluid Slug Flow under a Uniform Magnetic Field *by Haider Ali Khan, Afshin Goharzadeh* & Firas Jarrar*

14:30 - 16:30, Parallel Session 2B (Room B00107)

Time	Presentation
14:30-15:00	TFD 19: Capillary Driven Passive Fog Harvesting by 3D Printed Micro-Cones with Structured Surface *by Abubaker S. Omer*, Aikifa Raza, Maryam AlShehhi, Faisal AlMarzooqi & TieJun Zhang*
15:00-15:30	TFD 3: Analysis of the 3D Turbulent Wake Behind a Flat-Back Ahmed Body with Base Bleed using Stacked Stereoscopic Particle Image Velocimetry *by T.I. Khan*, V. Parezanović & I. Afgan*
15:30-16:00	TFD 22: Experimental Testing of Aeroelasticity *by Sheharyar Malik* & Shaaban Ali*
16:00-16:30	TFD 28: Modeling of Simultaneous Particle Deposition and Deposit Erosion *by Li Hongying, Md. Didarul Islam, Afshin Goharzadeh & Yap Yit Fatt**

14:30 - 16:30, Parallel Session 2C (Room B01048)

Time	Presentation
14:30-15:00	TFD 29: Multi-Modal Microfluidic Bend Sensor for Soft Sensing Applications *by Wael Othman & Mohammad A. Qasaimeh**
15:00-15:30	TFD 30: Modeling Molten Corium-Concrete (MCCI) Interaction with the CFD Approach: Impact of various Boundary Conditions on MCCI *by Ilyas Khurshid*, Yacine Addad & Imran Afgan*
15:30-16:00	TFD 31: Flow Mixing in Cold-Leg of a Pressurized Water Reactor: CFD Modeling Approach *by Mubashir Hassan*& Yacine Addad*

6 Invited Lectures

Lecture 1: Vistas of Synthetic Fuels - Case Studies on Light Alcohols and Ammonia

Dimitrios C. Kyritsis*
Department of Mechanical and Nuclear Engineering, Khalifa University, Abu Dhabi, UAE
dimitrios.kyritsis@ku.ac.ae

Abstract
Synthetic fuels are increasingly being considered as a major energy storage component of a completely sustainable power conversion scenario. Equally exciting is the fact that the utilization of such fuels can involve physical processes and chemical phenomena that are entirely unknown for conventional, hydrocarbon-based fuel blends. We will highlight this through two specific examples:

- Electrostatic manipulation of alcohol sprays and flames. It will be shown that through the application of simple inserts into practical injectors, it is possible to inject electrostatic charge that can affect both fuel dispersion and droplet size. This was proven with the use of Fraunhofer diffraction measurements of droplet size, and particle image velocimetry measurements of droplet velocity. Also, results will be presented that substantiate the effect of electrostatic charge on single-droplet and counterflow-flame combustion. The results will be rationalized through the notions of ionic wind and ambipolar diffusion and the introduction of the corresponding terms in the pertinent transport equations.
- Ammonia ignition and flame structure. Chemical and physical fundamentals will be presented that will facilitate the use of ammonia as a carbon-less fuel that can be used in existing infrastructure (gas turbines). Emphasis will be placed on ignition delay control, NOx emissions, and novel flame structure that has not been observed for hydrocarbons.

It will thus be suggested that the utilization of synthetic fuels will not only allow the incorporation of existing infrastructure in a sustainable energy conversion paradigm, but will necessitate the design of entirely novel energy conversion technologies and modules.

Biography

Dimitrios C. Kyritsis is Professor and Chair of the Department of Mechanical Engineering and Deputy Director of the Research and Innovation Center for CO2 and Hydrogen in Khalifa University. He received his Diploma in Engineering from the National Technical University of Athens in Greece in 1992 and his MA and PhD from Princeton University in 1995 and 1998, respectively. Before his current appointment, he was a post-doctoral associate and a lecturer at the Department of Mechanical Engineering at Yale University (2000-2002) and a faculty member in the University of Illinois at Urbana-Champaign (2002-2014). His research focuses in the areas of laser-based reactive flow diagnostics and chemical dynamics of reduced-carbon-footprint combustion. In the period 2005-2010 served as the co-PI in the US-Department-of-Energy-funded Graduate Automotive Technology Education Center of Excellence on Automotive Biofuel Combustion Engines in the University of Illinois. He is a Fellow of the ASME, an Associate Fellow of the AIAA, an Associate Editor of the Journal of Energy Engineering, a recipient of the NSF CAREER award, the Accenture Award for excellence in student advising, as well as the Ralph R. Teetor Award of the Society of Automotive Engineers, and has served in the editorial board of Combustion & Flame and the Proceedings of the Combustion Institute.

Lecture 2: Thermo-Fluid Dynamics of Solid-Liquid Phase Change Problems: Parabolic and Hyperbolic Approaches

Youssef Belhamadia*
American University of Sharjah, Department of Mathematics and Statistics, United Arab Emirates
ybelhamadia@aus.edu

Abstract
The study of thermo-fluid dynamics related to solid-liquid phase change problems is significant in numerous engineering and industrial applications, including crystal growth, continuous casting, and energy storage, among others. In recent years, a variety of numerical and theoretical techniques have been developed to provide the necessary tools for understanding the physical processes. Nevertheless, the numerical modeling of these systems is still a challenging and ongoing research area. The objective of this talk is to present recent developments in computational thermofluid dynamics of solid-liquid phase change systems. Firstly, we will derive the mathematical models for the parabolic phase change system with and without convection, which predict an infinite thermal wave speed of propagation. Secondly, we will present a hyperbolic approach to predict the finite speed of heat propagation in phase change systems. Suitable numerical methods for solving the derived models using both approaches will be illustrated. Numerical simulations on water solidification, gallium melting, and continuous casting will be explored to assess the performance of the proposed techniques.

Biography

Youssef Belhamadia is an Associate Professor of Mathematics at the American University of Sharjah (AUS) in the United Arab Emirates. He obtained his Ph.D. in Mathematics from Laval University, Qu´ebec, Canada. Prior to joining AUS, he was a contract assistant professor in Campus Saint-Jean at the University of Alberta, Canada, and an adjunct assistant professor in the Department of Biomedical Engineering and Department of Mathematical and Statistical Sciences at the University of Alberta. His research interests include the numerical modeling of phase-change heat and mass transfer and the development of numerical methods for cardiac electrocardiology models. His work has been awarded funds from the Natural Sciences and Engineering Research Council of Canada, Heart and Stroke Foundation of Canada, the Royal Society in UK, and the American University of Sharjah.

7 List of Abstracts

- TFD1: Numerical Solution to a One-Dimensional Nonlinear Problem of Heat Wave Propagation in a Rigid Thermal Conducting Slab *by N H Sweilam, A F Ghaleb, M S Abou-Dina & M M A Hasan**
- TFD 2: Hydrodynamics of wall-bounded turbulent flows through screens: a numerical study *by Walid Abou-Hweij* & Fouad Azizi*
- TFD 3: Analysis of the 3D Turbulent Wake Behind a Flat-Back Ahmed Body with Base Bleed using Stacked Stereoscopic Particle Image Velocimetry *by T.I. Khan*, V. Parezanović & I. Afgan*
- TFD 4: Numerical Simulation Study on Ice Accretion over NACA0012 airfoil Under Varying Airflow Conditions *by Aghyad B. Al Tahhan* & Sharul Sham Dol*
- TFD 6: Maxwell-Stefan Diffusion Approach for Modeling Water/Ethanol Separation using Sweeping Gas Membrane Distillation *by M .K. Fawzy* & S. Bandini*
- TFD 7: Comparison of Aerodynamic Models for Aeroelastic Analysis *by Sheharyar Malik* & Ali Khalid*
- TFD 8: Ionic Current Magnetic Fields in Charged Nanopores and Nanoslits by Abraham Mansouri*
- TFD 9: New Mini Portable AC using Peltier Modules *by Kamel Adref, Alyazya Abdulla Murad Karam Alblooshi*, Khalid Adel Khaleefa Naseeb Alshamsi & Mohamed Yousef Jassim Mohamed Al Ali*
- TFD 10: Characterization and Testing of the 3D-Printed Microfluidic Mixing Probe *by Dima Samer Ali*, Ayoub Glia, Muhammedin Deliorman & Mohammad A. Qasaimeh*
- TFD 11: Numerical Investigation of Tesla Valves for Insulin Drug Delivery *by Aayush Karna*, Seoyoung Kim & Sunil Kumar*
- TFD 12: Rational Design of Low-GWP Refrigerants as Drop-in Replacements through a 4E Analysis using Robust Thermodynamic Models *by Carlos G. Albà*, Ismail I. I. Alkhatib, Fèlix Llovell & Lourdes F. Vega**
- TFD 13: Design and Development of Indirect Evaporative Cooler with Enhanced Heat Transfer and Performance *by Hassan Ali*, Sabir Rasheed, Muzaffar Ali & Nadeem Ahmed Sheikh*
- TFD 14: Efficiency Assessment of Solar Thermal Radiation Utilizing Advanced Nanofluid on Stretching Sheet *by Saleem Nasir* & Abdullalh Berrouk*
- TFD 15: A Machine Learning Approach to Solving Inverse Problems in Radiative Transfer in Scattering-Absorbing Medium *by Wenjun Jiang* & Sunil Kumar*
- TFD 16: Numerical Study on Influence of Electromagnetic Force on Suppression and Growth of Convection Heat and Mass Transfer in an Asymmetric Enclosure *by Ranjit J. Singh*, Sanjairaj Vijayavenkataraman & Sunil Kumar*
- TFD 18: A Novel Process to Fabricate Paper Based Microfluidic Devices Using Laminating Plastic Sheets *by Shafeek A. Samad, Nityanand Kumawat, Soja Soman & Sunil Kumar*
- TFD 19: Capillary Driven Passive Fog Harvesting by 3D Printed Micro-Cones with Structured Surface *by Abubaker S. Omer*, Aikifa Raza, Maryam AlShehhi, Faisal AlMarzooqi & TieJun Zhang*
- TFD 20: Experimental Characterization of the Wake Fluxes Behind a Two-Dimensional Bluff Body at Varying Reynolds Number *by V. Parezanović**
- TFD 21: Flow and Heat Transfer from Cylinder under FIV: Effect of Spacing and Stagger Angle *by Md. Islam* & Ussama Ali*
- TFD 22: Experimental Testing of Aeroelasticity *by Sheharyar Malik* & Shaaban Ali*
- TFD 23: Thermo-acoustically Driven Refrigerator (TADR) *by Isam Janajreh* & Omar Al-Mufti*

