UNIVERSITY OF LJUBLJANA
Faculty of Mechanical Engineering

EUROTHERM 2024
Book of Abstracts

9TH EUROPEAN THERMAL SCIENCES CONFERENCE

10 – 13 June 2024, Rikli Balance Hotel, Bled, Slovenia

EUROTHERM

EUROTHERM 2024

Book of Abstracts

9TH EUROPEAN THERMAL SCIENCES CONFERENCE

10 – 13 June 2024, Bled, Slovenia

Prof. Dr. Božidar Šarler, Prof. Dr. Laura Vanoli, Dr. Tadej Dobravec
(Editors)

Ljubljana, June 2024

Original title: EUROTHERM 2024, Book of Abstracts, 9th European Thermal Sciences Conference

Editors:

Prof Božidar Šarler, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia & Institute of Metals and Technology, Slovenia.

Prof Laura Vanoli, Department of Engineering, Parthenope University of Naples, Italy.

Dr Tadej Dobravec, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia.

Technical editors: Krištof Kovačič, Matic Cotič, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia.

Published by: Faculty of Mechanical Engineering, University of Ljubljana, Slovenia

Year of publication: June, 2024

Year of printing or production: June 2024

Print run: online version

Price: free, available at: https://eurotherm2024.si/images/Book_of_abstracts.pdf

All abstracts in this publication have been peer-reviewed. The views expressed in the abstracts in this publication are those of the individual authors and are not necessarily shared by the editors.

ISBN: 978-961-7187-07-6

9789617187076

Kataložni zapis o publikaciji (CIP) pripravili v Narodni in univerzitetni knjižnici v Ljubljani
COBISS.SI-ID **198029827**
ISBN 978-961-7187-07-6 (PDF)

Eurotherm - 1 Birmingham, United Kingdom (1992)

Eurotherm - 2 Rome, Italy (1996)

Eurotherm - 3 Heidelberg, Germany (2000)

Eurotherm - 4 Birmingham, United Kingdom (2004)

Eurotherm - 5 Eindhoven, The Netherlands (2008)

Eurotherm - 6 Poitiers (2012), France

Eurotherm - 7 Krakow (2016), Poland

Eurotherm - 8 Lisbon (2020), Portugal

Eurotherm - 9 Bled (2024), Slovenia

This Book of Abstracts contains the abstracts of the papers presented at the Ninth European Thermal Sciences Conference (EUROTHERM 2024) held at Bled, Slovenia, from June 10 - 13, 2024.

The conference was chaired by Prof Božidar Šarler, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia, and co-chaired by Prof Laura Vanoli, Department of Engineering, Parthenope University of Naples, Italy.

The Scientific Advisory Board of the conference was composed of the Eurotherm Committee members:

Prof Yann Bartosiewicz (Belgium)

Prof Carlos Castro (Portugal)

Prof Gian Piero Celata (Italy)

Prof Pedro Coelho (Portugal)

Prof Alberto Coronas (Spain)

Prof Michel De Paepe (Belgium)

Prof Paolo Di Marco (Italy)

Prof Maria Founti (Greece)

Prof Stephan Kabelac (Germany)

Prof Reijo Karvinen (Finland)

Prof Sasa Kenjeres (The Netherlands)

Prof Christophe Le Niliot (France)

Prof Denis Lemonnier (France)

Prof Gianluca Morini (Italy)

Prof Hans Müller-Steinhagen (Germany)

Prof Andrzej Nowak (Poland)

Prof Björn Palm (Sweden)

Prof Ioannis Sarris (Greece)

Prof Tim Persoons (Ireland)

Prof Karl Ponweiser (Austria)

Prof Jean-Bernard Saulnier (France)

Prof Khellil Sefiane (United Kingdom)

Prof Mina Shahi (The Netherlands)

Prof Anton van Steenhoven (The Netherlands)

Prof Peter Stephan (Germany)

Prof Bengt Sundén (Sweden)

Prof Janusz Szmyd (Poland)

Prof Božidar Šarler (Slovenia)

Prof Vishwas Wadekar (United Kingdom)

The Organising Committee of the conference was composed of the following members of the Faculty of Mechanical Engineering, University of Ljubljana:

Prof Iztok Golobič

Prof Marko Hočevar

Prof Tomaž Katrašnik

Prof Andrej Kitanovski

Prof Jože Kutin

Dr Boštjan Mavrič

Prof Sašo Medved

Prof Jurij Prezelj

Prof Uroš Stritih

The Rector of the University of Ljubljana, Prof Gregor Majdič, and the Dean of its Faculty of Mechanical Engineering, Prof Mihael Sekavčnik, served on the conference's Honorary Committee.

The Head of the research of aluminium industry Impol 2000, Slovenia, Dr Peter Cvahte, and the Head of the research of steelworks Štore-Steel, Slovenia, Dr Miha Kovačič, served on the conference's Industrial Committee.

The conference was organised under the auspices of the Automatic Control Society of Slovenia, the Slovenian Society of Materials, the Slovenian Society of Mechanics, and the Slovenian Society of Simulation and Modelling.

The plenary European Thermal-Sciences Conferences have been taking place since 1992, more than 30 years ago. This year's follows successful conferences in Birmingham (1992, 2004), Rome (1996), Heidelberg (2000), Eindhoven (2008), Poitiers (2012), Krakow (2016), and Lisbon (2020).

The Ninth European Thermal-Sciences Conference is being organised by the Eurotherm Committee and the University of Ljubljana, Faculty of Mechanical Engineering, Slovenia.

This conference aims to provide a forum for the exposure and exchange of ideas, methods and results in heat transfer, fluid mechanics and thermodynamics.

Conference topics include, but are not limited to, the following fundamentals:

Adsorption and desorption, Boiling and condensation, Combustion, Computational/numerical methods, Conduction, Convection, Electrochemical transport, Evaporation, Mass transfer and drying, Measurement techniques, Melting and solidification, MHD and plasmas, Micro/Nano-scale heat transfer, Molecular transport, Multiscale modelling, Porous media, Thermal radiation, Thermophysical properties, Turbulent transport, Two-phase/multiphase flows.

and applications:

Aerospace and aeronautics technology, Biomedical engineering, Cryogenics, Electronic cooling, Energy engineering, Environmental applications, Fires, Heat and mass transfer education, Heat exchangers, Heat transfer in buildings, Inverse problems, Manufacturing processes, Materials processing, Micro-electro-mechanical systems, Miniaturised systems for chemistry and life sciences, Miscellaneous applications, Nanotechnology, Nuclear energy, Optimisation in heat and mass transfer.

We are grateful to the following special session organisers:

Prof Miguel Muñoz Rojo, Spanish National Research Council, Spain & University of Twente, Netherlands & Prof Andrej Kitanovski, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia: Special session on "Thermal Control Devices and Thermal Circuits".

Dr Dominic Groulx, Department of Mechanical Engineering, Dalhousie University, Canada & Dr Monica Delgado, Thermal Engineering and Energy Systems Group, University of Zaragoza, Spain: Special session on "Thermal Energy Storage".

Prof Sara Rainieri, University of Parma, Italy & Prof. Marcelo Colaço, Universidade Federal do Rio de Janeiro, Brazil, Special session on "Morphology Optimized Design for Heat Exchangers".

Prof Ching Shyang Chen, University of Southern Mississippi, USA & Dr Boštjan Mavrič, Institute of Metals and Technology, Slovenia: Special session on "Advances in Meshfree Methods with Applications in Thermal Sciences".

Prof Matteo Bucci, Massachusetts Institute of Technology, USA & Prof Iztok Golobič, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia: Special session on "Recent Advances in Boiling and Condensation Heat Transfer".

Prof Ryszard Białecki, Faculty of Energy and Environmental Engineering, Silesian University of Technology, Poland & Prof. Alain Kassab, Mechanical and Aerospace Engineering Department, University of Central Florida, USA, Special session on "Advanced and Multiscale Computational Methods in Bioengineering".

Dr Miha Založnik, CNRS - Institut Jean Lamour, France & Prof Božidar Šarler, Faculty of Mechanical Engineering, University of Ljubljana, Slovenia, Special session on "Solidification Science and Technology".

We thank the participants of the conference and the outstanding invited plenary speakers:

Dr Saša Bajt, DESY, Germany

Managing high heat loads in extreme X-ray optics

Prof Matteo Bucci, Massachusetts Institute of Technology, United States

Far away, so close: high-resolution investigations of boiling heat transfer, from cryogenic fluids to high-pressure water

Prof Nurupam Chakraborti, Czech Technical University in Prague, Czech Republic

Data-driven evolutionary deep learning in high-temperature basic oxygen steelmaking operation

Prof Alain Kassab, University of Central Florida, USA

Multiscale modelling in congenital heart disease

Prof Wojciech Lipiński, The Cyprus Insitute, Nicosia, Cyprus

Multiphase transport phenomena in high-temperature solar thermal systems

Prof Andreas Ludwig, Montan University Leoben, Austria

Multiphase/physic modelling of solidification with industrial relevance

Prof Perumal Nithiarasu, Swansea University, UK

Digital twins of thermal systems

Prof Dimos Poulikakos, ETH Zurich, Switzerland

The architecting of nanomaterials for water condensation applications

Special thanks go to the members of the Laboratory for Fluid Dynamics and Thermodynamics, Faculty of Mechanical Engineering, University of Ljubljana, Assist Prof Andrej Bombač, Assist Prof Matjaž Perpar, Assist Prof Anton Bergant, Assist Prof Boštjan Mavrič, Dr Zlatko Rek, Dr Katarina Mramor, Dr Tadej Dobravec, Dr Miha Kovačič, Dr Robert Vertnik, Dr Qingguo Liu, Dr Umut Hanoglu, Dr Rizwan Zahoor, Khush Bakhat Rana, Gašper Vuga, Izaz Ali, Krištof Kovačič, Bor Zupan, Matic Cotič and Zdenka Rupič for their technical support.

The conference could not have been organised without the excellent support of the Congress and Commercial Programme Director Mrs Breda Pečovnik, MA and the Congress Project Manager Mrs Mateja Peric, Cankarjev Dom, Cultural and Congress Centre, Ljubljana, Slovenia. We also thank Mrs Mateja Legat from Rikli Balance Hotel, Bled, Slovenia, for providing all the necessary local conference venue support.

The Slovenian Research and Innovation Agency supported the conference under the core funding P2-0162 Multiphase Systems and project J2-4477 Development of Innovative Meshless Methods for Multiphysics and Multiscale Simulation of Cutting-Edge Technologies.

All abstracts in this volume have been peer-reviewed. The number of papers presented at this year's conference represents an extraordinary vitality of thermofluid sciences. We hope these proceedings will document the Bled event and assess achievements and new paths in heat transfer, fluid mechanics and thermodynamics research. The level and quality of contributions demonstrate that thermofluid sciences maintain their position as a subject of central importance in traditional and newly developing areas.

Finally, we would like to congratulate the winner of the Eurotherm Prize for Young Scientists 2024 Dr Laura Nebot Andrés, Universitat Jaume I, Castelló de la Plana, Spain, and the winners of the Eurotherm Award for Young Scientists 2024, Dr Arianna Berto, Università degli Studi di Padova, Italy and Dr Alekos Ioannis Garivalis, University of Pisa, Italy.

The Editors

Contents

Plenary Lectures

Chairs: Božidar Šarler, Laura Vanoli, Janusz Szmyd, Stephan Kabelac, Ryszard Białecki, Pedro Coelho, Miha Založnik, Iztok Golobič

Plenary Speaker: Dr. Saša Bajt

Lead Scientist at DESY

Saša Bajt, studied physics at the Univ. of Ljubljana. She received her PhD from the Heidelberg University working at the Max Planck Institute for nuclear physics in Heidelberg (Germany). She then accepted a position at the University of Chicago (USA) and worked at the National Synchrotron Light Source at Brookhaven National Laboratory. There she pioneered micro-XANES (micro absorption near-edge spectroscopy) and micro-EXAFS (micro extended x-ray absorption fine structure) techniques for which she was awarded a Hawley medal by the Mineralogical Association of Canada.