- TFD 24: Herringbone Microfluidic Probe for Multiplexed Affinity-Capture of Prostate Circulating Tumor Cells *by Ayoub Glia*, Muhammedin Deliorman, Pavithra Sukumar, Farhad K. Janahi, Bisan Samara, Ayoola T. Brimmo & Mohammad A. Qasaimeh*
- TFD 25: Microfluidic Origami Platform for Edible Oil Quality *by Pavithra Sukumar, Muhammedin Deliorman, Fernando Castaño, Leen Sameer Ali, Dana H. Abujalban, Muhammad Alu Datt & Mohammad A. Qasaimeh**
- TFD 26: Spatial Prediction of Dust Distribution in Solar Fields using Two Phase CFD Approach *by Hamza Fiaz*, Ali Al Masabai, Maryam R. AlShehhi & TieJun Zhang*
- TFD 27: Experimental Study of Ferrofluid Slug Flow under a Uniform Magnetic Field *by Haider Ali Khan, Afshin Goharzadeh* & Firas Jarrar*
- TFD 28: Modeling of Simultaneous Particle Deposition and Deposit Erosion *by Li Hongying, Md. Didarul Islam, Afshin Goharzadeh & Yap Yit Fatt**
- TFD 29: Multi-Modal Microfluidic Bend Sensor for Soft Sensing Applications *by Wael Othman & Mohammad A. Qasaimeh**
- TFD 30: Modeling Molten Corium-Concrete (MCCI) Interaction with the CFD Approach: Impact of various Boundary Conditions on MCCI *by Ilyas Khurshid*, Yacine Addad & Imran Afgan*
- TFD 31: Flow Mixing in Cold-Leg of a Pressurized Water Reactor: CFD Modeling Approach *by Mubashir Hassan*& Yacine Addad*

TFD1: Numerical Solution to a One-Dimensional Nonlinear Problem of Heat Wave Propagation in a Rigid Thermal Conducting Slab

N H Sweilam[1], A F Ghaleb[1], M S Abou-Dina[1] & M M A Hasan[2]*
[1]Department of Mathematics, Faculty of Science, Cairo University, Giza, Egypt
[2]School of Mathematics and Data Science, Emirates Aviation University, Dubai, UAE.
*eaus419@eau.ac.ae

Keywords
Nonlinear heat wave equation; Rigid thermal conductor; Slab; Numerical solution; Weighted average nonstandard finite-difference method;

Abstract
This work aims at presenting a new numerical solution to a nonlinear, one-dimensional problem of heat wave propagation in a thick slab of a rigid thermal conductor. The model predicts dependence of second sound velocity on temperature and heat flux. For this, an unconditionally stable numerical scheme is constructed using a kind of weighted average nonstandard finite difference discretization. Stability analysis of this scheme is studied by von Neumann technique, and its accuracy is proved. Numerical simulations are given for all quantities of physical interest to confirm the reliability of the proposed method. The model and the numerical results shed light on characteristics of heat wave propagation within the theory of extended thermodynamics.

References
[1] X J Yang, J A Tenreiro Machado and D Baleanu Fractals 25(4) 7539 (2019) 1740006 (7 pages)
[2] X J Yang, F Gao and H-W Jing Therm. Sci. 23 1555 (2019)
[3] X J Yang Therm. Sci. 23 4117 (2019)
[4] X J Yang and J A Tenreiro Machado Math. Methods Appl. Sci. 42 7539 (2019)

TFD 2: Hydrodynamics of Wall-Bounded Turbulent Flows Through Screens: A Numerical Study

Walid Abou-Hweij[1]* & Fouad Azizi[2]
[1]School of Engineering, Emirates Aviation University, United Arab Emirates
[2]M. Semaan Faculty of Engineering and Architecture, American University of Beirut, Lebanon
*walid.abou@eau.ac.ae

Keywords
Dispersive mixing; extensional efficiency; hydrodynamics; macro-mixing; residence time distribution; static mixer

Abstract
The hydrodynamic performance of turbulent flows in circular pipes equipped with screentype static mixers is numerically assessed in this study. A three-dimensional computational fluid dynamics model is used to study the effect of changing the operating conditions and reactor configuration on the flow field. The accuracy of the numerical results is validated by comparing pressure drop predictions to empirical correlations where a maximum relative error of 13.3% is recorded. The macro-mixing performance of screen type static mixers is also assessed using residence time distributions. The study shows that the flow through screens is three dimensional by nature with secondary flows being prominent near the pipe walls. Moreover, the presence of the screen has a major impact on the turbulent velocity profile both up- and down-stream. The flow field and velocity gradients are interpreted using strain rate and vorticity. These parameters also show that the flow through screens is highly dispersive where 39.3% of the reactor volume has an extensional efficiency value greater than 0.6. This explains their good performance in processing multiphase flows and gives an insight on how to design systems that maximize this dispersive effect in their volume. The residence time distribution study shows that the presence of screens renders the flow closer to plug flow with the effect being more pronounced using finer mesh screens operating at high flow velocities [1].

Reference

[1] Abou-Hweij, W., and Azizi, F. Hydrodynamics of wall-bounded turbulent flows through screens: a numerical study. *Chemical Engineering Communications*, *210*(8), 1305-1327, (2023). DOI: 10.1080/00986445.2022.2084391

TFD 3: Analysis of the 3D Turbulent Wake Behind a Flat-Back Ahmed Body with Base Bleed using Stacked Stereoscopic Particle Image Velocimetry

T.I. Khan*, V. Parezanović & I. Afgan
Khalifa University of Science and Technology, Abu Dhabi, UAE
*tauha.irfan@ku.ac.ae

Keywords
Bluff body, turbulent wake, steady jet, flow control, PIV.

Abstract
The three-dimensional nature of the wake behind a flat-back Ahmed body is investigated using a stacked stereoscopic particle image velocimetry (SSPIV) technique. The wake is disturbed using steady blowing from the center of the base and the effects of actuation are quantified for several flow rates. The natural wake exhibits bi-stable dynamics which is completely subdued for the optimal blowing case. The result is a symmetrized wake with a base drag reduction of around 9%. Different scales of base blowing reveal a similar positive effect, where the optimal blowing coefficient is found to scale with the bleed-to-base area ratio as $(S_j/S)^{1/2}$ [1]. The reconstructed 3D flow field is used to evaluate the effect of base blowing on the entrainment capacity of the global shear layers. An estimation of fluxes feeding and emptying the recirculation region is also carried out in light of the budget proposed by Gerrard [2]. The drag reduction mechanism is identified through two regimes; the *mass regime* and the *favorable momentum regime*. The former is described through an affine relationship between drag reduction and base blowing, whereas the latter is observed to introduce non-linearities by affecting the flux feeding the recirculation region. The transition between the two regimes is identified through the increased drag reduction potential as well as pronounced effect on wake asymmetry suppression. Proper Orthogonal Decomposition of the 3D wake field reveal the coherent wake structures along with prominent mode shapes corresponding to the three velocity components. The wake asymmetric state presents an interaction of a major hairpin vortex with a secondary horseshoe structure, whereas the optimal blowing case comprises of a pair of horseshoe structures parallel to the base.

References
[1] Khan, T. I., Parezanović, V., Pastur, L., & Cadot, O. (2022). Suppression of the wake steady asymmetry of an Ahmed body by central base bleed. Physical Review Fluids, 7(8), 083902.
[2] Gerrard, J. H. (1966). The mechanics of the formation region of vortices behind bluff bodies. Journal of fluid mechanics, 25(2), 401-413.

TFD 4: Numerical Simulation Study on Ice Accretion over NACA0012 airfoil Under Varying Airflow Conditions

Aghyad B. Al Tahhan* & Sharul Sham Dol
Department of Mechanical and Industrial Engineering, Abu Dhabi University, Abu Dhabi
*1070819@students.adu.ac.ae

Keywords
Aerodynamics; Ice accretion; NACA0012; FENSAP-ICE; Inflight icing

Abstract
The following study examines the heat and mass transfer characteristics on a two-dimensional aircraft wing profile of a NACA0012 airfoil during in-flight icing. The flow behavior is modelled using the k-ω SST turbulence model. Moreover, the model is physically and thermally modeled using the compressible Reynolds-averaged Navier-Stokes (RANS) equations, and the results obtained yielded an approximation of the total mass collected by the airfoil at varying flow speeds of 80, 102, 150, and 200 m/s. Furthermore, the model suggests consistent ice accretion results compared to experimental results obtained using Lagrangian or Eulerian methods from literature at similar atmospheric conditions. Moreover, the influence of icing on the aerodynamic characteristics of the airfoil was examined by comparing the coefficient of lift and drag (C_L) and (C_D), respectively, at various Angles of Attack(AoA). Finally, the Ice-Protection System (IPS) heating load required to evaporate the ice accreted on the airfoil fully is quantified using the conventional thermal balance during specified free stream conditions.

References

[1] Z. A. Janjua, B. Turnbull, S. Hibberd and K.-S. Choi, "Mixed ice accretion on aircraft wings," *Physics of Fluids,* vol. 30, no. 2, 2018.
[2] X. Bua, G. Lina, X. Shena, Z. Hub and D. Wena, "Numerical simulation of aircraft thermal anti-icing system based on a tight-coupling method," *International Journal of Heat and Mass Transfer,* vol. 148, 2020.
[3] H. Dai, C. Zhu, H. Zhao and S. Liu, "A New Ice Accretion Model for Aircraft Icing Based on Phase-Field," Applied Sciences, vol. 11, no. 5693, pp. 1-14, 2021.
[4] K. Fukudome, Y. Tomita, S. Uranai, H. Mamori and M. Yamamoto, "Evaluation of Anti-Icing Performance for an NACA0012 Airfoil with an Asymmetric Heating Surface," *Aerospace,* vol. 8, no. 10, p. 294, 2021.

TFD 6: Maxwell-Stefan Diffusion Approach for Modeling Water/Ethanol Separation using Sweeping Gas Membrane Distillation

M .K. Fawzy[1]* & S. Bandini[2]
[1]Department of Petroleum Engineering Technology, Abu Dhabi Polytechnic, Abu Dhabi, UAE
[2]Department of Civil, Chemical, Environmental and Materials Engineering, University of Bologna, Bologna, Italy
*mohamed.fawzy@actvet.gov.ae

Keywords
Hydrophobic Ceramic Membranes; Maxwell-Stefan Diffusion; Mass and Heat Transfer; Sweeping Gas Membrane Distillation

Abstract
Ethanol is deemed to be one of the commonly studied volatile organic compounds (VOCs) that can be separated from aqueous solutions by membrane distillation (MD) due to the possible applications in bioethanol dewatering, ethanol separation from fermentation broth and in wastewater treatment [1].
The employment of sweeping gas membrane distillation (SGMD) in (water/ethanol) separation using hydrophobic ceramic membranes was investigated in this work using modeling studies.