She then moved to California where she led a group at Lawrence Livermore National Laboratory working on the development of highly reflective mirrors for extreme ultraviolet lithography (EUVL) program. She invented new multilayer structures that resulted in word-record reflectivity EUV mirrors and developed novel capping layers to prolong their lifetime. Her work on EUV optics has had a major impact to the semiconductor lithography industry. Simultaneously, she developed EUV optics that enabled many of the pioneering experiments at FLASH, at that time the only X-ray free electron laser (XFEL) operated at DESY in Hamburg (Germany).

This led to an invitation from DESY to move back to Europe and establish a new laboratory and a research group there. Her group is currently developing novel multilayer-based x-ray optics such as multilayer Laue lenses (MLL) and pulse compressors. Her MLLs can focus X-rays to below five nanometres. Such highest intensities will bring X-ray imaging methods like microscopy and holography to new levels. When combined with XFELs beams they can help to achieve a long-term goal to image single macromolecules. Dr. Bajt is also leading efforts to develop and improve sample delivery of small protein nanocrystals into XFEL beam. She has established a strong collaboration with the Faculty of Mechanical Engineering (Univ. of Ljubljana) and is actively promoting cooperation with Slovenia by linking Slovenian scientists to cutting-edge research at DESY and its partners. For this she has been awarded the title Ambassador of Science of the Republic of Slovenia, one of the highest national prizes of the Republic of Slovenia. Dr. Bajt was recently promoted to a Lead Scientist at DESY, a position equivalent to a full professorship in Germany. She has over 230 peer reviewed publications, 7 book chapters and 18 patents. She is a Fellow of Optica, SPIE Senior Member and a recipient of many awards.

Managing High Heat Loads in Extreme X-ray Optics

Saša Bajt

DESY, Germany

Email: sasa.bajt@desy.de

X-ray Free Electron Lasers (XFELs) are large accelerator-based X-ray sources that produce extremely intense X-ray light in pulses of femtosecond duration. They can be used, for example, for flash photography of macromolecular crystals to decipher how photosynthetic proteins convert and utilize light energy, to generate states in materials that resemble the conditions in the interior of large planets and stars, or to image shock waves and crack propagation. All of these applications require the ability to control the extreme X-ray pulses—to focus or shape them as needed for a particular measurement. Not only must X-ray optics for XFELs be of exquisite precision to obtain the best optical properties but they must also be able to withstand extreme irradiation conditions.

We are exploring the performance of novel diffractive X-ray optics to focus XFEL beams using multilayer Laue lenses (MLLs). High intensities and nanometer spots were demonstrated with both synchrotron and XFEL beams [1,2]. MLLs are based on synthetic multilayer structures and can be fabricated to a precision below 1 Å by magnetron sputtering. We made substantial progress in the preparation of these 3D nanostructured optical elements due to better understanding of the material properties and at-wavelength optical metrology, which is used to characterize their performance. Based on first measurements at the European XFEL, we are currently developing MLLs that are based on new materials and an improved mounting of these lenses. This development is guided by heat load numerical simulations.

[1] S. Bajt et al., Light Sci. Appl. 7, 17162 (2018).

[2] M. Prasciolu et al., SPIE Proc. Vol. 11886, 118860M-1 (2021).

Plenary Speaker: Prof. Matteo Bucci

Associate Professor of Nuclear Science and Engineering at the Massachusetts Institute of Technology (MIT)

Matteo Bucci is the Esther and Harold E. Edgerton Associate Professor of Nuclear Science and Engineering at the Massachusetts Institute of Technology (MIT). He has published over 40 journal articles and 70 conference papers.

His research group studies two-phase heat transfer mechanisms in nuclear reactors and space systems, develops high-resolution non-intrusive diagnostics and surface engineering techniques to enhance two-phase heat transfer, and creates machine learning tools to accelerate data analysis and conduct autonomous heat transfer experiments. He has won several awards for his research and teaching, including the MIT Ruth and Joel Spira Award for Excellence in Teaching (2020), ANS/PAI Outstanding Faculty Award (2018 and 2023), the UIT-Fluent Award (2006), the European Nuclear Education Network Award (2010), and the 2012 ANS Thermal-Hydraulics Division Best Paper Award.

Matteo serves as the Editor of Applied Thermal Engineering, is the founder and coordinator of the NSF Thermal Transport Café, and works as a consultant for the nuclear industry.

Far Away, So Close: High Resolution Investigations of Boiling Heat Transfer, from Cryogenic Fluids to High-Pressure Water

Matteo Bucci

Massachusetts Institute of Technology (MIT), United States

Email: mbucci@mit.edu

In every field of science, the possibility of discovering and understanding new phenomena or testing new hypotheses is strongly related to and limited by the capability of observation. Here, we will discuss recent advances in experimental boiling heat transfer research made possible by unique experimental facilities and non-intrusive high-resolution optical diagnostics. We will analyze the capabilities and limitations of these techniques in supporting the understanding of fundamental two-phase heat transfer problems, with a focus on extreme boiling conditions such as the boiling of water at high pressure and temperature, close to nuclear reactor conditions, the boiling of dielectric fluids for electronic cooling applications, or the boiling of cryogenic fluids relevant to space propulsion and energy storage. The use of these diagnostics has been instrumental in providing answers to long-standing fundamental questions on the fluid dynamics and heat transfer nature of these processes.

Plenary Speaker: Prof. Nirupam Chakraborti

Co-Editor of the prestigious journal Philosophical Magazine Letters

Professor Nirupam Chakraborti was educated in India and in the USA, receiving his B.Met.E from Jadavpur University followed by an MS from New Mexico Tech, USA and subsequently PhC and PhD degrees from University of Washington, Seattle, USA. He joined Indian Institute of Technology, Kanpur as a member of the faculty in 1984 and switched to Indian Institute of Technology, Kharagpur in 2000. He has now joined the Faculty of Mechanical Engineering of Czech Technical University in Prague after superannuating from Indian Institute of Technology, Kharagpur where he was a Higher Academic Grade Professor of Metallurgical and Materials Engineering.

Professor Chakraborti is a Co-Editor of the prestigious journal Philosophical Magazine Letters.

Internationally known for his pioneering work on evolutionary computation in the area of Metallurgy and Materials, globally, Professor Chakraborti is rated among the top 2% highly cited researchers in the Materials area, as per the recent reports published from Stanford University in the USA. Recently CRC Press has published his book Data-driven Evolutionary Algorithms in Materials Technology simultaneously from the UK and the USA. This book is a culmination of Professor Chakarborti's decades of research and teaching efforts in this area.

He is a former Docent of Åbo Akademi, Finland, former long term Visiting Professors of Florida International University, Miami, USA and POSTECH, Korea, and he also taught and conducted research at several other academic institutions in Austria, Brazil, Finland, Germany, Italy and the US.

An international symposium, under the KomPlasTech 2019, which is the world's longest running conference series in the area of computational materials technology, was organized in Poland in January 2019 to honor him. In 2020, an issue of a prominent Taylor of Francis journal, Materials and Manufacturing Processes was dedicated to him as well. Also, in 2021 Indian Institute of Technology, Kharagpur and Indian Institute of Metals, a professional body, organized another international seminar in his honor.

Data-Driven Evolutionary Deep Learning in High Temperature Basic Oxygen Steelmaking Operation

Nirupam Chakraborti

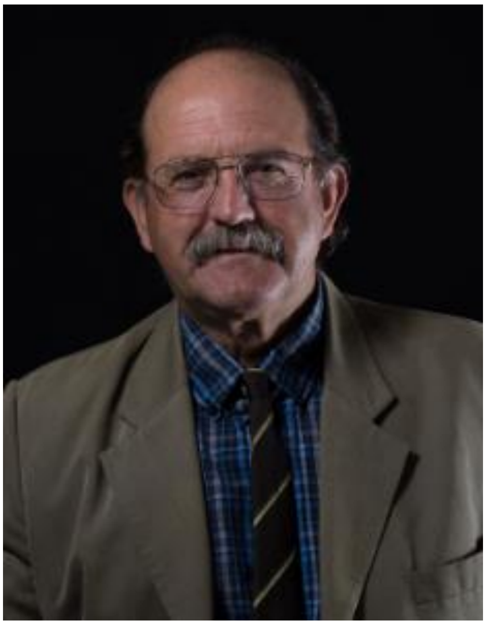
Czech Technical University in Prague, Czech Republic

Email: nirupam.chakraborti@fs.cvut.cz

Basic oxygen steelmaking operation (BOF) is a very complicated high temperature industrial operation. Here in an environment of extreme turbulence, some very complicated chemical reactions take place in order to produce steel. Analytical models exist for this process but they are however often too difficult and cumbersome to apply effectively in a real life scenario. In this situation surrogate models constructed from actual industrial data using evolutionary data-driven modelling can actually make a breakthrough. Here efficient intelligent models are constructed from the nonlinear and noisy plant data. Recent introduction of evolutionary deep learning has further augmented the capabilities of this approach, particularly in the presence of unstructured big data. In this work, the data collected from a basic oxygen furnace is used in the data-driven evolutionary algorithms developed in author's group to come up with the surrogate models. These are evolutionary neural network (EvoNN), bi-objective genetic programming (BioGP), and evolutionary deep neural network (EvoDN2). The target was to reduction of phosphorus to an acceptable level, to limit the carbon content of the steel and also to control the operational temperature effectively. The objectives were constructed using three essential process parameters, temperature, carbon and phosphorus contents, and intelligent models for them were created using 91 process variables of the BOF operation in the plant. The analysis involved a total of around 17000 operational observations and coming up with intelligent surrogate models from them was an extremely difficult task, where deep learning appeared to be essential and only the EvoDN2 algorithm performed at high proficiency. With the generated data-driven models, the optimization work was performed simultaneously on three objectives using a constraint-based reference vector evolutionary algorithm (cRVEA). The optimized results in a multi-dimensional hyperspace were analyzed, and their effectiveness in the BOF steel making was examined in detail.

Plenary Speaker: Prof. Alain Kassab

Director of the Biomedical Engineering (BME) at the University of Central Florida



Professor Kassab earned his BS in Engineering Science 1983 at the University of Florida (UF), his MS in Mechanical Engineering 1985 at UF, and his PhD in Mechanical Engineering in 1989 at UF. He joined the Mechanical and Aerospace Engineering Department (MAE) at the University of Central Florida (UCF) in Fall of 1991, where he served as MAE Graduate Program Coordinator 1996-2003 and 2008-2015. Since 2015 he holds the position of Director of the Biomedical Engineering (BME) Program having launched both the MS in BME (2016) and PhD in BME (2023) programs at his institution. His research interests include bioengineering, computational heat transfer and fluid flow, inverse problems, boundary elements, and meshless methods.

He has authored or co-authored over 400 journal, conference papers, books, and other scientific publications. His research has been supported by the NSF, NASA, NIH, American Heart Association, Orlando Health, Additional Ventures, the Children's Heart Foundation, United Launch Alliance, Siemens, and local industry.

Professor Kassab has received several teaching and research awards at UCF including the 2000 University Excellence in Graduate Teaching Award. He is UCF Trustee Chair Professor and UCF Pegasus Professor. Professor Kassab is editor of the journal Engineering Analysis with Boundary Elements, is Associate Editor of Critical Reviews in Bioengineering, and serves on several editorial boards. He has organized and chaired 11 international conferences including the 5th International Conf. on Engineering Frontiers in Pediatric and Congenital Heart Disease in 2016 in Orlando and has served on numerous international conference scientific committees.

Professor Kassab is Fellow of the American Society for Mechanical Engineering (FASME), Fellow of the American Institute for Medical and Biological Engineering (FAIMBE), and Fellow of the Wessex Institute of Technology (FWIT).

Multiscale Computational Fluid Dynamics Modeling in Congenital Heart Disease

Alain Kassab, Ray Prather, Arka Das, Eduardo Divo, William DeCampli

University of Central Florida, USA

Email: Alain.Kassab@ucf.edu

Computational fluid dynamics (CFD) is seen to be increasingly utilized in a wide range of medical applications including device design and surgical treatment planning. CFD has reached the bedside in the context of personalized medicine. For example, the approach is currently approved to estimate the flow fraction reserve in the treatment planning for coronary artery disease. In cardiovascular applications, imaging from a CT-scan or an MRI of a patient is utilized to render a solid model of the geometry of the section of the vasculature to be analyzed using specialized commercial or open-source medical segmentation software. Alternatively, representative solid models of the anatomical region may be constructed using CAD tools. Pulsatile hemodynamics are then resolved in 3D time-accurate CFD computations of the anatomical region of interest. The time-dependent flow and pressure waveforms boundary conditions driving flow are provided by a 0D lumped parameter model (LPM) of the unresolved peripheral vasculature based on electric circuit analogies of fluid flow. This multiscale 3D CFD - 0D LPM coupling can be carried out in loosely or strongly coupled manner.

Two applications are presented investigating hemodynamics of palliative treatment of hypoplastic left heart syndrome, which is a congenital heart defect affecting nearly 1/3800 babies born each year in the USA, the hallmark of which is the presence of only one functioning ventricle. Palliation involves a series of three staged surgical interventions aimed at restoring a serial circulation powered by the single functional right ventricle, establishing the so-called Fontan circulation. The first example explores the complex hemodynamics of the novel hybrid comprehensive stage 2 operation, focusing on areas of potential clinical concern. Next, the developments and modifications of the injection jet shunt (IJS) that taps into the reserve power of the heart to energize the Fontan circulation and reduce inferior vena cava (IVC) pressure are presented. The IJS is a passive device that may be effective in addressing the mortality and morbidity caused by elevated IVC pressure experienced by many if not most Fontan patients. Management of the complex flow interactions of a confined pulsating jet interacting with a low-speed co-flow within the confines of the physiological constraints of the Fontan is made possible by the multiscale CFD model, which also incorporates an oxygen transport model. Computational results show the feasibility of achieving an IVC pressure reduction of 3 mmHg or more along with clinically acceptable levels of oxygen saturation.

Plenary Speaker: Prof Wojciech Lipiński

Professor at the Cyprus Institute

Wojciech Lipiński is a professor at the Cyprus Institute. He obtained his Master of Science degree in Environmental Engineering from Warsaw University of Technology (2000), doctorate in Mechanical and Process Engineering from ETH Zurich (2004), and habilitation in Energy Technology from ETH Zurich (2009). He previously held academic positions at ETH Zurich (2004–2009), the University of Minnesota (2009–2013), and the Australian National University (2013–2021).

Prof. Lipiński's research interests encompass optical, thermal and chemical aspects of solar energy science and technology. His basic research focuses on advances in transport and reactive flow phenomena, in particular for problems with significant radiative transfer effects. His applied research primarily underpins developments in concentrated solar thermal energy for power generation, processing of fuels and materials, and environmental separations. Prof. Lipiński is currently serving on editorial boards of Solar Energy, Journal of Quantitative Spectroscopy and Radiative Transfer, and Thermopedia. He is involved, among others, in the International Centre for Heat and Mass Transfer, the Eurotherm Committee, and the American Institute of Chemical Engineers.

Multiphase Transport Phenomena in High-Temperature Solar Thermal Systems

Wojciech Lipiński

The Cyprus Insitute, Nicosia, Cyprus

Email: w.lipinski@cyi.ac.cy

High-flux solar irradiation obtained with optical concentrators is a viable source of clean process heat for high-temperature physical and chemical processing. Traditionally, the progress in concentrating solar thermal technologies has been driven by advancements in concentrated solar power, in particular in the context of large-scale dispatchable power generation. Solar thermochemistry is concerned with direct thermochemical production of chemical fuels and materials processing, without intermediate electricity generation, promising high energy conversion efficiency. In this presentation, recent advances in numerical modelling of multiphase transport phenomena in high-temperature solar thermal systems are discussed. Two types of multiphase flows recently investigated for efficient collection, conversion and storage of concentrated solar energy are focused on: (1) particle–gas flows featuring polydisperse particle transport under direct concentrated solar irradiation, and (2) boiling sodium flows. Governing equations and numerical solution methods are elaborated along with selected results obtained for free-falling particle and liquid sodium solar receivers. Examples of on-sun demonstration and pilot systems and the potential for improving the efficiency of solar energy collection, conversion and storage processes are discussed.

Plenary Speaker: Prof. Andreas Ludwig

Board member of the Austrian Science Fund

Andreas Ludwig studied Physics at the University of Düsseldorf, Germany, obtained his Ph.D. in 1992 at the RWTH Aachen, Germany, and a habilitation on Material Physics in 1999 also at the RWTH Aachen. In 1993/1994 he was a scholar of the German Science Foundation at EPFL Lausanne, Switzerland.

In 2003 he became full Professor and head of the chair of 'Simulation and Modelling of Metallurgical Processes', Department Metallurgy, Montanuniversity Leoben, Austria. From 2009 to 2021, he was a board member of the Austrian Committee of the International Institute for Applied System Analysis in Laxenburg, Austria.

Since 2014, he has acted as a board member of the Austrian Science Fund (FWF) responsible for 'Engineering Technology' and now 'Material Science'. Andreas published over 200 peer-reviewed journal papers most in the field of 'modelling solidification at an industrial length scale'. The database SCOPUS reveals that Andreas is the world-leading scientist with regard of publications on macrosegregation (since Spring 2013). In the last years, his focus also comprises magnetohydrodynamics in metal processing, as well as peritectic solidification under microgravity conditions.

Multiphase/physics Modelling of Solidification with Industrial Relevance

A. Ludwig, H. Zhang, Z. Zhang, A. Kharicha, M. Wu

Department Metallurgy, Montanuniversity Leoben, Austria

Email: ludwig@unileoben.ac.at

Many industrial-relevant metallurgical processes reveal multiphase and multiphysical features that need to be properly accounted for when describing the process. Besides the drastic increase in computation power, an increasing number of new advanced models allows nowadays to more and more use numerical tools to optimise production processes and even help to develop new production methods. In this contribution, examples are given that demonstrate the state-of-the-art in using advanced numerical tools to understand phenomena that occur during the pro-duction of single-crystal turbine blades and to optimise the process control by applying electro-magnetic stirring during continuous casting. The higher the complexity the more possible mis-understanding and resulting errors might be. Therefore, we also demonstrate experimental ways of evaluating sophisticated new-model approaches. Finally, we give an example of how a new semi-continuous casting method applying a time-dependent and moving electromagnetic stirring device should be operated; a promising new process route that was developed solely by ad-vanced numerical tools.

Plenary Speaker: Prof. Perumal Nithiarasu

Computational Engineering Professor and the Associate Dean for Research, Impact, and Innovation at the Faculty of Science and Engineering.

Perumal Nithiarasu (PN) is a Computational Engineering Professor and the Associate Dean for Research, Impact, and Innovation at the Faculty of Science and Engineering, Swansea University. With a career spanning nearly three decades, PN has made significant contributions to computational fluid dynamics, biomedical engineering, and heat transfer. PN's expertise in computation heat transfer is currently focused on the interface between physics-based models and machine learning.

PN has a prolific publication record, with over 300 articles and two books to his credit. PN's contributions have been recognised with prestigious awards, including the Zienkiewicz Silver Medal from ICE London in 2002, the ECCOMAS Young Investigator Award in 2004, and the esteemed EPSRC Advanced Fellowship in 2006. He holds the position of Founding Editor-in-Chief of the International Journal for Numerical Methods in Biomedical Engineering, published by Wiley-Blackwell. PN was elected a Fellow of the Learned Society of Wales in 2018.

Digital Twins of Thermal Systems

Perumal Nithiarasu

Swansea University, UK

Email: p.nithiarasu@swansea.ac.uk

Recent strides in artificial intelligence (AI) have brought about transformative changes across various engineering domains, particularly manufacturing and healthcare [1]. This impact extends to thermal sciences, where machine and deep learning integration is becoming increasingly prevalent [2]. This lecture aims to provide an insightful overview of the current state of the art in AI applications within thermal engineering, focusing on the emerging realm of digital twins.

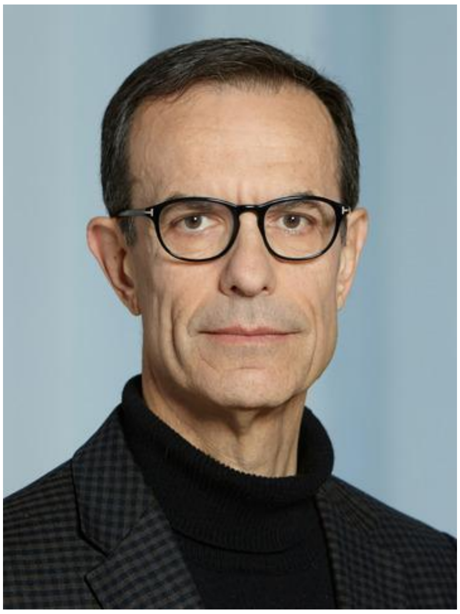
Digital twinning, defined as a virtual platform where a physical entity and its digital counterpart continually influence each other until achieving a desired outcome, presents distinct challenges. In thermal sciences, where measurements are often sparse and prone to noise, the first significant challenge lies in integrating this sparse data into physics-based models to reconstruct a comprehensive solution [4].

The lecture will address this challenge's intricacies, emphasising the need for accurate reconstruction models. Even if such models are feasible, they often are not cost-intensive and real-time solution reconstruction is a challenge. For digital twins to effectively control physical entities, especially in extreme thermal environments, nearly real-time transient solution reconstruction becomes imperative [5].

1. N.K. Chakshu and P. Nithiarasu, An AI based digital-twin for prioritising pneumonia patient treatment, IMECHE Journal – Part H, 236,1662-1674, 2022.
2. W. Bielajewa, M. Tindall and P. Nithiarasu, Comparative study of transformer- and LSTM-based machine learning methods for transient thermal field reconstruction, Computational Thermal Sciences: An International Journal, (2024).
3. A Di Meglio, N. Massarotti and P. Nithiarasu, A physics-driven and machine learning-based digital twinning approach to transient thermal systems, International Journal of Numerical Methods for Heat & Fluid Flow, (2024).
4. H.T.Jahromi, N.K.Chakshu, I. Sazonov, L. Evans, H. Thomas and P. Nithiarasu, Data-driven inverse modelling through neural network (deep learning) and computational heat transfer, Computer Methods in Applied Mechanics in Engineering, 369, 113217, 2020.
5. W. Bielajewa, M. Tindall and P. Nithiarasu, A novel, finite-element based framework for sparse data solution reconstruction and multiple choices, (submitted, 2024)

Plenary Speaker: Prof. Dimos Poulikakos

Director, Laboratory of Thermodynamics in Emerging Technologies, Department of Mechanical and Process Engineering



Professor Dimos Poulikakos held the Chair of Thermodynamics at ETH Zurich from July 1st, 1996 until January 31st, 2024. He served as the Vice President of Research of ETH Zurich in the period 2005-2007. He was the ETH director of the IBM-ETH Binnig-Rohrer Nanotechnology center, (2008-2011). He served as the Head of the Mechanical and Process Engineering Department at ETH Zurich (2011-2014). He was the Chairperson of the Energy Science Center of ETH Zurich (2018-2021). He is a member of the Swiss National Academy of Engineering. He is currently the president of the division Programs of the Swiss National Science Foundation (SNF).

His research is in the area of interfacial transport phenomena, thermodynamics and related materials nanoengineering, with a broad range of related applications. The focus is on understanding the related physics, in particular at the micro- and nanoscales and employing this knowledge to the development of novel, disruptive technologies.