A mathematical model was implemented for modeling water/ethanol separation using SGMD. The morphological properties and geometrical parameters for the modeled membranes were based on a module of an unbaffled bundle of tubular membranes inserted in a shell. The membranes are macroporous multilayer titania membranes that were manufactured by the Fraunhofer Institute for Ceramic Technologies and Systems-IKTS, Germany. The membranes are coated in the top layer with a combination of fluoroalkyl silanes and a polymer (confidential materials) to provide hydrophobicity (tested by liquid breakthrough measurements with pure water).

According to the studied SGMD process, the feed side is a liquid binary mixture of water and ethanol while the permeate side is a gaseous ternary mixture of water, ethanol, and nitrogen (the sweeping gas). Transmembrane heat and mass transfer occur simultaneously. The fluid mechanics on each side were examined to estimate the mass transfer and convective heat transfer coefficients. Besides, the vapor-liquid equilibrium of water/ethanol was considered in the model.

The Maxwell-Stefan approach was followed in expressing the molecular diffusion of the water and ethanol molecules across the membrane putting into account the effect of the mutual interaction between the permeating species on the final resulting diffusion rate of each species. A MATLAB code was developed for the required computational execution.

Modeling results showed that the liquid temperature and gas pressure could have significant effects on the fluxes and the ratio between the mass fluxes of the permeating species. Raising the liquid temperature and/or decreasing the pressure enhanced the total flux and augmented the selectivity of ethanol.

By modeling the coupled effect of the inlet liquid temperature and the total length of the SGMD vessel for the counter-current flow configuration, the total flux was found to decrease by increasing the vessel's total length. However, the liquid inlet temperature and the total vessel length could be optimized for maximizing the ethanol selectivity as demonstrated in Fig (1). This could be interesting for process development for the related applications.

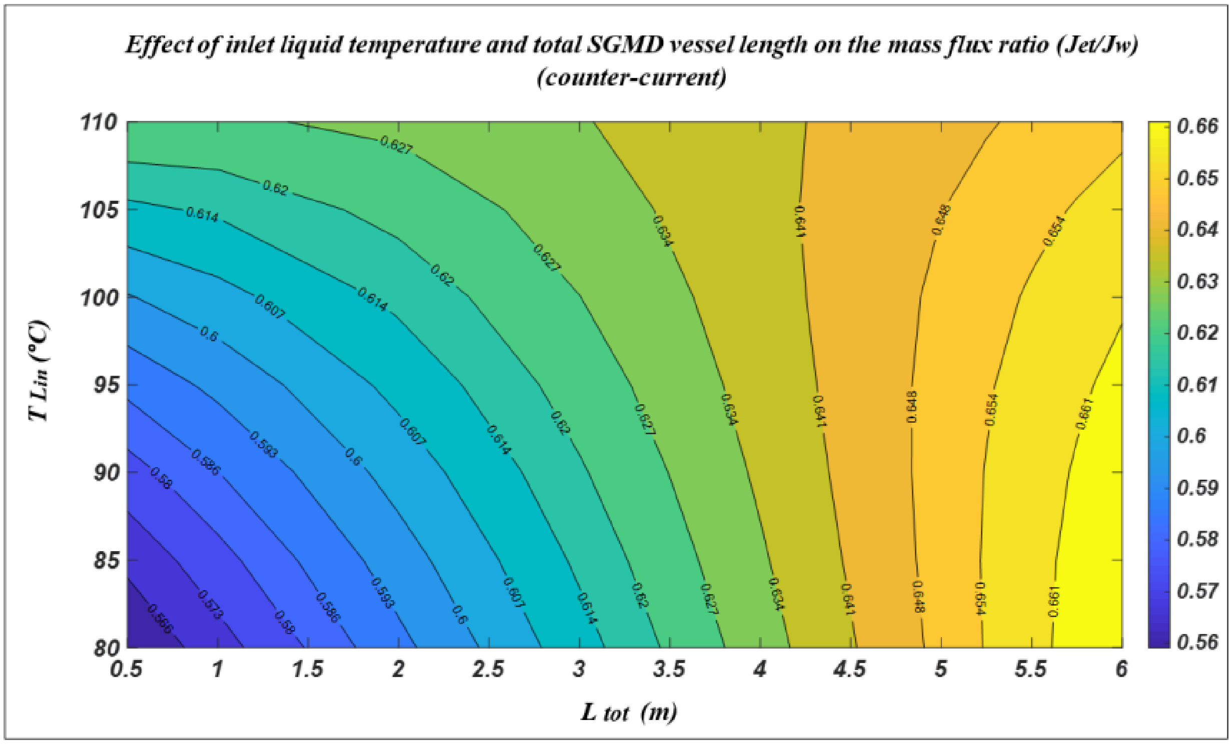


Figure 1: Results of SWGM modeling of ethanol (aq.) in counter-current flow configuration showing the coupled effect of the inlet liquid temperature and the total axial separation length (SGMD vessel length) on the ratio (ethanol mass flux/water mass flux)

Reference

[1] M.M.A. Shirazi, A. Kargari, M. Tabatabaei, Sweeping Gas Membrane Distillation (SGMD) as an Alternative for Integration of Bioethanol Processing: Study on a Commercial Membrane and Operating Parameters, Chem. Eng. Commun. 202 (2015) 457–466. doi:10.1080/00986445.2013.848805.

TFD 7: Comparison of Aerodynamic Models for Aeroelastic Analysis

Sheharyar Malik* & Ali Khalid
Abu Dhabi Polytechnic, Al Ain, UAE
*sheharyar.malik@actvet.gov.ae

Keywords
Aeroelastic, Aerodynamics models, comparison

Abstract
The parts of the aircraft have resonating ability while interacting with the airflow. Therefore, aeroelastic analysis is used to predict the response of an aircraft flexible structure while interacting with the airflow and it includes analyses such as divergence, flutter and buffeting etc. Aeroelastic analysis couples the structural behavior with the unsteady aerodynamics. Structural analysis can be performed through well-known Finite Element Method, while the aerodynamic analysis can be performed by analytical, empirical or well-known Computational Fluid Dynamic (Navier-Stokes, Boundary layer). The structural and aerodynamic results are represented in terms of state space models, the resulting model order is large and when systems (for control surface) of the aircraft are included, it increases further. There are several strategies that are used to reduce the order of the state space model. For structural analysis it assumes that the structure can be modelled as a bar or beam element instead of solid elements, this will significantly reduce the order, computation load without compromising on the results accuracy. Nonetheless, there is always a possibility of correcting the state space model based on Ground Vibration Tests prior to Wind Tunnel Tests. For aerodynamic analysis, there are many options as well, ranging from stirp theory to Panel methods to Navier-Stokes equations and much more. Final choice depends upon the fidelity level and computation time. Strip theory is computationally efficient however, has mediocre fidelity level with validity only for low speed flight regime and it neglects tip effects. Effects of one strip along chord does not transfer to the other strip in the spanwise direction. This demands for more accurate representation of flow as the changes in the spanwise and chordwise on the wing. Panel methods provide good approximation of the flow over the entire wing. Navier-stokes solution yields accurate results but takes lot of time as compared to strip theory. Data driven CFD models are also used to reduce the computational time [2]. So, commercially panel methods (Vortex lattice Method and Doublet Lattice Method) are the most used tools in the aeroelastic analysis of the wing. Panel method has reduced accuracy to predict in transonic flow regimes. So, often wind tunnel readings are inserted and corrections are made for the transonic regimes [1-3].

References

[1] M. M. Kasem, Analysis, control, and optimization of aeroelastic systems: an introduction to the recent literature for the new investigator, Journal of Physics: Conf. Ser. 2299 012005, (2022).
[2] M. D. Maughmer and J. G. Coder, Comparisons of Theoretical Methods for Predicting Airfoil Aerodynamic Characteristics, The Pennsylvania State University.
[3] S. Malik, S. Ricci, L. Riccobene, Aeroelastic Analysis of a Slender Wing, Council of European Aerospace societies, CEAS Journal, 2020.

TFD 8: Ionic Current Magnetic Fields in Charged Nanopores and Nanoslits

Abraham Mansouri*
Department of Mechanical Engineering, HCT, Dubai, UAE
*amansouri1@hct.ac.ae

Keywords
Ionic currents, nanopore, nanoslit, magnetic fields, FEM

Abstract
This study explores internal and external magnetic fields generated by pressure-driven ionic currents in charged nanopores and nanoslits. Utilizing finite element solutions for governing equations i.e. Poisson-Nernst- Planck, Ampere, and Navier–Stokes (i.e., PNPANS), the analysis reveals opposite-signed magnetic field components above and below the nanoslit. While non-uniform ionic currents create varying magnetic fields within the nanoslit, external magnetic fields remain constant outside. Also the study shows highly non-uniform magnetic fields within the nanopore that decay as 1/r outside the nanopore. The findings suggest potential applications, such as non-invasive ionic current measurements for estimating zeta potential in solid-liquid interfaces and accurate sequencing of DNA and biomolecules in nanopores and sufficiently long and wide nanoslits. [1]

References
[1] Tabatabaei, S. A., Mansouri, A., Tarokh, A., & Chini, S. F. (2022). Ionic current magnetic fields in 3D finite-length nanopores and nanoslits. The European Physical Journal Plus, 137(3), 312.
[2] Mansouri, A., Tarokh, A., & Taheri, P. (2018). Ionic current magnetic fields in a two dimensional nanoslit. *Chemical Physics Letters*, *710*, 150-156.

TFD 9: New Mini Portable AC using Peltier Modules

Kamel Adref, Alyazya Abdulla Murad Karam Alblooshi*, Khalid Adel Khaleefa Naseeb Alshamsi & Mohamed Yousef Jassim Mohamed Al Ali
Higher Colleges of Technology, Baniyas campus
*H00385532@hct.ac.ae

Keywords
Peltier, Mini AC, Portable

Abstract
Peltier module are finding many applications that require small cooling capacities. A new approach to develop, is presented. This work is the result of capstone design project the objective of which is to develop, build and test a new mini portable air conditioner using Peltier modules as a cooling source. The materials used to build it are carefully selected to make it light. In this regard focus was put on using as light material as possible. Since the amount of cooling produced is expected to be small, light insulating materials are used to minimize any cooling losses from the prototype. The prototype is at its infancy and the measurements are yet to be conducted to analyze its performance. Improvement to the existing prototype are already figured out and recommended for future investigations. It is of great pleasure to share the idea with people in the domain of HVAC to acquire effective feedback, inputs and recommendations.