Among the awards and recognitions, he has received for his contributions are the White House/NSF Presidential Young Investigator Award in 1985, the Pi Tau Sigma Gold Medal in 1986, the Society of Automotive Engineers Ralph R. Teetor Award in 1986, the University of Illinois Scholar Award in 1986 and the Reviewer of the Year Award for the ASME Journal of Heat Transfer in 1995. He is the recipient of the 2000 James Harry Potter Gold Medal of the American Society of Mechanical Engineers.

He was a Russell S. Springer Professor of the Mechanical Engineering Department of the University of California at Berkeley (2003) and the Hawkins Memorial Lecturer of Purdue University in 2004. He received the Heat Transfer Memorial Award for Science in 2003 from ASME. In 2008 he was a visiting Fellow at Oxford University and a distinguished visitor at the University of Tokyo. He is the recipient of the 2009 Nusselt-Reynolds Prize of the World Assembly of Heat Transfer and Thermodynamics conferences. He is the 2012 recipient of the Max Jacob Award, for eminent scholarly achievement and distinguished leadership in the field of fluidics and heat transfer. He was presented with the Outstanding Engineering Alumnus Award of the University of Colorado in Boulder in 2012. He received the Dr.h.c. of the National Technical University of Athens in 2006.

The Architecting of Nanomaterials for Water Condensation Applications

Dimos Poulikakos

ETH Zurich, Switzerland

Email: dpoulikakos@ethz.ch

In this lecture I will show that the combination of fundamental knowledge from phase change thermofluidics, surface wetting and rational surface nanofabrication, can lead to significant advances in condensation heat transfer, with a broad range of industrial applications, ranging from power generation to the chemical, food, and pharmaceutical industries. I will present examples of so designed surface architectures and I will explain the physics behind their performance of significant heat transfer enhancement. I will then focus on rationally designed nanostructured superhydrophobic surfaces and discuss the derived peculiar behavior of condensate droplets on such surfaces. When condensate microdroplets coalesce, they can spontaneously propel themselves omnidirectionally on the surface independent of gravity and grow by feeding from droplets they sweep along the way. I will explain the physics behind this “droplet roaming” phenomenon of coalescing condensate microdroplets on such designed, solely nanostructured superhydrophobic surfaces, where the droplets are orders of magnitude larger than the underlaying surface nanotexture. This phenomenon is then utilized to prevent condensate flooding of the surface, remarkably improving heat transfer.

Boiling and Condensation Heat Transfer

Chairs: Paolo Di Marco, Vadim Nikolayev, Maria Rosaria Vetrano, Catherine Colin

Direct Numerical Simulation of Single Bubble Dynamics in Nucleate Pool Boiling with Micro-Region Modeling and Thermal Coupling to a Solid Wall

Linkai Wei, Guillaume Bois, Vadim Nikolayev

Commissariat a l'energie atomique et aux energies alternatives

Email: linkai.wei@cea.fr

In this study, we conduct two-dimensional, axisymmetric simulations to investigate the growth and departure of a single bubble originating from a nucleate seed, with a particular focus on the transient conjugate heat transfer between the fluid domain and the adjacent solid wall. A multiscale modeling approach is employed to account for microscopic effects near the liquid-vapor-solid triple contact line (TCL). At the macro (bubble size) scale, the capture of the interface dynamics and phase change are provided by the mixed volume-of-fluid/front-tracking algorithm and ghost-fluid method, using the open-source code TRUST/TrioCFD. The macro scale algorithm is coupled with a sub-grid micro-region model. It is driven by the wall superheat at the TCL as input, and predicts the macroscopic apparent angle and heat fluxes in the TCL region at both liquid-vapor and solid-liquid interfaces. To validate our modeling approach, we first perform a mesh sensitivity analysis, followed by quantitative comparisons of our results with experimental data. Notably, we observed a significant spatial variation in temperature near the nucleation site during the expansion and contraction of the bubble base. This variation greatly affects the thermal boundary layers in the fluid and solid domains. By comparing the results of simulations carried out earlier with a fixed wall temperature, our results highlight the impact of non-uniform distributed wall temperature on the bubble growth, demonstrating the need to resolve the conjugate heat transfer between the solid wall and the fluid domain in the cases considered. The impact of micro-region on the bubble growth is also analyzed.

Effect of Artificial Cavity on the Microlayer and Contact Line Dynamics at Bubble Growth in Nucleate Boiling

Cassiano Tecchio, Iacopo Regoli, Benjamin Cariteau, Gilbert Alczer, Pere Roca I Cabarrocas, Pavel Bulkin, Jérôme Charliac, Simon Vassant, Vadim Nikolayev

Commissariat a l'energie atomique et aux energies alternatives
Université Paris-Saclay, CEA Paris-Saclay
Institut Polytechnique de Paris
Centre National de la Recherche Scientifique

Email: cassiano.tecchio@cea.fr

We report an experimental study on the near-wall phenomena during the growth of a single bubble in saturated pool boiling of water at atmospheric pressure. Our focus is on the dynamics of triple contact line and liquid microlayer that can form between the heater and the liquid-vapor interface of the bubble. The microlayer thickness, the wall temperature distribution and the bubble shape are measured simultaneously and synchronously at 4000 fps by white light interferometry, infrared thermography and side wise shadowgraphy, respectively. In order to address the effect of cavities we perform two experiments using different heaters. In the first experiment, the bubble grows on a smooth surface of nanometric roughness whereas in the second the bubble growth occurs on a cylindrical artificial cavity of 25μm diameter and 50μm depth fabricated by focused ion beam. With the artificial cavity, the results show that the required wall superheating to trigger the bubble growth is decreased by a factor three. The radii of macroscopic bubble shape, microlayer and dry spot were reduced by half. The macroscale bubble dynamics is also slowed down. The initial microlayer thickness is thinner and detectable in a large portion of its extent. Based on the absence of interference fringe near the contact line (due to high interfacial slopes) and on recent numerical simulations we understand the microlayer profile consisting of two regions: a growing over time dewetting ridge near the contact line followed by a flatter and wider region that thins over time. The microlayer can be seen as a film deposited by the receding meniscus and its profile is thus controlled by the viscous and surface tension effects; its thinning over time is due to evaporation only. The ridge is a result of liquid accumulation due to contact line receding and strong viscous shear in the film.

The Influence of Liquid Height to Bubble Size Ratio on the Boiling Crisis in Pool Boiling

Rodrigo Cavalcanti Alvarez, Matteo Bucci

Massachusetts Institute of Technology

Email: rcaval@mit.edu

We conducted experiments to analyze water boiling under 10K subcooling and atmospheric conditions in a horizontal pool boiling test setup, focusing on the effect of liquid height above the heating surface. Our main interest was the transition in boiling crisis mechanisms for liquid heights comparable to bubble size. For pool heights above 4 mm, we observed a relatively constant critical heat flux (CHF) of approximately 1.4MW/m^2. Conversely, CHF significantly dropped to around 0.19 MW/m^2 for liquid heights below 2.5 mm. For depths between 2.5-4 mm, the CHF oscillated within these limits. Our pool boiling facility featured a stainless-steel chamber filled with deionized water heated by external circulating oil. A ceramic cartridge supported a transparent heater at the cell's center-bottom, which utilized an indium tin oxide layer on a sapphire substrate for heat dissipation. We monitored bubbles' footprint dynamics during boiling using infrared and high-speed video cameras. Preliminary analysis of high-speed data revealed different hydrodynamic mechanisms corresponding to the obtained CHFs. For heights above 4.0mm, the formation of an irreversible dry patch is triggered by bubble interactions. Conversely, at heights below 2.5mm, one of the initial nucleating bubbles, shortly after the onset of boiling, failed to detach from the surface. Instead, it continued to grow in size and eventually spread over the entire surface, prematurely reaching the CHF. We attributed this behavior to a lack of buoyancy forces to detach the bubble from the surface as it grew above the liquid level. Pool heights between 2.5–4.0 mm showed both behaviors. Further investigations will explore the transition in boiling mechanisms shifting from buoyancy- to surface-tension-dominated processes by varying bubble dimensions through the introduction of surfactants to the water. Ultimately, we aim to analyze fundamental boiling parameters and heat flux to characterize CHF variations concerning the bubble-to-liquid height ratio.

Confined Boiling on an Immersed Heated Corner: Influence of One Wall on the Other

Nicolas Baudin, Antonio Della Volpe, Stéphane Roux, Robert Yu, Jean-Michel Fiard, Jerome Bellettre

Nantes Universite, France

Renault Group, France

Email: nicolas.baudin@univ-nantes.fr

An experimental test section with horizontal and vertical heated plates forming an inverted L-shape is filled with 3M Novec7000 dielectric fluid at atmospheric pressure. Both plates of $64x40mm^2$ dimensions are heated with wall heat fluxes ranging from 0 to 10 W/cm^2. There is natural convection for low heat fluxes, medium heat fluxes trigger boiling and for higher power, the critical heat flux is reached. The wall temperature and heat flux are measured from several thermocouples in steady state. The bubbly flow on the vertical surface is observed with a high-speed camera through a window. The confinement between the vertical heated surface and the wall is varied from 0.5 to 15 mm. In the range of this study, the heat transfer coefficient in natural convection and nucleate boiling regimes increases when the spacing decreases. However, since the critical heat flux decreases when it is more confined, the film boiling regime is reached faster. From the visualizations and the thermal measurements we try to model these different behaviors. Depending on which plate is switched on or off we observed different behaviors. The confinement of the vertical plate does not have any influence on the horizontal one since the plate width is much bigger than the gap change. Also, whether the horizontal plate is heated or not does not influence the heat transfer on the vertical plate in this study. However, there is an influence of the vertical boiling on the horizontal one, stronger when there is a confinement than in free configuration.

Subgrid Moving Contact Line Model for Direct Numerical Simulation of Bubble Dynamics in Pool Boiling of Pure Fluids

Vadim Nikolayev, Linkai Wei, Guillaume Bois

Commissariat a l'energie atomique et aux energies alternatives

Email: vadim.nikolayev@cea.fr

The model is based on the hydrodynamic multiscale theory of contact line vicinity under phase change. Essentially, the main control parameter of the dynamic model is the static apparent contact angle (Voinov angle) that forms due to evaporation only. The dynamic apparent contact angle qapp is obtained with the Cox-Voinov formula. The calculation of the Voinov angle is performed with the generalized lubrication approximation and is tabulated as a function of the local superheating. The model is suitable for the partial wetting case and includes several nanoscale effects like those of Kelvin and Marangoni, hydrodynamic slip length and interfacial kinetic resistance. For conventional fluids, the microscopic length of the Cox-Voinov formula (Voinov length) is controlled mainly by the hydrodynamic slip. The integral heat flux Q passing through the contact line vicinity is almost independent of the nanoscale phenomena, with the exception of the interfacial kinetic resistance. Both the dynamic apparent contact angle and the integral heat flux are the main output parameters of the subgrid model, while the local superheating and the microscopic contact angle are the main input parameters. The model is suitable for the grid sizes $\gtrsim 1$ μm.

Wettability Patterning of Titanium Surfaces Through Pulsed Laser Melting for Enhanced Condensation Heat Transfer

Avnish Chokshi, Rahul N, Rajdeep Singh Devra, Soumyadip Sett, Madhu Vadali

Indian Institute of Technology Gandhinagar

Email: chokshiavnish@iitgn.ac.in

Laser surface modification techniques have been in the spotlight for the last few decades due to its widespread capability in tailoring topography and altering surface wettability. Titanium-based alloys, especially Ti grade 5 (Ti-6Al-4V) have remarkable mechanical properties - strong durability and high density to weight ratio, corrosion resistance, and biocompatibility. Ablation based surface modifications on Ti grade 5 surfaces using femtosecond lasers are prevalent. However, the high capital cost of femtosecond lasers coupled with the high temperature requirement for Ti ablation makes the process energy intensive. In this work, we utilize microsecond laser as an alternative to texture Ti-based surfaces. Our work focuses on using microsecond pulsed laser melting at various scanning speeds to develop a wide range of scalable surface structures on Ti grade 5 substrate. The increased surface roughness due to laser-based melting alters the surface wettability, making the Ti-surfaces superhydrophilic. We further render these surfaces hydrophobic through atmospheric pressure chemical vapor deposition of silane. Water vapor condensation experiments were conducted on the engineered surfaces with dropwise condensation being realized on the hydrophobic, laser-melted Ti surfaces. For marine applications, such Ti surfaces which are known to resist microbial growth can be promising for enhanced heat transfer applications. Our work here demonstrates a facile method to develop scalable, microstructured Ti surfaces with controlled wettability and promoting dropwise condensation, thereby enhancing heat transfer performance.

Cryogenic Quenching Process Enhancement Through Coating and Microstructure Optimization

Marco Graffiedi, Francis Dent, Sepideh Khodaparast, Sepideh Khodaparast, Matteo Bucci

Massachusetts Institute of Technology
University of Leeds

Email: mgraff@mit.edu

In this work, I explore the impact of various coatings and microstructures on heat transfer during the cryogenic quenching process. An easily reproducible quenching test is presented as a benchmark for testing different technological solutions. The study involves two different flat polymeric coatings as well as three porous microstructures. The results show that pairing a low-conductive coating with an appropriate porous surface microstructure on top of a stainless-steel plate can reduce the chill down time, accelerating the transition from room temperature to liquid nitrogen temperature, by a factor of five. Focusing on explaining the different regimes of the quenching process, high speed video recordings have been used to analyze the quenching process with different coatings and microstructures showing how the suppression of the film boiling regime is the key to enhancing the quenching process.

Condensation of Water on PDMS-Coated Copper Surfaces for Fresh Water Harvesting

Till Pfeiffer, Tatiana Gambaryan Roisman, Peter Stephan, Hans-Jurgen Butt, Michael Kappl

Technische Universitat Darmstadt
Max Planck Inst For Polymer Research
Max-Planck-Gesellschaft

Email: pfeiffer@ttd.tu-darmstadt.de

To solve fresh water scarcity for approximately 2.2 billion people on earth, condensation shows great potential by accessing earth's atmospheric water. The efficiency of dropwise condensation can exceed the efficiency of filmwise condensation by up to a 5-7-fold factor. The dropwise condensation is promoted on hydrophobic surfaces with a low contact angle hysteresis. Metals, which are commonly used in condensation applications, are characterized by a high surface energy and are therefore hydrophilic. One of the ways to enable the dropwise condensation on metallic substrates is to modify their surfaces to achieve the desired wetting behavior. In this work, we study the condensation of water vapor from a humid air on vertical copper substrates which are modified by applying an ultrathin 5-8 nm thick PDMS-coating. The modified surfaces show a hydrophobic behavior with a low contact angle hysteresis. For comparison, the condensation experiments are also performed with uncoated copper surfaces. We investigate the condensate mass flow rate and the surface wetting behavior as functions of the air temperature, relative humidity and surface coating. While filmwise condensation is the dominant condensate wetting regime on uncoated copper surfaces, continuous dropwise condensation is observed on PDMS-coated copper surfaces during the single day measurement campaign. The more efficient dropwise condensation mode leads to an increased condensate mass flow rate. With increasing far field relative humidity from 40% to 95%, the condensation efficiency enhancement factor compared to an uncoated surface increases from 1.3 to 1.5. With an increasing far field temperature from 20°C to 40°C, the condensation efficiency enhancement factor compared to an uncoated surface decreases from 1.4 to 1.3. The results prove that the of PDMS-coated surfaces are very promising for fresh water harvesting, since an enhancement of condensate mass flow rate is observed for all realistic environmental conditions, comprising temperature and relative humidity.

Effect of Composition on Bubble Behaviour During Boiling of Mixtures on Superhydrophobic Surfaces with Artificial Cavities

Ningxi Zhang, Daniel Orejon, Khellil Sefiane

The University of Edinburgh

Email: s2137751@ed.ac.uk

Bubble behaviour during the boiling process is a critical phenomenon that significantly impacts heat transfer processes involving phase-change. Observing the bubble dynamics is a crucial method for understanding the mechanism of heat transfer during boiling. This study experimentally investigates bubble behaviour from an isolated artificial cavity, utilising a binary mixture of Novec649 and Novec7000 with mole fractions between 0.05 – 99.5 at saturation pressure. The superhydrophobic coating, Glaco, was used as a boiling surface to investigate the bubble dynamics while a temperature sensor under artificial cavity was implemented to measure and calculate its superheat. Compared to previous research works, our results show an earlier onset of bubble departure as well as clearer trends in bubble departure diameter in the presence of the binary mixture. The bubble departure diameter is at its minimum at of 0.512 mm with the shortest growth rate, and the time constant is 0.151 when the mole fraction is approximately 0.50. As the mole fraction deviates from 0.5 the bubble departure diameter and growth time increase. On the other hand, the maximum bubble departure size of 0.634 mm is observed for pure Novec7000 fluid, where higher surface tension force plays a pivotal role in the force balance acting on the bubble. It should be noted that the concentration of the mixture evolves during bubble growth which in turn influences the buoyancy and surface tension forces determining departure diameters. The magnitudes of other forces remain similar across different concentrations. In various force models, the results highlight the significance of micro-layer evaporation when both components are present in equal concentrations. Conversely, thermal conduction becomes more critical in the process of bubble growth in other scenarios. Therefore, insights from this work provide valuable information for improving our understanding of heat transfer mechanisms during boiling of binary mixtures and hydrophobic surfaces.

Flow Boiling in Microchannels Coupled with Micro-Nano-Modified Surfaces

Ana Moita, Mariana Perez, Pedro Pontes

Universidade de Lisboa
Instituto Superior Técnico, University of Lisbon

Email: anamoita@tecnico.ulisboa.pt

High Concentration Photovoltaic (HCPV) Panels are one of the most efficient photovoltaic technologies in converting solar energy to electricity, reaching standard efficiencies of up to 40%. However, higher radiant heat fluxes also cause thermal management issues, which penalize conversion efficiency. To cope with this problem, an appropriate cooling solution is needed. In this context, a project is under progress, aimed at the development of a heat sink based on microchannels to cool small PV concentrated panels. Working conditions are most favorable when using two-phase flows, but flow instabilities are still difficult to control. Hence, the present work addresses the experimental study the effect of microchannel cooling using two-phase flow conditions. These microchannels are combined with micro-modified surfaces, to enhance heat transfer and flow stability. The surfaces made from AISI 304 stainless steel, are modified using microcavities strategically inserted to control and promote nucleation in specific regions. Flow and heat transfer are characterized combining infrared time resolved thermography and high-speed images, while pressure drop on the channel was also measured. Two different power values were supplied, and two different flow rates were considered. Pressure drop results were inconclusive regarding the effect of surface micro-structuring. Nevertheless, the analysis proposed was able to identify where nucleation sites were formed and study the fluid dynamics and heat transfer processes. Slug flow interfacial heat transfer could be observed and accurately described in the thermal and heat transfer maps.

Flow Boiling Heat Transfer in a Plate Heat Exchanger with Mixed Chevron Angle Plates

Chien-Yuh Yang, Mu, Ting Hsieh

National Central University

Email: cyyang@ncu.edu.tw

Plate heat exchanger (PHE) has been widely used in refrigeration and air-conditioning, chemical processes and many other industrial applications attributed to its high heat transfer performance and high compactness. While applying the plate heat exchanger as an evaporator in an air-condition system, refrigerant flows into the heat exchanger at two-phase condition. Since the length of flow path in the channels near the inlet port is shorter than that in the channels farther away from the inlet port, most of the liquid will flow through the channels near the inlet port. This causes a liquid deficient part of the plates away from the inlet port, results in a superheat region and degrades the entire heat exchanger performance. This study designed a 120-plates PHE with combination of 60 plates with 65° chevron angle and 60 plates with 35° chevron angle. The plates with 65° chevron angle were assembled in the half part of the PHE which near the inlet port to provide higher flow resistance in comparing to the other 35° plates. Since the flow resistance through the 65° plate is higher than that in 35° plates, it is aimed to cause a uniform liquid flow through the plates, reduce the liquid deficient region and therefore increase the heat transfer performance of the heat exchanger. Refrigerant R-410A flow boiling heat transfer performance in the mixed angle PHE and another PHE with 65° chevron angle was tested for comparison. The test results show that heat transfer coefficient in the mixed chevron angle PHE is 20% higher than that in the 65° chevron angle PHE with 22.9% lower pressure drops. The mixed angle plates apparently reduced the liquid refrigerant flow deficient region and provided a better heat transfer performance.

Vapor Bubble on a Single Nucleation Site: Temperature and Heat Flux Measurements

Himanshi Kharkwal, Mohammed Zamoun, Magali Barthes, François Lanzetta, Hervé Combeau, Lounès Tadrist

Universite de Technologie de Belfort-Montbeliard
Universite de Franche-Comte
Aix-Marseille Universite

Email: himanshi.kharkwal@femto-st.fr

This work is part of the ANR TraThI project which aimed at gaining a detailed understanding of heat exchange during convective and boiling phenomena. More specifically, this work focuses on a better understanding of thermal transfers occurring at wall-fluid interfaces in natural convection and nucleate boiling regimes in pool boiling experiments. To avoid parasitic effects, the vapor bubble is created at a single artificial nucleation site. The boiling cell is instrumented with sensors for temperature measurement, pressure control, and parietal heat flux measurement. The fluid tested is Fluorinert FC-72. The first boiling-meter consists of a heating resistance sandwiched between two commercial heat flux sensors (CAPTEC) instrumented with thermocouples. This boiling-meter enables the measurement of both temperature and heat flux, and can be rotated through 360°, enabling the influence of inclination to be studied. The boiling-meter is indented at its center to create a single vapor bubble. Heat flux measurements are obtained and studied for different fluid temperatures, pressure, and inclination of the nucleation surface. Moreover, a calibration test for the boiling-meter was conducted for symmetrical and non-symmetrical conditions, further, the variations were plotted. In parallel to this, we are currently developing a miniature normal heat flux sensor NHF (based on Fourier's law) thanks to microfabrication techniques. This sensor is composed of wafers separated by an insulating layer. On each face of this assembly, platinum resistors (RTD) are deposited in a spiral shape, with an average diameter varying between 1 and 2 mm. In the center of the RTD spiral, a cylindrical nucleation site is etched onto the silicon surface by DRIE. Heat flux as well as temperature measurements obtained with the first boiling-meter are given and discussed. Then in a second part, the design, and the microfabrication of the miniature NHF are given.

Convective Boiling Heat Transfer Enhancement via Femto-Laser Texturing

Maria Rosaria Vetrano, Frederik Mertens, Balasubramanian Nagarajan, Sylvie Castagne, Johan Steelant

Katholieke Universiteit Leuven
European Space Agency ESTEC

Email: rosaria.vetrano@kuleuven.be

Designing efficient cooling devices is crucial in a multitude of applications. Although cooling technologies have existed for decades, there is a lack of efficient heat dissipation methods, especially for space applications where thermal management devices must dissipate heat in reduced gravity and where convective boiling heat transfer is frequently employed. In this framework, we evaluate the heat transfer enhancement produced by different types of surface textures produced on 200 µm thick 316L SS foils using a femtosecond laser. The investigated textures include microscale grooves and conical holes with straight and tilted (45°) geometry. In addition, we also evaluate the effect of laser-induced periodic surface structures (LIPSS), which are sub-micron scale features fabricated by fs laser processing. The fluid used is the PP1, widely used to replace 3M™ FC-72 in heat transfer applications. We investigate the impact of the textures on the heat transfer coefficient (HTC) and Critical Heat Flux (CHF) as a function of the gravity direction using a set-up in which a channel can be oriented at different angles. Our experimental set-up is a typical closed fluidic loop in which a rotatable square 5mm x 5mm test section, with one heated side and embedded thermocouples, is mounted. The fluid can be pre-heated, and the Reynolds number can be varied to attain fully turbulent conditions. A high-speed camera visualizes the flow boiling, allowing us to determine the impact of the textures on several parameters, such as bubble detachment diameter and detachment frequency. This allows us to assess the nucleation site density and link it to the geometry of the texture and the boiling heat transfer regime. The results have shown a general improvement of HTC using textures up to 50% in the early nucleate boiling regime for the 45° inclined holes with respect to plain surfaces for the horizontal channel.