Reference
[1] A. AuthorOne, B. AuthorTwo, Title of the paper, volume, page number, (year)

TFD 10: Characterization and Testing of the 3D-Printed Microfluidic Mixing Probe

Dima Samer Ali[1,2]*, Ayoub Glia1, Muhammedin Deliorman[1] & Mohammad A. Qasaimeh[1,2]
[1]New York University Abu Dhabi (NYUAD), Abu Dhabi, UAE,
[2]New York University, New York, USA
*dsa337@nyu.edu

Keywords
Microfluidic Probe, Micromixer, spatiotemporal Analysis, concentration gradient

Abstract
Concentration gradients (CGs) are the gears that run major cellular activities in our bodies including growth, inflammation, and wound healing. Reproducing CGs in vitro within a controlled lab setting helps us better understand these phenomena and has vital other applications including drug testing. Spatiotemporal control in drug testing is of paramount importance as it provides precise localization, dosage, and timing of drug application improving efficacy and safety of the drug evaluation process. In the realm of pharmaceutical research, achieving an intricate balance between quantity of administered drug, its specific moment of delivery, and duration needed for its treatment to act can yield profound insights into its efficacy. To this end, we have developed an innovative open microfluidic system integrated with a micromixing function. This cutting-edge technology, the Microfluidic Mixing Probe (MMP), empowers us to apply controlled dosages (ranging from 0% to 100%) in a localized and confined manner, using hydrodynamic flow confinement (HFC), while maintaining a controlled duration of exposure on a targeted subpopulation of adherent cancer cells [1]. This breakthrough approach ensures precise drug administration without jeopardizing the integrity of the entire cell tissue culture sample, enabling us to delve deeper into the complexities of drug-cell interactions and enhance our understanding of drug responses. We have utilized 3D printing technology to speed up the design cycle and the fabrication duration needed for achieving the final product [2,3].

References
[1] D. S. Ali, et al., "Microfluidic Probes Integrated with Micromixers," MicroTAS 2022.
[2] A. Brimmo, et al., "Noncontact Multiphysics Probe for Spatiotemporal Resolved Single-Cell Manipulation and Analyses," Small, 2100801, 2021.
[3] A. Glia, et al., "Herringbone Microfluidic Probe for Multiplexed Affinity-Capture of Prostate Circulating Tumor Cells," Advanced Materials Technologies, 2100053, 2021.

TFD 11: Numerical Investigation of Tesla Valves for Insulin Drug Delivery

Aayush Karna*, Seoyoung Kim & Sunil Kumar
New York University Abu Dhabi
*ak7717@nyu.edu

Keywords
Tesla valve, diodicity, drug delivery

Abstract
Current drug delivery mechanisms often rely on micropumps equipped with valves, such as check valves, to prevent backflow which contain moving parts susceptible to breakdowns. The Tesla valve, with its passive operation, offers a promising alternative to enhance the reliability and durability of these systems by eliminating the need for moving components. In this study, we examine the influence of geometric parameters on the Tesla valve's efficiency—quantified by its diodicity—when utilizing fast-acting insulin aspart (IAsp) as the working fluid. We utilized a 2D model of a Tesla valve to examine the effects of varying the Y-junction angle/valve angle and channel width on the valve's diodicity within a Reynolds number range of 10 to 300. Our results indicated an increase in diodicity when the angle was increased from 35° to 55°, after which the diodicity began to decline. Similarly, an increase in channel width from 65 µm to 125 µm corresponded with an increase in diodicity. Following the parametric study, we implemented a design featuring a Tesla valve with a Y-junction angle of 55° and a channel width of 125 µm to evaluate changes in diodicity across a multistage Tesla valve setup, with the number of stages extending up to 10 for this study. An increase in the number of stages also led to an improvement in the diodicity of the valve, with a maximum diodicity of 3.55 for a 10-stage Tesla valve at a Reynolds number of 300. The enhanced diodicity achieved through optimized geometric configurations of the Tesla valve is crucial for the precise control and delivery of insulin. This ensures unidirectional flow and reduces the risk of dosing inaccuracies caused by backflow, which is especially vital for the effective management of blood glucose levels in diabetic patients.

TFD 12: Rational Design of Low-GWP Refrigerants as Drop-in Replacements through a 4E Analysis using Robust Thermodynamic Models

Carlos G. Albà[1,2*], Ismail I. I. Alkhatib[1], Fèlix Llovell[2] & Lourdes F. Vega[1*]

[1]Research and Innovation Center on CO_2 and Hydrogen (RICH), Dept. of Chemical Engineering, Khalifa University of Science and Technology, Abu Dhabi (UAE)
[2]Dept. of Chemical Engineering, Universitat Rovira i Virgili, Tarragona (Spain)
* carlos.garriga@ku.ac.ae, lourdes.vega@ku.ac.ae

Keywords
Polar soft-SAFT, 4E Analysis, Refrigeration, Sustainable Refrigerants, Molecular Modeling

Abstract
In light of the F-gases emission prospects in the near future, the Kigali Amendment has pledged to limit the production and distribution of high global warming potential (GWP) third-generation refrigerants in contemporary refrigeration and air conditioning systems (RAC). Significantly, approximately 20% of current RAC units employing hydrofluorocarbons (HFCs) are expected to undergo complete replacement in the ensuing years, aligning with eco-conscious European directives [1,2]. To address this concern, the polar soft-SAFT molecular-based equation of state (EoS) has been employed predictively for the identification and development of alternative low-GWP fluorinated refrigerants as direct drop-ins for prevalent HFC refrigerants in today's RAC systems, specifically R134a (GWP=1300) and R410A (1924) [3]. A comprehensive and precise thermodynamic model has been developed for this purpose, accurately simulating the properties of refrigerants, including vapor-liquid equilibrium, thermophysical, derivative properties, in addition to binary mixtures at a coarse-grain level [4]. The modeling framework is systematically applied to evaluate the suitability of alternative refrigerants, ensuring retrofitting criteria based on a multi-dimensional analysis (4E) encompassing energy, exergy, environmental, and economic characteristics, along with a detailed assessment of safety and technical considerations. This comprehensive methodology is proven effective across various operational conditions and system designs, highlighting its applicability, transferability and accuracy. Additional analysis of environmental impact and projected cost is included to quantify the impact associated with their use and emissions, aiding in the identification of appropriate substitutes from a techno-environmental-economic perspective. The compatibility KPIs are also reproduced for R513A (608), R450A (682), and R32 (675), which are well-recognized alternatives for R134a and R410A in domestic and commercial settings nowadays, providing a detailed comparison among mid- to low-GWP cooling agents. Moreover, a sensitivity analysis is conducted next, focusing on realistic legislative agendas in the near future to enhance their economic feasibility and promote a shift towards their manufacturing, deployment and use. Such innovative approach not only aligns with current scientific needs but also sets a precedent for future explorations in the field, demonstrating the potential of fine-tunning techniques to significantly enhance our understanding and capabilities in the realm of molecular thermodynamics. Overall, this contribution highlights the crucial role of developing an accurate molecular-based equation of state in formulating new agents that boast enhanced safety, and technical performance.

Acknowledgments: This research is supported by Khalifa University of Science and Technology (project RC2-2019-007), and the Spanish Ministry of Science and Innovation MCIN/AEI/10.13039/501100011033, under R + D + I projects STOP-F-Gas, (PID2019-108014RB-C21) and NEW-F-TECH (TED2021-130959B-I00). Carlos G. Albà acknowledges a FI-SDUR fellowship granted by the Catalan Government.

References

[1] M. Schulz, D. Kourkoulas, Off. J. Eur. Union 2014(517) (2014) L150/195-230. https://doi.org/10.4271/1999-01-0874.
[2] European Commission, Off. J. Eur. Union (161) (2006) 12–8.
[3] C.G. Albà, I.I.I. Alkhatib, F. Llovell, L.F. Vega, Renew. Sustain. Energy Rev. 188(September) (2023) 113806. 10.1016/j.rser.2023.113806.
[4] C.G. Albà, I.I.I. Alkhatib, F. Llovell, L.F. Vega, ACS Sustain. Chem. Eng. 9(50) (2021) 17034–48. 10.1021/acssuschemeng.1c05985.

TFD 13: Design and Development of Indirect Evaporative Cooler with Enhanced Heat Transfer and Performance

Hassan Ali[1*], Sabir Rasheed[2], Muzaffar Ali[2] & Nadeem Ahmed Sheikh[3]
[1]Engineering and Energy Program, De Montfort University Dubai, United Arab Emirates.
[2]Mechanical Engineering Department, University of Engineering and Technology, Taxila, Pakistan
[3]Mechanical Engineering Department, International Islamic University, Islamabad, Pakistan
*hassan.ali@dmu.ac.uk

Keywords
Air-conditioning, Indirect Evaporative cooling, Building Energy, Net Zero, Climate Change

Abstract
Cooling demand rises due to climate change and rapid increase in population growth in developing countries. One of the challenges of the present-day is to meet exponentially growing cooling requirements. In addition, according to sustainable development goals (SDGs), specifically SDG 03, SDG 07, and SDG 13, the need of sustainable environment friendly cooling technique for human thermal comfort is essential. Therefore, it is necessary to identify new ways for cooling the built environment. Current paper shall present design evolution of indirect evaporative cooler with focus on materials, design and energy efficiency using the locally available cheap materials. This Maisotsenko cycle based indirect evaporative cooling (IEC) is an eco-friendly and energy efficient technology alternative for space air cooling especially in hot environment conditions. The current work presents seven different working design prototypes and extensive experimental investigations of indirect evaporative cooler. The experimental results showed that the thermal effectiveness of the IECs increases with decreasing the relative humidity and increasing the ambient temperature of the incoming air. Whereas the maximum dew point and wet bulb effectiveness of 0.83 and 1.20 was achieved respectively. Maximum coefficient of performance and cooling capacity of 28 and 4KW was achieved. The developed prototypes have significant potential of scaling up for building air-conditioning and industrial applications in UAE.

The presentation shall focus on earlier publications in international journals, status, future and shall be interesting to share it with wider UAE heat transfer community.

TFD 14: Efficiency Assessment of Solar Thermal Radiation Utilizing Advanced Nanofluid on Stretching Sheet

Saleem Nasir[1,2*] & Abdullalh Berrouk[1,2]
[1]Mechanical Engineering Department, Khalifa University of Science and Technology P.O. Box 127788 Abu Dhabi United Arab Emirates
[2]Center for Catalysis and Separation (CeCas) Khalifa University of Science and Technology P.O. Box 127788 Abu Dhabi United Arab Emirates
*saleem.nasir@ku.ac.ae

Keywords
Thermal radiation; Ternary nanofluid; Bvp4c scheme; Heat source, Artificial neural network

Abstract
Utilizing nanostructured materials to optimize thermal processes for renewable energy is an important strategy that has been applied in numerous engineering disciplines for diverse interferences such as solar energy, solar thermal aircraft and photovoltaic thermal systems. The current research focuses on enhancing the thermal performance of convective flow of a water-based trihybrid nanofluid across a stretching surface incorporating carbon nanotubes (CNTs) in cylindrical, graphene (GNP) in spherical and Al_2O_3 in platelet shapes. Several mechanisms like non-linear heat production, thermal radiation, velocity and thermal slip conditions have all been investigated regarding the flow describing equations. This study assesses the capability of soft computing techniques to determine the behavior of a problem with various connected parameters to resolve this problem. With the help of numerical MATLAB's bvp4c package reference data, the artificial neural network is trained. The investigation illustrates the benefits of using soft computing methods to accurately investigate the behavior of complex flow models. The most important discovery is that, in comparison to mono- and binary-hybrid nanofluids, ternary hybrid nanofluids have a greater thermal responsiveness.

References
[1] X. Wang, X. Xu, S.U.S. Choi, Thermal conductivity of nanoparticle fluid mixture, J. Thermophys. Heat Transfer 13 (1999) 474–480.
[2] S.U.S. Choi, Enhancing thermal conductivity of fluids with nanoparticles, in: ASME Int. Mech. Eng. Congress and Exposition, Vol. 66, San Francisco, CA (United States), 99–105, 1995.
[3] P. Keblinski, J.A. Eastman, D.G. Cahill, Nanofluids for thermal transport, Mater. Today 8 (6), 36–44, 2005.

TFD 15: A Machine Learning Approach to Solving Inverse Problems in Radiative Transfer in Scattering-Absorbing Medium

Wenjun Jiang* & Sunil Kumar
New York University Abu Dhabi, P.O. Box 129188, Saadiyat Island, Abu Dhabi, United Arab Emirates
*wj634@nyu.edu

Keywords
Inverse Problem, Machine Learning, Radiative Transfer

Abstract
In recent years, machine learning has shown great potential in solving problems related to radiative transfer. Traditional numerical methods of radiative transfer are computationally expensive, and machine learning methods are shown to reduce cost while maintaining accuracy. For some inverse problems, machine learning methods can be used to improve the accuracy of the results and provide solutions where traditional methods fail. This study uses a simulated diffuse optical tomography (DOT) system to investigate the feasibility of neural network models to solve the inverse problem and reconstruct images in the scattering-absorbing medium. The inverse process of such a problem is nonlinear, ill-posed, and underdetermined, and the high scattering feature increases the difficulty of solving accurately. Here, we proposed a neural network model with a customized loss function to solve the inverse problem of DOT. The customized loss function embedded the idea of peak signal-to-noise ratio (PSNR) and the image correlation coefficient (ICC) to improve both the optical property predictions and the correlation to the ground truths of the reconstructed images. The results show that the proposed neural network model is more accurate and less time-consuming than traditional image reconstruction methods. With our customized PSNR-ICC loss function, the image correlation coefficient between the original and the reconstructed images improves to 0.99, compared to the 0.5-0.8 range of traditional methods. Also, noisy data leads to only minor image degradation with the neural network model, whereas the traditional reconstruction method is noise-sensitive.

TFD 16: Numerical Study on Influence of Electromagnetic Force on Suppression and Growth of Convection Heat and Mass Transfer in an Asymmetric Enclosure

Ranjit J. Singh*, Sanjairaj Vijayavenkataraman & Sunil Kumar
Division of Engineering, New York University Abu Dhabi, Abu Dhabi, UAE
*rjs9414@nyu.edu

Keywords
Electromagnetic force, Magnetic field, Heat and mass transfer, Natural convection

Abstract
The presence of magnetic fields on heat and mass transfer has a wide range of applications in metallurgical applications, nuclear industries, magnetic separators in lubricants and dryers, etc. However, most of the researchers focused merely on the influence of the magnetic field on heat transfer in a symmetric enclosure [1]. In this study, authors are interested in the study of three-dimensional numerical analysis on the influence of magnetic field and its orientation on the growth and suppression of heat and mass transfer in an asymmetric enclosure at a fixed buoyancy ratio (Thermal buoyancy/Solutal buoyancy). The temperature and concentration are mutually coupled by Dufour and Soret parameters respectively, at fixed Soret and Dufour numbers. The thermal buoyancy in the flow is altered by varying Rayleigh number Ra = 104, 105, and 106 at fixed Prandtl number Pr = 0.025. The present solver is framed in the open-source CFD platform, OpenFOAM 5.0, which incorporates the incompressible Navier-Stokes equation coupled with Maxwell's equation of electrodynamics, energy equation, and concentration equation. The intensity of the magnetic field in x (B_x), y (B_y) and z-direction (B_z) are regulated by the variation in the Hartmann numbers Ha = 0, 25, 50, 100 and 200. The external magnetic field induces the Lorentz force (electromagnetic force) in the flow, which opposes the flow and hence it severely affects the heat and mass transfer. The asymmetric enclosure shows a spectacular variation in the flow pattern for all orientations of the magnetic field. However, the magnetic field parallel to the isothermal/isosolutal walls and orthogonal to gravity (B_z) assists in the regulation of convection flow with improvement in the heat and mass transfer compared to other magnetic fields (B_x, and B_y).

Reference
[1] M. Sathiyamoorthy, A. Chamkha, Effect of magnetic field on natural convection flow in a liquid gallium filled square cavity for linearly heated side wall(s), International Journal of Thermal Sciences 49 (2010) 1856–1865.

TFD 18: A Novel Process to Fabricate Paper Based Microfluidic Devices Using Laminating Plastic Sheets

Shafeek A. Samad[1], Nityanand Kumawat[1], Soja Soman & Sunil Kumar[1,2]
[1]Division of Engineering, New York University Abu Dhabi, Abu Dhabi, P.O. Box 129188, UAE
[2]Department of Mechanical Engineering, New York University, Brooklyn, NY 11201, USA
*sa7490@nyu.edu

Keywords
Paper microfluidic analytical devices (μPADs), low-cost, colorimetric sensor

Abstract
Microfluidic paper-based analytical devices (microPADs) are emerging as simple-to-use, low-cost point-of-care testing platforms. Mostly the devices are fabricated by creating hydrophobic barriers using wax or photoresist patterning on the porous paper sheets. Even though, devices fabricated using these methods are used and tested with wide variety of analytes, still they pose many serious practical limitations for low-cost automated mass fabrication for their widespread applicability. We present an affordable and simple two-step process – Cut and Heat (CH-microPADs) - for the fabrication of hydrophilic channels and reservoirs selectively on a wide variety of porous media such as tissue / printing / filter paper and cloth types such as cotton and polyester, by a lamination/heating process. The technique presents many distinct advantages as compared to existing commonly used methods. The devices possess excellent mechanical strength against bending, folding and twisting, making them virtually unbreakable. They are structurally flexible and showed good chemical resistance to various solvents, acids and bases, presenting a widespread applicability in the areas such as clinical diagnostics, biological sensing applications, food processing, and chemical industry. Fabricated paper media 96 well-plate CH-microPAD configurations were tested for cell culture applications using mice embryonic fibroblasts and detection of proteins and enzymes using ELISA. With a simple two-step process and minimal human intervention, the technique presents a promising step towards mass fabrication of inexpensive disposable diagnostic devices both for resource-limited and developed regions.

References

[1] M. Aghababaie, E. S. Foroushani, Recent Advances In the development of enzymatic paper-based microfluidic biosensors. Biosensors & Bioelectronics, 226, 115131 (2023)
[2] A. W. Martinez, S. T. Phillips, Diagnostics for the Developing World: Microfluidic Paper-Based Analytical Devices. Analytical Chemistry, 82(1), 3, (2010)

TFD 19: Capillary Driven Passive Fog Harvesting by 3D Printed Micro-Cones with Structured Surface

Abubaker S. Omer[1*], Aikifa Raza[1], Maryam AlShehhi[2], Faisal AlMarzooqi[3] & TieJun Zhang[1]
[1] Department of Mechanical and Nuclear Engineering, Khalifa University of Science and Technology, P.O. Box 127788, Abu Dhabi, United Arab Emirates
[2] Department of Civil Infrastructure and Environmental Engineering, Khalifa University of Science and Technology, P.O. Box 127788, Abu Dhabi, United Arab Emirates
[3] Department of Chemical and Petroleum Engineering, Khalifa University of Science and Technology, P.O. Box 127788, Abu Dhabi, United Arab Emirates
*100060963@ku.ac.ae

Keywords
Fog Harvesting; Capillary driven Flow; Surface Structure; Micro-cones

Abstract
Capillary pressure induced by the shape gradient of micro-cones can enable self-propelled droplet transport and promise great potential in many applications including atmospheric water harvesting, self-cleaning surfaces and condensation heat transfer. In order to enhance droplet mobility for fog harvesting, micro-cones with structured surfaces are fabricated by high-resolution 3D printing. Small water droplets in fog form a clamshell shape on surface initially, then they transition into an asymmetric or symmetric barrel shape when growing and moving along the micro-cone axis. In comparison with smooth micro-cone, structured micro-cone exhibits a reduction in water contact angle from 49^{o} to 35^{o} and causes the different dynamics of droplet growth and transport. Preliminary results from a three-minute fog harvesting experiment show that clamshell water droplets on smooth cones remain stationary and only half of them coalesce after growth, while barrel-shaped droplets on structured micro-cones fully coalesce and travel longer distance along the axis. The final droplet collected near the structured micro-cone base has 70% higher surface free energy than that on smooth cone, confirming that micro-cones with surface structures can accelerate the axial movement of droplets for capillarity-driven passive water harvesting.

TFD 20: Experimental Characterization of the Wake Fluxes Behind a Two-Dimensional Bluff Body at Varying Reynolds Number

V. Parezanović*
Khalifa University of Science and Technology, Abu Dhabi, UAE
*Vladimir.parezanovic@ku.ac.ae

Keywords
Bluff body, turbulent wake, wind tunnel experiments, PIV.