Comparison of Temperature-Sensitive Coatings Immobilized in Polymeric Matrices for Fast/High-Resolution Heat Transfer Transient Characterization

Maria Rosaria Vetrano, Matevz Vodopivec, Donato Fontanarosa, Erin Koos, Yanshen Zhu

Katholieke Universiteit Leuven

Email: rosaria.vetrano@kuleuven.be

Efforts in developing new measurement techniques are required to capture the surface temperature dynamics with submicron and submillisecond resolution. In this regard, two types of temperature measurement can be used: contact measurement and optical measurement. Even though it can reach a very high acquisition speed, the former is local and intrusive, making it limited to 2D/3D maps of discretized measures where the sensor size is a severe limiting factor for the maximum spatial resolution, typically being in the order of sub-millimeter to millimeter. The latter is extensive and non-intrusive, making it ideal and preferred with respect to the former. It is typically based on high-speed IR imaging, thanks to the recent introduction of advanced and expensive long-distance IR cameras that detect medium and long wavelengths with high spatial and temporal resolution. A promising alternative are temperature-sensitive paints (TSPs) consisting of a temperature-sensitive fluorescent dye immobilized into a polymeric binder. Excitation and emission wavelengths typically range in the UV and VIS spectra, respectively, making TSP-based thermography exploitable by using general-purpose (high-speed) cameras. This work provides an experimental assessment of static and dynamic performances of TSPs using acrylic binders, 3 fluorescent dyes (Ru(phen)3Cl2, PtTFPP, EuTTA), and 2 halide perovskite quantum dots (CsPbBr3 and CsPbBr1.2I1.8). In combination with a UV lamp and a high-speed camera, TSPs are applied to one side of an Inconel 600 thin foil with 25 µm-thickness, which is electrically fed to generate Joule heating and control the temperature of the foil. A high-emissivity (> 97.5%) black paint is applied on the opposite side of the metallic foil, allowing for synchronized and simultaneous high-speed IR used for both TSP-based thermography calibration and validation. The dynamic response is assessed by imposing a power step on the metal foil. The most relevant insights resulting from their comparison are outlined and discussed.

Measuring Liquid Film Thickness During Downflow Condensation in a Minichannel

Arianna Berto, Marco Azzolin, Pascal Lavieille, Stefano Bortolin, Marc Miscevic, Davide Del Col

Universita degli Studi di Padova
Université Paul Sabatier (Toulouse III)

Email: arianna.berto@unipd.it

The availability of experimental liquid film thickness data during in-tube condensation heat transfer is fundamental for the development of reliable correlations for the design of condensers. During annular flow condensation, the heat transfer is directly associated to the liquid film thickness, as the main thermal resistance is due to thermal conduction through the liquid film. Therefore, understanding the liquid film characteristics is necessary for modelling two-phase annular flows. Although several prediction methods for liquid film thickness are available in the literature, they are mainly related to air–water adiabatic experiments. More liquid film thickness data are required for a better understanding of the condensation phenomena and for widening the range of experimental conditions (i.e. number of fluids, mass fluxes, …) to assess the validity of present models or develop new ones. In the present work, the liquid film thickness during vertical downflow condensation has been measured inside a 3.38 mm inner diameter channel. Condensation tests have been performed with refrigerants R245fa, R134a and HFE-7000 at mass flux from 20 to 150 kg m^{-2} s^{-1}. Two complementary non-invasive optical methods were used for measuring instantaneous liquid film thickness data: shadowgraph technique (applied to a sequence of flow pattern images recorded by a high speed camera) and chromatic confocal imaging. The existence of a dependence of the liquid film thickness on dimensionless numbers has been addressed in the present work for capturing the underlined physics governing the vapor–liquid interactions inside the tube. The accuracy of several well-known models for the prediction of the liquid film thickness has been assessed with respect to the experimental data of the tested fluids.

Development of Physically Based Modelling of Bubble Behaviour for Subcooled Boiling Applications

Jakub Cranmer, Giovanni Giustini, Alex Skillen, Ryan Tunstall

The University of Manchester
Rolls-Royce

Email: jakub.cranmer@postgrad.manchester.ac.uk

Nucleate boiling is of critical importance for the nuclear energy sector, where it can provide a highly efficient heat transfer mechanism in water-cooled reactors. Due to the multi-scale and multi-physics nature of the phenomenon in subcooled flow conditions, a comprehensive understanding remains elusive. This work is a step toward the development of a CFD model, for the investigation of heat transfer and hydrodynamics of flow boiling in the presence of surface defects. The volume-of-fluid technique is applied to directly analyse individual bubble behaviour, and a mechanistic, physically derived model is used to simulate the heat and mass transfer due to phase change. With such a model, micro-scale analyses of the impact of defects can be performed. Using data-driven techniques, data so generated can inform improved closure models for component-scale simulations, to assess the effect of surface finish on heat transfer in fuel assemblies. In this paper, the model is verified against analytical solutions for three classical evaporation problems, demonstrating excellent agreement for interface position against time. It has been shown that the methodology can account for movement of the interface purely due to phase change, as well as due to the generation of vapour volume. To demonstrate applicability to subcooled flows, verification is performed against the analytical solution for a planar condensation problem. Additionally, a case is selected from the existing literature, where the heat-controlled collapse of a spherical vapour bubble in subcooled water is studied by analysis and experimental observation, presenting results for bubble radius against time. Verification against the analytical model shows excellent agreement, therefore demonstrating the capability of the numerical model to capture not only evaporation, but also condensation phase change phenomena. Such verification is a step toward establishing the reliability of the methodology, and therefore toward its application in the prediction of flows of practical interest.

Vapor Bubble Growth and Detachment in a Shear Flow in Microgravity Conditions

Catherine Colin, Julien Sebilleau, Mbaye Modou, Md. Qaisar Raza

Centre National de la Recherche Scientifique

CNRS

Government of India

Email: colin@imft.fr

Boiling is of significant importance to many technical processes and applications on earth and in space. It is a complex process since it involves various physical parameters which govern the heat and mass transfer. Despite the many correlations that exist, the prediction of boiling heat transfer remains difficult. For better modeling, it is therefore important to improve our understanding of local physical phenomena, such bubble growth and detachment of isolated bubbles. The continuous detachment of bubble due to buoyancy during boiling in terrestrial gravity condition further complicates the process. Microgravity experiments are therefore relevant for observing larger bubbles on longer time scales. It is the objective of the Multiscale Boiling experiment RUBI designed for more than a decade by various European teams, under the framework of ESA (European Space Agency) Project BOILING. RUBI was operated on the International Space Station from 2019 to 2021. Experiments have been performed in pool boiling and in shear flow, both with and without the presence of electric field. The fluid used is a refrigerant N-perfluorohexane. Bubbles are nucleated on an artificial cavity located on a surface heated with heat flux from 0.5 to 1.5W/cm^2. Different liquid subcooling up to 10°C and flow rates are investigated. High-speed Black and White camera records the images of the bubble growth and departure. The temperature field at the surface of the heated wall is measured by an infrared camera. We report here some analysis of the bubble growth rate and departure in a shear flow. The bubble diameter evolves as $t_{0.5}$, which is characteristic of a growth controlled by thermal diffusion. The bubble departure diameter by sliding along the wall is found to be well predicted by a balance between the drag force exerted by the flow and the capillary force acting at the contact line.

Hydrodynamic Characteristics of the Microlayer Under Boiling Vapour Bubbles

Xiaolong Zhang, Ismail El Mellas, Mirco Magnini

University of Nottingham

Email: xiaolong.zhang@nottingham.ac.uk

This study investigates the hydrodynamic behaviour of the microlayer, a thin liquid film of a few micrometres thick that forms beneath vapour bubbles during boiling under specific conditions. This microlayer serves as a rapid heat transfer medium through evaporation due to its low thermal resistance, making it a compelling subject for enhanced heat transfer research. To gain insights into this phenomenon, we employ the geometric Volume-Of-Fluid (VOF) method for numerical simulations, allowing us to accurately replicate the growth of nearly hemispherical bubbles on solid surfaces. The computational approach offers a level of detail and resolution that can compensate for certain aspects of microlayer behaviour absent in experimental methods. Our simulations reveal the complete spatial distribution of the microlayer and the profiles of the bubble interface near the meniscus front (the part of the interface near the solid). To validate our findings, we compare the results with experimental data and predictions from analytical models on the microlayer thickness. Additionally, we conduct simulations with various bubble growth laws to explore how the microlayer responds to different growth scenarios. These findings lay the foundation for incorporating the microlayer's contribution to overall boiling heat transfer in future models, potentially unlocking new avenues for enhanced heat transfer applications.

Drag Effect of Steam Flow on Droplet Removal During Dropwise Condensation at Different Surface Inclinations

Antonio Abbatecola, Marco Tancon, Stefano Bortolin, Davide Del Col

Universita degli Studi di Padova

Email: antonio.abbatecola@studenti.unipd.it

Since its first observation in the 30s, dropwise condensation (DWC) has generated significant interest among researchers. This is due to the suppression of the thermal resistance associated to conduction through the liquid film, characteristic of traditional filmwise condensation (FWC). DWC is a quasi-cyclic process characterized by the nucleation, growth, and removal of discrete liquid droplets on a subcooled surface. The removal of condensate is a critical aspect, usually achieved by exploiting the gravity force, the drag force of vapor or the surface wettability gradients. In most of the studies present in the literature, the droplets' removal is gravity-driven in quiescent vapor conditions. However, considering potential industrial applications, it becomes interesting to investigate DWC when the steam velocity varies. This paper presents an experimental study of the vapor drag action on condensate removal, with a focus on droplet's departing radius (rmax). Specifically, for the experimental campaign, vapor velocity was varied from 3 to 14 m s^{-1} considering three different surface inclinations: vertical, 45° inclined, and horizontal. The results showed that, as the velocity increases, the difference in departing radii among the three different configurations decreases and, consequently, the difference in heat transfer coefficients (HTCs) decreases too. In fact, at the highest vapor velocity (14 m s^{-1}), rmax was almost equal for all the inclinations and this implies similar heat transfer coefficients (~120 kW m^{-2} K^{-1}). Interesting to mention, on a horizontal surface at a vapor velocity of 3 m s^{-1}, despite the lack of gravity's contribution to droplet removal, there was no transition to filmwise condensation.

Analysis of Microscale Inner-Grooved Tubes During Condensation of Low-GWP R1234ze(E): Flow Regime Visualization and Enhancement Factors

Nima Irannezhad, Luisa Rossetto, Andrea Diani

University of Padua

Email: nima.irannezhad@studenti.unipd.it

In an attempt to attenuate the global warming contribution in the refrigeration industry, several regulations have been legislated. Decommissioning the currently implemented refrigerants with high global warming potential is one of the salient parts of such regulations. Hydrofluoroolefins (HFOs) could be construed as feasible candidates, among which R1234ze(E), with GWP≈1 and suitable for multiple applications, has been subjected to experimental investigations. Owing to its prevalence in many industrial applications, flow condensation and boiling inside tubes have received particular attention for such equipment. Since R1234ze(E) is classified as mildly flammable and the collective experimental conclusion regarding its two-phase heat transfer coefficient showed slightly lower values compared to that of R134a, the implementation of tubes with enhanced surfaces such as inner grooved tubes with small diameters is encouraged to augment heat transfer coefficient and to reduce the refrigerant charge, thus averting the flammability wariness. The current experimental investigation aims to determine the degree of augmentation of the thermal performance with the usage of inner grooved tubes. In this case, tubes with various geometrical properties are employed for flow condensation of R1234ze(E) at a saturation temperature of 30°C. Measuring the experimental heat transfer coefficient allows obtaining the enhancement factor of the enhanced tube which is defined as HTC of an inner grooved tube to that of a conventional smooth tube of the same diameter. Furthermore, since the efficacy of enhanced surfaces is directly linked to prevailing flow patterns, for different operating conditions (G=100 to G=400 m^{-2} s^{-1}, x=0.01 to 0.95) the flow pattern has been identified with the aid of a high-speed camera. Lastly, results of flow pattern visualizations and enhancement factors are provided on a non-dimensional graph of EF (enhancement factor) versus JG (non-dimensional gas velocity) that allows a comprehensive performance analysis of enhanced surfaces.

Effect of Hydrophobic Coating on Optimization of Dropwise Condensation of Pure Vapor Over Hybrid Surfaces

Nicola Suzzi, Giulio Croce

University of Udine

Email: nicola.suzzi@uniud.it

Dropwise condensation of pure vapor on hybrid surfaces, characterized by alternate hydrophobic-hydrophilic regions, is numerically investigated via phenomenological, Lagrangian modelization of dropwise condensation on hydrophobic regions, while revised film theory is used to predict heat transfer over hydrophilic regions. The effect of hydrophobic coating thermal resistance on the heat transfer performance is analyzed and the optimization of hybrid surface geometry is conducted for different coating characteristics, using a previously developed and validated in-house code. Numerical results are compared with literature, statistical based models, showing that improvement of such simplified modelization is required to accurately predict heat transfer performance of hybrid surfaces.

Spray Cooling of Micropillared Steel Plates: Two-Stage Quenching Phenomenon

Yutaku Kita, Taihei Matsunaga, Takaaki Ariyoshi, Hiroyuki Fukuda, Yasuyuki Takata, Masamichi Kohno

University of London
Kyushu University
JFE Steel Corporation

Email: yutaku.kita@kcl.ac.uk

Spray cooling is a crucial process of steel manufacturing, nuclear power plants and electronics thermal management. However, its precise control is challenging due to the various boiling regimes involved at different temperature levels. Transition from film boiling to nucleate boiling, namely, the onset of quench is our particular interest as it remarks the thermal history of the hot object. Based on our previous findings that quench occurs at a specific temperature defined at the droplet-surface interface we demonstrate that micropillars on steel plates can increase the quench temperature T_q. We performed water spray cooling tests with stainless-steel samples, on which various configurations of micropillars were fabricated using laser texturing. Interestingly, we observed two stages of quench for a specific sample: a drastic increase in the cooling rate at ca. 450 °C, followed by another increased at ca. 250 °C. The first cooling rate increase is attributed to the initiation of stable liquid-solid contact at the pillar tops. Eventually, water wets the entire pillars and the bottom surface, further increasing the cooling rate. Samples with higher pillars showed no film boiling regime in the experiment, implying higher quenching temperature (T_q > 600 °C). We attribute the increased apparent T_q to the narrow heat conduction path within the micropillar, resulting in a significant temperature drop at the tip which enhances liquid-solid contact.

A Hybrid Atomistic-Continuum Framework for Multiscale Simulations of Boiling

Mirco Magnini, Edward Smith, Gabriele Gennari, Gavin Pringle

University of Nottingham
Brunel University Uxbridge
Otto-von-Guericke Universität Magdeburg
The University of Edinburgh

Email: mirco.magnini@nottingham.ac.uk

Boiling is a striking example of a multiscale physics process, where the nucleation of vapour bubbles occurs due to molecular-scale interactions between the fluid and a heated wall, but it also depends on the larger-scale hydrodynamics and thermal boundary layers determined by the outer system boundary conditions. Unravelling the essential nature of boiling inception and bubble growth from the nanoscale is still a prohibitive task for experiments owing to the small spatiotemporal scales of the process. Likewise, modelling boiling from the nanometre/nanosecond up to the millimetre/millisecond scales at which bubble departure occurs is not possible via state-of-the-art simulation methods. This is due to the fact that Molecular Dynamics simulations, which can capture nucleation from first principles, are limited to scales on the order of nanometres due to excessive computational cost. On the other hand, traditional continuum-scale simulations based on the solution of the Navier-Stokes equations on a computational mesh cannot capture nucleation. Here, we present a novel multiscale simulation method which merges molecular dynamics (MD) and continuum-scale (CS) descriptions into a single modelling framework, where MD resolves the near-wall region where molecular interactions are important, and a computational fluid dynamics solver resolves the bulk flow; a one-way coupling is realised, with the MD providing the boundary conditions for the CS solver. We model the progressive heating of a Lennard-Jones fluid via contact with a solid wall until a vapour bubble nucleates in the MD region of the domain and grows by entering in the CS domain. Our results show that an incompressible CS flow model based on the Volume Of Fluid method with interphase mass transfer calculated via the Hertz-Knudsen-Schrage equation is sufficient to obtain seamless coupling of phase fraction, velocity and temperature fields, with the hybrid MD-CS framework yielding bubble dynamics closely matching those of MD alone.

The Effective Use of Focused Shadowgraphy for Single Bubble Nucleate Pool Boiling Investigations

Marilize Everts, Matthias Welzl, Dieter Brüggemann

University of London
University of Bayreuth

Email: marilizeeverts@gmail.com

Nucleate pool boiling is known for its high heat transfer coefficients and is widely implemented from macroscales, such as boilers in power plants, to microscale, such as the thermal management of electronic equipment. Although it has been widely implemented, the prediction of nucleate pool boiling mechanisms remains complex and single bubble pool boiling heat transfer analysis is helpful to simplify the problem. Single bubble dynamics have been investigated for almost a century using different experimental and numerical approaches. Owing to the interesting, yet complex, bubble behaviour during nucleate pool boiling, publications tend to give a quick overview of the experimental setup before focusing on the bubble dynamics. Small, but critical information on the experimental setup is often omitted, which makes it challenging or even impossible to reproduce or compare experimental data. Therefore, the purpose of this study is to provide a systematic approach to using focused shadowgraphy for the investigation of single bubble dynamics using R245fa. A pool boiling experimental setup has been built and equipped with temperature, pressure and heat flux sensors, as well as a high-speed camera and light source to investigate bubble dynamics. The nucleate pool boiling takes place at a single cavity on an electrically heated copper block mounted at the bottom of a stainless steel chamber. The influence of the heat transfer surface area, cavity geometry, light source, parallelised light, as well as diffuser films were investigated to provide a systematic approach to the use of focused shadowgraphy for the investigation of single bubble dynamics during nucleate pool boiling.

Investigating the Effect of Differing Triangular Topologies and Roughnesses on the Bubble Dynamics of R32 During Nucleate Boiling

Marilize Everts, Wilhelm J. Van Den Bergh, Panagiotis Theodorakis

University of London
University of Pretoria

Email: marilizeeverts@gmail.com

There is an urgent need for low cost, easy to manufacture surface engineered heat sinks for use with immersion based two-phase cooling of electronics. Surface enhancement, one way of improving nucleate boiling performance, can take a range of forms. This ranges from porous coatings, adjusting the wettability of the surface through various methods, and fabricating nano- and microscale structures. The introduction of microscale structures on the surface is one of the simplest and cheapest methods of enhancing the surface. Ease of manufacture with the most basic production methods and tools play a large role in deciding on the cost effectiveness of a surface. Bubble dynamics have a definite impact on the nucleate boiling process, so it is logical to investigate the effect different achievable topologies have on bubble growth and departure rates. With the advent of modern computational fluid dynamics software, it is possible to rapidly investigate and pursue or discard novel surfaces. The focus of this study was to investigate the effect differing average surface roughnesses and patterns had on bubble dynamics. Triangular machining profiles, easily achievable by using a lathe, were investigated, using a full axi-symmetric surface coupled with conjugate heating of R32. The triangular profile was kept at a nominal average surface roughness of 150 micron, with bubble growth and departure time for different tool angles being investigated. The results confirmed that shallow angles of indentation slow down the bubble departure compared to a smooth surface, but had no real effect on the departure diameter. In contrast, beyond a critical angle, the bubble departure diameter and time was much less than that of the smooth surface. Surface tension, and the physical limitation of bubble diameter were the major cause of this behaviour.

Enhanced Pool Boiling Heat Transfer with Porous Ti-6Al-4V-Coatings Produced by Cold Spray Metal Additive Manufacturing

Alekos Ioannis Garivalis, Evgeny Shatskiy, Yan Chen, Anthony Robinson, Paolo Di Marco, Rocco Lupoi

The University of Dublin Trinity College
University of Pisa

Email: alekos.garivalis@unipi.it

In advancing industrial heat transfer mechanisms, surface coatings offer significant potential. This research elucidates the efficacy of the metal additive manufacturing Cold Spray deposition technique for producing enhanced boiling surfaces, specifically focusing on TiH coatings on aluminium substrates. This offers a rapid and low-cost fabrication method for producing enhanced and light weight enhanced boiling surfaces. The Cold Spray method is typically used to create dense metal deposits. Here, the process has been specially tuned to create highly inhomogeneous honeycomb-type porous TiH coatings. Critical Cold Spray deposition parameters, such as particle velocity, preheat temperature, and deposition rate have been identified to create repeatable porous coatings, with thicknesses of up to 3.0 mm achievable. Following deposition, samples were subjected to systematic boiling heat transfer tests in a purpose-built pool boiling apparatus. Boiling curves were generated for the augmented Cold Spray surfaces as well as a bare surface, with the later acting as a baseline to which enhancement levels were assessed. Initial data analysis suggests a notable increase in boiling heat transfer coefficient and Critical Heat Flux with the TiH-coated substrates. This enhancement is potentially attributed to increased surface area, increased nucleation site density, capillary wicking, and mitigation of lateral bubble coalescence. In summary, the novel TiH surface structures developed using the Cold Spray deposition technique exhibits high potential for industries necessitating superior boiling heat transfer performance. Importantly, the manufacturing process is industrially scalable, offering capacity to rapidly coat large areas at low cost compared with subtractive manufacturing other metal additive manufacturing methods.

Experimental Investigation of Condensation Heat Transfer on Grooved Aluminum Surfaces

Evgeny Shatskiy, Parth S Kumavat, Seamus O'Shaughnessy

The University of Dublin Trinity College

Email: shatskie@tcd.ie

Condensation heat transfer plays a pivotal role in enhancing efficiency in diverse industrial applications, including thermal power generation, nuclear reactors, and heating, ventilation, and air conditioning (HVAC) systems. This study delves into the experimental analysis of condensation heat transfer on both unmodified and structurally modified vertical flat aluminum surfaces. A specialized experimental setup was utilized, consisting of a boiler, a condensation chamber, a drainage vessel, a vapor trap, and a vacuum pump, with distilled water employed as the working fluid. A 50 mm x 50 mm aluminum test surface was meticulously aligned with the condensation chamber's wall, facilitating precise measurements. The heat flux removal from this surface was quantified using the meter bar technique, complemented by a copper heat exchanger using water as a coolant on its reverse side. Initial experiments were conducted on a polished aluminum surface to establish a baseline. These results were corroborated against established heat transfer correlations for filmwise condensation under varying subcooling conditions. The study then progressed to analyzing surfaces modified with square-profile grooves and an inverted V-shaped longitudinal structure. A key finding was that the surface superheat is significantly affected by the orientation of the grooves in relation to the gravitational vector, enabling the identification of an optimal angle for the maximal heat transfer coefficient. A comprehensive evaluation of all test surfaces revealed a consistent improvement in the heat transfer coefficient by over 20% in comparison to the unmodified aluminum surface. This indicates a substantial potential for enhancing condensation heat transfer efficiency through strategic surface modifications, presenting valuable insights for application in industrial heat transfer systems.