Abstract
The wake behind a two-dimensional bluff body is delimited by separated shear layers rolling-up and coupling to create a global mode, the so-called Karman Vortex Street. The distance from the bluff body rear at which this region of recirculating flow is closed is called the vortex formation length. Around this length, the separated shear layers destabilize and start to form large vortices which are being shed downstream alternatively. Phenomena that determine this length have been discussed by Gerrard [1] in his proposed model, stating that the formation length is established when the fluxes of fluid going in and out of the wake are in equilibrium. These fluxes are due to recirculation region being replenished with fluid by the external flow being drawn across the axis of symmetry of the wake, and the expulsion of fluid through entrainment of the large roll-up vortex and also through the separated shear layer along its length. Recently, Stella et al. [2, 3] have proposed a methodology to account for such fluxes from experiments, using velocity fields from particle image velocimetry (PIV) for the case of a two-dimensional flow along a ramp. Based on their approach and using the interpretation of Gerrard, we apply a similar methodology to a wake behind a two-dimensional bluff body at different Reynolds numbers. The near wake velocity field is captured using PIV and fluxes due to the global mode and separated shears are accounted for. The results show how the formation length changes and which fluxes are dominant in different Reynolds number conditions.

References
[1] Gerrard, J. H. (1966). The mechanics of the formation region of vortices behind bluff bodies. Journal of fluid mechanics, 25(2), 401-413.
[2] Stella, Francesco, Mazellier, Nicolas & Kourta, Azeddine 2017 Scaling of separated shear layers: an investigation of mass entrainment. Journal of Fluid Mechanics 826, 851–887.
[3] Stella, F., Mazellier, N., Joseph, P. & Kourta, A. 2018 Mass entrainment-based model for separating flows. Physical Review Fluids 3 (11), 114702.

TFD 21: Flow and Heat Transfer from Cylinder under FIV: Effect of Spacing and Stagger Angle

Md. Islam* & Ussama Ali
Khalifa University of Science and Technology
*didarul.islam@ku.ac.ae

Keywords
Flow-induced vibrations; Heat transfer enhancement; Spacing ratio; Stagger angle; Vortex shedding

Abstract
This study focuses on investigating the heat transfer and flow-induced vibrations in two circular cylinders. The Reynolds number (*Re*) is set at 100, and the reduced velocity (*Ur*) is increased from 2 to 14. The cylinders' dimensionless spacing (*L/D*), arranged in tandem, is changed from 1.5 to 6, and the stagger angle (α) between the cylinders is varied from tandem (0°) to staggered (45°) and side-by-side arrangement (90°). The lift (C_L) and drag (C_D) coefficients, streamwise and transverse vibrational amplitudes (*Ax/D* and *Ay/D*), average Nusselt number (Nu_{avg}), Strouhal number (*St*), and vortex shedding patterns and temperature distribution are analyzed. The results demonstrate that the Nu_{avg} for the downstream cylinder is consistently lower than the upstream cylinder in the tandem arrangement due to the upstream cylinder providing shielding effect. Rising *L/D* to 6 from 1.5 increases the Nu_{avg} by 8.6% and 9.8% for downstream and upstream cylinders, respectively. As the *L/D* increases, the Nu_{avg} for both cylinders increases, implying a reduction in the effect of each cylinder on the other. However, in the staggered and side-by-side arrangement, the Nu_{avg} for both cylinders are similar, indicating that the cylinders have little to no influence on each other. The highest Nu_{avg} is observed at the lock-in condition for all *L/D* and α values. The temperature contours mirror the vorticity contours, suggesting that vortex shedding has a positive effect on heat transfer.

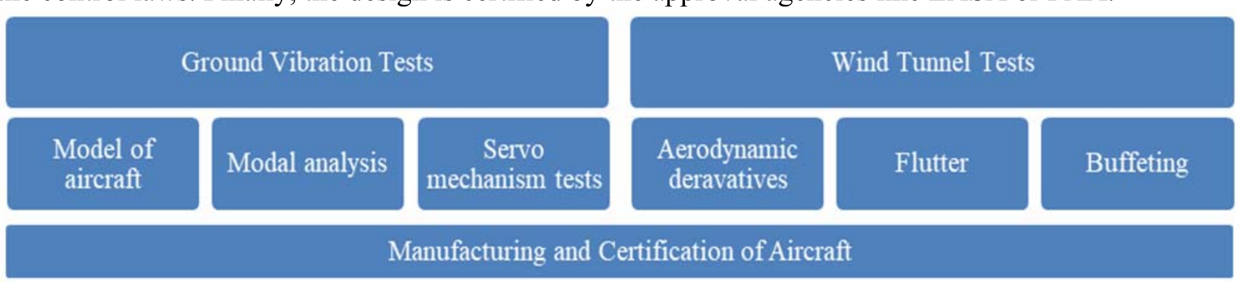
TFD 22: Experimental Testing of Aeroelasticity

Sheharyar Malik* & Shaaban Ali
Abu Dhabi Polytechnic, Al Ain, UAE
*sheharyar.malik@actvet.gov.ae

Keywords
Aeroelasticity, numerical, experimental, validation.

Abstract
Aeroelasticity refers to the elastic motion produced by the structure of the aircraft as a result of coupling with aerodynamic forces. This phenomenon can make the structure vibrate or produce uncontrolled deflection that may leads to fracture. Field of aeroelasticity is broadly categorized into to two main branches i.e. static and dynamic aeroelasticity, this categorization is based on the type of movement or vibration the structure experiences. Divergence, control surface reversal and load redistribution are classified as static aeroelasticity. While, flutter and buffeting are associated with dynamic aeroelasticity. Aeroelastic behavior of the structure imposes the limits on the flight envelope. This emphasizes on the importance of aeroelastic analysis during the design phase of the aircraft. It is pertinent to carry out this analysis for all the aircraft newly designed models.

The validation of aeroelastic behavior initiates from the modelling of the structure, followed by numerical analysis which is supported by experimental testing. Not only one type of test is enough to ascertain the accuracy of aeroelastic analysis, it is achieved by the series of tests, which includes tests in the wind tunnel and ground tests. The experimental validation at the early stage helps to improve the model of the structure and also gives insight into material properties. The ground tests are used to validate the modal analysis at zero wind velocity. The performance and bandwidth are also ascertained for the control surfaces. Test in wind tunnel simulate the flight conditions which are used to evaluate the aircraft aerodynamic derivatives and conditions of flutter and buffeting [1 - 3]. As the structural model has already been corrected through ground vibrational tests, the tests in the wind tunnel helps to validate the aerodynamic models. Sometimes structural coupling is used to ascertain the control laws. Finally, the design is certified by the approval agencies like EASA or FAA.



References
[1] A. De Gaspari, S. Ricci, L. Riccobene, and A. Scotti, Active Aeroelastic Control Over a Multisurface Wing: Modeling and Wind-Tunnel Testing, AIAA Journal, Vol. 47, No. 9, (2009).
[2] S. Malik, S. Ricci, L. Riccobene, Aeroelastic Analysis of a Slender Wing, Council of European Aerospace societies, CEAS Journal, (2020).
[3] S. Malik, L. Riccobene, S. Ricci, D. Monti, Development of a Buffet Load Mitigation system based on multi surface control", International Forum on Aero elasticity and Structural Dynamics, IFASD 2017, Como, Italy, (2017).

TFD 23: Thermo-Acoustically Driven Refrigerator (TADR)

Isam Janajreh* & Omar Al-Mufti
Khalifa University, Abu Dhabi, United Arab Emirates
*isam.janajreh@ku.ac.ae

Keywords
Thermoacoustic engine, thermoacoustic refrigeration, waste energy, CFD simulation

Abstract
Thermo-acoustically Driven Refrigerator (TADR), depicted in Fig. 1, presents a promising and environment-friendly sustainable cooling technology. It consists of 1) a thermoacoustic engine that converts solar/waste heat to acoustic power, and 2) a thermoacoustic refrigerator (TAR) that utilizes the generated acoustic power to pump heat from a lower temperature reservoir. TADRs have no moving parts and uses environment-friendly gasses, such as Argon, Air, and Helium. We will present some of our progress that involves experimental setup and utilizing high-fidelity CFD analysis to investigate the potential of such configuration. Expecially, we focused on the refrigeration components and the obtained temperature difference under different acoustic pressure and drive ration. Numerically, we initially segregated the problem into engine and refrigeration then we coupled the two system in a single simulation. Parametric analyses are done on the stack position, length, and working gas. This lead into two optimal TADR configurations: a) Helium with stacks at xn = 0.43 (TAE) and xn = 0.77 (TAR), prioritizing cooling power, achieving 60.87 W at 5 K temperature difference; b) Argon with stacks at xn = 0.25 (TAE) & xn = 0.58 (TAR), favoring efficiency, achieving up to 22.8%, and a maximum temperature drop of 34 K below ambient (300 K).

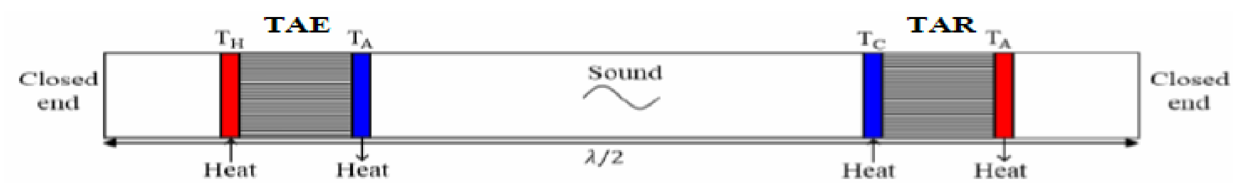


Fig. 1: standing wave Thermo-acoustically Driven Refrigerator schematic

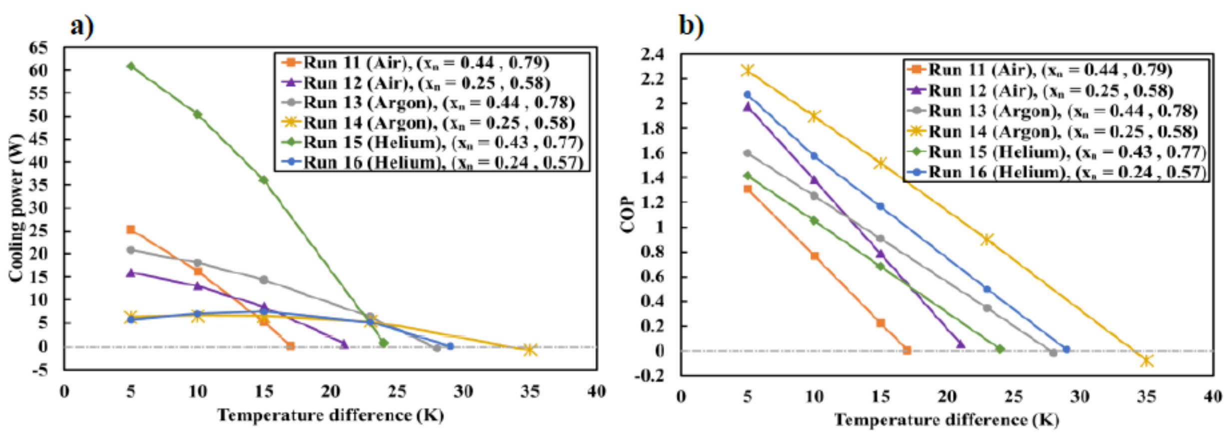


Figure 2: Effect of the working fluid on (a) Cooling power and (b) COP (x_n=TAE normalized position, TAR normalized position)

TFD 24: Herringbone Microfluidic Probe for Multiplexed Affinity-Capture of Prostate Circulating Tumor Cells

Ayoub Glia[1,2]*, Muhammedin Deliorman[1], Pavithra Sukumar[1], Farhad K. Janahi[3], Bisan Samara[1], Ayoola T. Brimmo[1,2] & Mohammad A. Qasaimeh[1,2]

[1]New York University Abu Dhabi, UAE
[2]New York University, USA
[3] Mohammed Bin Rashid University of Medicine and Health Sciences, UAE
*ag6042@nyu.edu

Keywords
Microfluidics, Laminar flow, 3D printing, Circulating Tumor Cells.

Abstract
In this study, we present a novel approach for capturing circulating tumor cells (CTCs) using an open bioactivated substrate equipped with multiplexing capabilities. This achievement is made possible through the development of a novel microfluidic probe (MFP) that integrates radially staggered herringbone (HB) elements for microvortex generation. Referred to as the herringbone microfluidic probe (HB-MFP), this tool represents a channel-less microfluidic system featuring a physically separated bottom capture substrate and a top microfluidics delivery system. The innovative concept enables the functionalization of the capture substrate with multiple biorecognition ligands, such as stripes of different capture antibodies. The microfluidics delivery system can then scan across the substrate in a 2D printing-like movement. Utilizing the HB-MFP, we efficiently captured CTCs from prostate cancer blood samples based on their specific EpCAM, PSMA, and PSA antigens in a single run. Our findings indicate that the multiplex profiles of CTCs reveal distinct cellular phenotypes based on PSMA and PSA expression levels. The developed HB-MFP is characterized by its simplicity and robustness, facilitating high throughput sample processing. Moreover, it provides seamless access to the captured CTCs for further downstream characterization. This novel approach holds promise for advancing our understanding of cancer biology and improving diagnostic capabilities.

TFD 25: Microfluidic Origami Platform for Edible Oil Quality

Pavithra Sukumar[1], Muhammedin Deliorman[1], Fernando Castaño[1], Leen Sameer Ali[4], Dana H. Abujalban, Muhammad Alu Datt[3] & Mohammad A. Qasaimeh[1,2]*

[1]New York University Abu Dhabi (NYUAD), Abu Dhabi, UAE
[2]New York University, New York, USA
[3]Jordan University of Science and Technology, Jordan
[4]Khalifa University, Abu Dhabi, UAE
*maq4@nyu.edu

Abstract

Microfluidics is used in analytical sensing devices for chemical and bio-chemical detection assays. These devices are used to manipulate fluids and detect analytes in the agro-food industry by colorimetric, fluorometric, electrochemical methods. Among all the assays, colorimetric assay is used as qualitative and quantitative method to test food pathogens, allergens, small molecules. To detect small molecules (peroxides in edible-oil), various microfluidic platform are developed [1]. Paper based microfluidic methods are utilized as "point of use" platforms due to its low-cost, portability, applicability. Fluids are manipulated on paper by capillary wicking by different fabrication methods such as valves, geometry cut, fluidic diodes, chemical treatment, dielectric electrowetting, surface effects, dissolvable bridges. Also, three-dimensional paper device by stacking, origami paper platform by folding and folding increases quick response time [2]. In this work, we developed a vertical singleplex and multiplex origami paper device for colorimetric peroxide assay by manipulating fluids to test edible oil quality. Origami device consist of fin-like structures acting as a valve to manipulate deposition of reagents. Origami device and singleplex device utilizes low-volume to measure peroxide level and peroxide value in edible oil. Origami based multiplex device with three herringbone "fin-like" structures consisting of two arms (part 1 and part 2) as a passive valve at an angle of 45° to deposit different chemical reagents for peroxide assay. Part 1 arm is loaded with potassium iodide and part 2 loaded with starch for the reaction. After the reaction, visualization of color change from brown (low peroxide value) to deep brown (high peroxide value) as "yes or no" to assess edible oil quality. The developed device could test simultaneously phenols, peroxides in edible oils to be used as multiplex tool.

References

[1] Arshad F, Deliorman M, Sukumar P, Qasaimeh MA, Olarve JS, Santos GN, Bansal V, Ahmed MU. "Recent developments and applications of nanomaterial-based lab-on-a-chip devices for sustainable agri-food industries," Trends in Food Science & Technology, 136, 145-158, 2023.

[2] Fu, H., Song, P., Wu, Q. *et al.* "A paper-based microfluidic platform with shape-memory-polymer-actuated fluid valves for automated multi-step immunoassays," *Microsyst Nanoeng* 5, 50, 2019.

TFD 26: Spatial Prediction of Dust Distribution in Solar Fields using Two Phase CFD Approach

Hamza Fiaz[1*], Ali Al Masabai[2], Maryam R. AlShehhi[3] & TieJun Zhang[1]

[1]Department of Mechanical Engineering, Khalifa University of Science and Technology, UAE
[2]Shams Power Company, P.O. Box 54115, Abu Dhabi, UAE
[3]Department of Civil Infrastructure and Environmental Engineering, Khalifa University of Science and Technology, UAE
*100058650@ku.ac.ae

Keywords
Spatio-temporal prediction, Two Phase CFD technique, Dust deposition

Abstract
Conventional mirror and panel cleaning strategies for solar power plants are water-intensive and inducing high operational and maintenance costs (O&M). A better approach is required to identify the dust-concentrated solar collectors and panels for targeted cleaning. This reserach aims at developing two-phase computational fluid dynamics (CFD) approach to study the spatial dust distribution of solar field in concentrated solar power (CSP) plants. A 3D euler-lagrange CFD approach is implemented and validated for SHAMS-1 CSP plant, which is located in the desert of western Abu Dhabi. Meteorological parameters (wind speed, wind direction and relative humidity) and geometrical parameters (plant size, plant shape, number and size of solar collectors) are taken from the actual CSP plant to build a CFD model, in which air is the primary phase and dust particle is the secondary phase deposited in solar field. Two extreme cleaning cycles of solar collectors with and without sandstorm are considered for benchmark studies. Spatial validation is done by comparing the obtained reflectivity values with the measured database from SHAMS-1 plant. Our findings indicate that the root mean square error (RMSE) values for regular cleaning cycle (without sandstorm) varies from 7.1% to 8.1%, while for irregular cleaning cycle (with sandstorm) it varies from 7.2% to 12.4% between the first and the last day of the cleaning cycle. Hence, this CFD model is well qualified (rmse < 10%) to identify the spatial reflectivity of solar field, and it can also be applicable to similar solar power plants in desert environment. Instead of time-consuming experiments, the proposed CFD approach can predict the dust-concentrated mirrors and solar panels for on-demand cleaning, which can significantly reduce the water usage and O&M costs.

References
[1] F. Terhag, F. Wolfertstetter, S. Wilbert, T. Hirsch, O. Schaudt "Optimization of Cleaning Strategies Based on ANN Algorithms Assessing the Benefit of Soiling Rate Forecasts," AIP Conference Proceedings, vol. 2126, 220005, 2019
[2] C. Fountoukis, B. Figgis, L. Ackermann, and M. A. Ayoub, "Effects of atmospheric dust deposition on solar PV energy production in a desert environment," Solar Energy, vol. 164, pp. 94–100, 2018
[3] H. Lu and W. Zhao, "Effects of particle sizes and tilt angles on dust deposition characteristics of a ground-mounted solar photovoltaic system," Applied Energy, vol. 220, pp. 514–526, 2018
[4] R. J. Mustafa, M. R. Gomaa, M. Al-Dhaifallah, and H. Rezk, "Environmental impacts on the performance of solar photovoltaic systems," Sustainability, vol. 12, no. 2, 2020

TFD 27: Experimental Study of Ferrofluid Slug Flow under a Uniform Magnetic Field

Haider Ali Khan[1], Afshin Goharzadeh[1]* & Firas Jarrar[2]
[1]Khalifa University of Science and Technology, Abu Dhabi, UAE.
[2]Al Hussein Technical University, Jordan.
*afshin.goharzadeh@ku.ac.ae

Keywords
Two phase flow, Ferrofluid, Magnetic field effect, Ferrohydrodynamic

Abstract
The transport of two-phase Ferrofluid liquid-gas slug flow inside a transparent capillary tube and under the influence of external magnetic field is studied (Fig.1). The uniform magnetic field (250G) is generated by two separated electromagnets located along the horizontal axis of the capillary tube having an internal diameter of 2.5mm. The Ferrofluid is flowing with a constant Reynolds number of Re=36. Measurements were made using pressure transducers and a CCD camera. Recorded images were processed using in-house MATLAB image processing tool. Liquid slug front velocity is measured for two aspect ratios (L/D) of 16 and 8 respectively. It was observed that when slug flow travels through two magnets, its velocity is significantly affected and depends on liquid slug length. The dynamics of slug flow will depend on the interaction of the Ferrofluid with the magnetic field. The velocity profile of slug front is similar for both aspect ratio when they pass through the first magnetic field. However, their significant changes in the velocity profile is observed when slugs reach the second magnet.

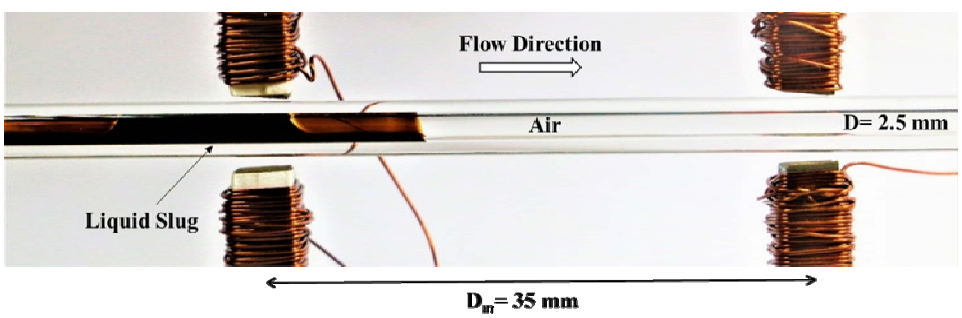


Figure 1: Image represents the test section containing the horizontal glass capillary tube placed under two electromagnets with a ferrofluid slug flowing from left to right [1].

References
1. Khan H. A., Goharzadeh A. and Jarrar F. (2023) “Effect of a Uniform Magnetic Field on a Two-Phase Air-Ferrofluid Slug Flow”, *J. Magn. Magn. Mater*. Vol. 580, pp. 170944.
2. Khan H.A., Goharzadeh A., and Jarrar S.F. (2022) “Design of an Innovative Magnetic-Nanofluid Heat Pipe for Space Applications” *in Proceedings of the* 73rd The International Astronautical Congress (IAC2022), 18-22 September, 2022, Paris, France.

TFD 28: Modeling of Simultaneous Particle Deposition and Deposit Erosion

Li Hongying, Md. Didarul Islam[2], Afshin Goharzadeh[2] & Yap Yit Fatt[2]*
[1]Institute of High Performance Computing, Astar, Singapore.
[2]Khalifa University of Science and Technology, Abu Dhabi, UAE.
*yap.fatt@ku.ac.ae

Keywords
Particle deposition, deposit erosion, level-set method

Abstract
Particle deposition with simultaneous deposit erosion occurs in many engineering applications. It can occur on both stationary surfaces (e.g. heat exchangers with the consequences of an increase pressure drop and a decrease heat transfer) and moving surfaces (e.g. turbine blades with the consequences of a decrease in aerodynamic performance accompanied possibly with stability issues). In effect, the overall efficiency of the engineering systems is compromised.

In a particle deposition process, particles are transported through convection, diffusion, thermophoresis, turbophoretic, etc to the solid surface and then attach themselves to the surface through flocculation, coagulation, etc to form a deposit layer on the solid surface. The formed deposit layer is often simultaneously subjected to deposit erosion due to for example fluid flow generated shear stress. As the deposit layer is partially porous or even impermeable to fluid flow, its growth changes fluid, particle and energy transports in a fully-coupled manner, leading to the complication of having a moving boundary problem in terms of modeling.

The present study develops a numerical model to address such a moving boundary problem. A numerical model with flow, heat transfer, particle transport, particle deposition and deposit erosion fully-coupled is employed. The fluid-deposit interface is evolved implicitly using a level-set method with the interfacial velocity derived from the net effect of combined particle deposition and deposit erosion. The governing equations for conservation of mass, momentum, energy and species are solved using a finite volume method on a fixed regular mesh.

The applicability of the numerical model is then demonstrated for various problems including fouling on cross-flow heat exchanger and plates. The effect of various interesting dimensionless parameters will be briefly discussed.

TFD 29: Multi-Modal Microfluidic Bend Sensor for Soft Sensing Applications

Wael Othman[1,2] & Mohammad A. Qasaimeh[1,2]*
[1]Mechanical and Biomedical Engineering, New York University Abu Dhabi, United Arab Emirates
[2]Mechanical and Aerospace Engineering, Tandon School of Engineering, New York University, NY 10012, USA
*maq4@nyu.edu

Keywords
Soft Sensing, Bending, Twisting, Pressure, Microfluidics

Abstract
Bend sensors play a pivotal role in monitoring the dynamic behavior of flexible mechanical systems and enabling precise feedback control [1]. This study introduces an innovative microfluidic-based multi-modal bend sensor and highlights its potential for soft sensing applications. Our novel sensor consists of a flexible substrate enclosing a serpentine microchannel filled with Galinstan liquid metal acting as the sensing element. Upon bending, the deformation of the sensor induces change in the length and cross-sectional area of the microchannel, as seen in the figure. Correspondingly, the electrical resistance changes according to the direction of bending, enabling a differential sensing. To enhance the sensing response, we incorporated rectangular through-voids underneath the microchannels that amplify the deformation induced on the microchannel. Our optimized fabrication protocol utilized 3D printing and soft lithography techniques, and the experimental characterization demonstrates the sensor's effectiveness and versatility under various bending conditions. Moreover, the functionality of our sensor extends to measuring applied pressure and determining the degree of twisting. Overall, our soft and flexible multi-modal bend sensor has the potential of seamless integration into wearable technology and soft robotics applications, providing real-time insights into bending, pressure, and twisting for enhanced safety and control.

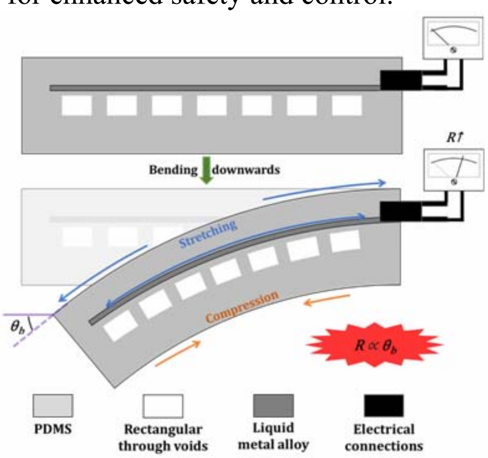


Figure 1

References
[1] Y. Hao et al. (2022) Chinese Journal of Mechanical Engineering, 35, 37.

TFD 30: Modeling Molten Corium-Concrete (MCCI) Interaction with the CFD Approach: Impact of various Boundary Conditions on MCCI

Ilyas Khurshid*, Yacine Addad & Imran Afgan
Khalifa University, Abu Dhabi, P.O. Box 12277, United Arab Emirates.
*ilyas.khurshid@ku.ac.ae

Keywords
Molten Corium, Concrete Ablation, Computational Fluid Dynamics, boundary conditions

Abstract
In the event of a severe nuclear accident resulting in a core meltdown, the molten corium which is a mixture of the molten fuel, cladding, and structural elements, originating in the reactor core could penetrate through the reactor pressure vessel to interact with concrete structure underneath. This research paper presents the numerical modeling of molten corium concrete interaction. The complex phenomena of molten corium concrete interaction and melting of concrete are simplified by considering multi-region with the change in phases taking place only within each predefined region with the assumption that the corium is a homogenous mixture of molten nuclear fuel, cladding, thermo-hydraulic and structural element. This study presents the use of the OpenFoam a Computational Fluid Dynamics (CFD) simulator where a new solver is developed to model the molten corium concrete interaction, its melting, solidification, and concrete ablation for the first time. Two sets of experimental data are used to validate the developed solver and demonstrate the thermal modeling and heat transfer capabilities of the developed solver for concrete ablation under severe conditions. We analyzed different boundary conditions and found that they had a pronounced effect on mitigating ablation and reactor integrity in case of a nuclear accident. In addition, the water-cooled boundary condition was found to be the controlling boundary condition to mitigate concrete ablation. The concrete ablation mechanisms during MCCI are very case-dependent on the concrete solidus, liquidus, and ablation temperatures.

TFD 31: Flow Mixing in Cold-Leg of a Pressurized Water Reactor: CFD Modeling Approach

Mubashir Hassan* & Yacine Addad
Khalifa University, Abu Dhabi, P.O. Box 12277, United Arab Emirates.
*mubashir.hassan@ku.ac.ae

Keywords
Cold-leg mixing, Large Eddy Similar, Unsteady Reynolds-average Naiver-Stokes

Abstract
The cold leg mixing phenomenon of the pressurized water reactor (PWR) was investigated applying the computational fluid dynamics (CFD) simulations. The current study employs CFD simulation to characterize the transient mixing of the two fluids in cold leg and downcomer regions in order to provide information associated with buoyancy effects on nuclear reactor safety. Therefore, the CFD simulations were performed on the two approaches, specifically the Large-Eddy Simulation (LES) and unsteady Reynolds-Average Navier-Stokes (URANS) simulation, to assess their capabilities for the prediction of transient mixing behavior in Cold Leg Mixing (CLM) phenomena. The mixing behavior and instabilities of the two miscible fluids were investigated applying the LES. The Subgrid-scale (SGS) models were used in a series of simulations, i.e., Smagorinsky model, Dynamic model, and WALE model, to identify the effect of the SGS models on the simulation results.

The time-average mean and root-mean-square (RMS) velocity were validated with PIV data in cold leg and downcomer for the Subgrid-scale models. The mean and RMS velocity prediction in the cold leg and downcomer for the first interval is a remarkable agreement with PIV data. Furthermore, the large Kelvin-Helmholtz instabilities are developed on the interface of the two opposite streams during the fluid propagation in the cold leg, after the heavy fluid release in the downcomer producing a strong turbulent mixing.

URANS simulation was conducted for the cold leg mixing phenomenon. The assessment of the turbulence models (i.e., realizable *k*-ε, standard *k*-ε, and *k*-ω SST) with PIV data was studied. The base mesh well predicted the mean and RMS velocity compared to fine mesh. The RKE and SKE turbulence models accurately predicted the mean and RMS with PIV for the first interval in the cold leg, according to the turbulence models assessment. Likewise, as compared to the *k*-ω SST model, the RKE and SKE turbulence models overestimated the mean velocity of the downcomer.

Furthermore, the transient CFD techniques URANS and LES were compared. Both techniques were capable of predicting very well the numerical data with PIV in the cold leg. LES resolved the significant development of Kelvin-Helmholtz instabilities in the cold leg, where the URANS simulation was unable to resolve the Kelvin-Helmholtz instabilities.

References
[1] Hassan, Mubashir, Jinbiao Xiong, and Xu Cheng. "Large-eddy simulation of the OECD-NEA cold-leg mixing benchmark." Annals of Nuclear Energy 167 (2022): 108826.
[2] Hassan, Mubashir, Jinbiao Xiong, and Xu Cheng. "Unsteady RANS simulation of OECD-TAMU cold-leg mixing benchmark." Nuclear Engineering and Design 372 (2021): 110978.

8 Contact Us

For inquiries and to obtain further information, please visit the website (https://ku.events/thermo-fluid/)

For other inquires or feedback, please email us at either hamid.abderrahmane@ku.ac.ae, afshin.goharzadeh@ku.ac.ae or yap.fatt@ku.ac.ae.