Investigation of Dropwise Condensation of Water Through an Efficient Individual-Based Model

Marco Tancon, Antonio Abbatecola, Matteo Mirafiori, Stefano Bortolin, Davide Del Col

University of Padua

Email: marco.tancon@unipd.it

In recent years, researchers have directed their studies toward solutions aimed at enhancing the efficiency of heat exchangers and associated phase-change processes. In this context, dropwise condensation (DWC) has been identified among the most promising solutions to increase the condensation heat transfer coefficient (HTC). In fact, DWC provides heat transfer coefficients up to ten times higher than those achievable during filmwise condensation (FWC), resulting in both economic and energy benefits. The DWC phenomenon is usually modeled by combining the heat exchanged through a single droplet and the drop-size distribution. The latter can be divided into a distribution of large droplets N(r), determinable analytically by semi-empirical models, and a distribution of small droplets n(r), typically determined through statistical approaches called population-based models. Another possibility for the determination of the droplet-size density is to simulate the DWC process by an individual-based model (IBM). In this case, each drop is tracked throughout its entire life cycle (nucleation, growth, coalescence, and sliding), and the drop-size distribution is obtained without the need for any statistical assumptions. In this paper, a new IBM suitable for simulating dropwise condensation of water vapor is presented. The developed model proves to be more efficient than the models currently available in the literature, allowing for the simulation of more than 10 million droplets while keeping simulation times low thanks to the implementation of parallel computing. The predictions obtained from the model, in terms of drop-size distribution and condensation heat flux, were compared against both PBMs and experimental data. One of the most important outcomes is that, despite being more computationally intensive, IBMs are more accurate in approximating experimental data than PBMs. It is worth mentioning that, through the implementation of IBMs, it becomes feasible to investigate aspects that, due to technological limitations, cannot be observed experimentally.

Experimental Investigation of Surface Tension and Artificial Nucleation Site Geometry on Isolated Bubble Dynamics

Maria Rosaria Vetrano, Matevz Vodopivec, Mulugeta Gebrekiros Berhe, Mattia Bucci, Matevž Zupančič, Donato Fontanarosa, Matic Može, Sylvie Castagne, Iztok Golobič

Katholieke Universiteit Leuven
University of Ljubljana

Email: rosaria.vetrano@kuleuven.be

The recent advancements in micro-manufacturing technology have attracted new research focused on enhancing nucleate boiling (NB) by locally altering the wettability and or the thermal conductivity. However, a complete mechanistic understanding of the fast fluid-solid heat transfer dynamics occurring at micro- and submicrometric scales in the proximity of the solid wall is largely unexplored, resulting in a large discrepancy among researchers on the heat transfer performance of enhanced surfaces and the subsequent impossibility of exploiting the manufacturing potentials. Overcoming these limitations requires moving back to the understanding of nucleate boiling fundamentals. The heat transfer mechanisms in NB are multiple and interconnected: natural and micro convection, liquid microlayer evaporation, superheated thermal layer evaporation, transient heat conduction during the rewetting process, and forced convection induced by bubble agitation. Their dependency on the fluid and heat flux outlines the need to segregate the bulk and the interface-induced liquid accessibilities, i.e., the far-field and the near-surface hydrodynamics. Furthermore, the estimation of NB parameters (bubble growth time, bubble departure diameter and frequency, rising bubble velocity, wall superheat, nucleation temperature, local heat flux) can be explored by studying the isolated bubble generated by artificial nucleation sites (ANSs). The present work provides an experimental assessment of the impact of bulk liquid accessibility on the isolated bubble dynamics. In particular, non-ionic surfactant is dissolved into water at different concentrations, thus varying the surface tension to typical values of dielectric coolants without modification of other thermodynamic properties. The experimental setup consists of a pool boiling chamber equipped with a thin titanium foil as a heater. The diagnostics toolbox consists of high-speed visualizations and high-speed thermography for wall superheat and heat flux computations. ANSs are produced via femtosecond laser texturing to promote gas entrapment and bubble nucleation, and the most relevant insights resulting from the comparison are outlined and discussed.

Grooved Hybrid Copper Surfaces for Condensation Heat Transfer

Amit Goswami, Suresh C. Pillai, Gerard Mcgranaghan

Atlantic Technological university Sligo

Email: amit.goswami@research.atu.ie

Condensation is commonly utilized in numerous thermal management applications and has two modes termed filmwise and dropwise, the latter providing superior heat transfer performance. However, sustaining dropwise condensation mode is a challenge due to condensate flooding at high subcooling. To combat this, hybrid surfaces consisting of regions with different wettability have emerged as a promising solution. Of such designs, hybrid grooved surfaces consisting of grooves with varying wettability in the groove valleys and on ridge tops have shown significant potential for reducing flooding. To date, all such surfaces studied so far have either been based on silicon or in the case of metals, lack superhydrophobic regions. Superhydrophobic surfaces present more nucleation sites compared to hydrophobic surfaces which is favourable for improved heat transfer and can also benefit from droplet jumping motion. In this work, condensation experiments are performed on a range of grooved hybrid surfaces fabricated on a copper tube. The aim is to take advantage of anisotropic wetting due to the presence of grooves and condensate drainage from higher wettability regions to superhydrophobic low wettability regions. The experimental results quantify the heat transfer with consideration of the degree of subcooling. Additionally surface design parameters such as groove depth and region width for the two different wettability regions are considered in relation to the droplet dynamics and heat transfer.

Advancements in Laser-Based Spatiotemporal Measurements of Flow Boiling

Surya Narayan, Zengchao Chen, Aleksei S. Lobasov, Konstantin S. Pervunin, Matteo Bucci, Christos Markides

Imperial College London
Massachusetts Institute of Technology

Email: s.lakshmi-narayanan@imperial.ac.uk

Obtaining simultaneous, spatiotemporally resolved information on multiple physical quantities of interest in flow boiling is challenging for experimentalists. The recent progress in laser and imaging systems, and in computational processing capabilities, have made quantitative optical imaging (often employing laser illumination) highly adaptable, robust and reliable. The inherent advantages of these advanced methods are non-intrusiveness, high spatiotemporal resolution, whole-field measurement of scalar (e.g., temperature, concentration, phase) and/or vector (e.g., velocity) quantities, which makes it possible to overcome the limitations of conventional measurement techniques. In particular, optical diagnostic techniques including planar laser-induced fluorescence (PLIF) and particle image velocimetry (PIV) offer the possibility of the simultaneous measurement of resolved temperature and velocity maps in the liquid phase at boiling conditions. Here, we first review the applicability of PLIF in which the ratio between individual fluorescent emissions from the uniformly dispersed two or more dyes is used for thermographic imaging of liquid phase in the presence of moving vapour-liquid interfaces often found in boiling flows. The conventional implementation of PLIF necessitates uniformity in the concentration of dyes in the liquid phase. However, in the case of multiphase flows such as boiling, thermophoresis can lead to inhomogeneity in the concentration of dyes. To overcome this issue, a single-dye multispectral planar laser-induced fluorescence (SDMS-PLIF) probes the fluorescent emissions from a single dye (Nile red) across two or more selected band of wavelengths have been developed. The present work uses the experimental data from an UV-VIS spectrometer to identify the sensitivity bands for temperature measurements for a wide-range of concentration of dye from 0.3 mg/L to 30 mg/L. Following this, a demonstration of the SDMS-PLIF thermography is performed to study the hydrodynamic and thermal interactions between the vapour bubble and the heated substrate inside a miniaturised vertical square channel.

Bubble Departure and Sliding in High-Pressure Water Flow Boiling

Matthew Hughes, Artyom Kossolapov, Bren Phillips, Matteo Bucci

Massachusetts Institute of Technology

Email: mthughes@mit.edu

High resolution measurements of key parameters relevant to bubble departure processes in high-pressure water systems has been a major challenge over the past several decades due to the small size and fast motion of vapor bubbles, making conventional optical measurements unreliable. Addressing this difficulty is critical, as accurate bubble departure data and computationally inexpensive models are needed for future two-phase heat transfer models. To meet this need, we utilize an optical technique that combines phase-detection and shadowgraphy to measure the optical and physical bubble footprint with a high-speed camera, thereby enabling high spatial (6 µm) and temporal (33 µs) resolution measurements of bubble growth and departure. These optical measurements are made at pressures ranging from 10 – 40 bar, where the water is subcooled by ~10°C and flows at mass fluxes ranging from 500 – 2,000 kg m^{-2} s^{-1}. It is observed that bubbles are almost always perfectly spherical in shape and slide along the boiling surface immediately after nucleating. To predict sliding, we develop a mechanistic force balance model to derive an equation of motion for bubble sliding. Interestingly, we find that at high pressure conditions, the equation of motion can be significantly simplified, resulting in an analytical expression for the bubble motion, growth time and departure diameter, which reveals a non-dimensional number that is related to the slip velocity between the bubble and bulk liquid. Differences between the results from the analytical and numerical equations of motions are immeasurable across all operation conditions, and allow us to develop simple and computationally efficient equations and a departure criterion for bubble sliding. The applicability of the model and departure criterion across a broader range of high-pressure conditions is discussed, and recommendations for further characterization of bubble departure processes are made.

Heat Transfer Mechanisms During Boiling on Low Thermal Capacitance Heaters

Mattia Bucci, Matevž Zupančič, Iztok Golobič

University of Ljubljana

Email: mattia.bucci@fs.uni-lj.si

Nucleate boiling is a highly efficient heat transfer mode distinguished by the liquid-vapor phase change, which occurs through the formation, growth, and detachment of vapor bubbles from a heated surface. Its crucial role in various industrial applications, such as nuclear power plant operation and effective heat management in small electronic devices, has driven significant research efforts. However, despite extensive research dedicated to boiling investigations, there are still substantial knowledge gaps that hinder our ability to accurately predict heat removal rates. These knowledge gaps arise from the complex nature of small-scale boiling phenomena, which are further complicated by their strong dependence on operating conditions and the interactions between walls and fluids. In an effort to address some of these gaps, we conducted tailored horizontal pool boiling experiments using low thermal capacity metal heaters and fluids characterized by different thermophysical properties. We utilized a nanosecond pulsed fiber laser to perform surface texturing and create artificial cavities on the surface where bubbles nucleate and grow. We captured bubble dynamics through multiple synchronized diagnostic sources, including high-speed backlit imaging to track bubble growth, synchronized high-speed infrared thermometry to capture the corresponding thermal footprint on the boiling surface, and in-house developed fast-response micro-thermocouples to measure temperature at multiple locations within the liquid. Our findings reveal that the heat removal mechanisms during boiling exhibit distinct behavior on surfaces with low thermal capacitance compared to thick heated walls.

Nucleation, Growth and Bubble Detachment in Liquid-Vapor Phase Change on Structured Surfaces

Klara Arhar, Matic Može, Matevž Zupančič, Iztok Golobič

University of Ljubljana

Email: klara.arhar@fs.uni-lj.si

Comprehensive understanding of heat and mass transfer in applications involving liquid-vapor phase change hinges on management of nucleation, growth, and detachment of vapor bubbles. Various parameters influence the dynamics of phase-change heat and mass transfer and thus dictate the interactions between the surface, the liquid, and the vapor, profoundly impacting the underlying processes. To tailor these phenomena and harness them for technical applications involving high heat flux densities and intense mass transfer, such as boiling and electrolysis, surface functionalization is under intense development. By designing structured surfaces and creating preferential nucleation sites that promote heterogeneous nucleation, one can exert control over the locations and density of active nucleation sites on the surface. This, in turn, enables the regulation of bubble growth and detachment from the surface. With the aim of identifying optimal surface treatments for surface functionalization to enhance their performance in phase-change applications, we studied the nucleation, growth and detachment of a single bubble in a liquid-vapor phase change on untextured and laser-textured surfaces during water electrolysis. Platinum electrodes were used as platinum exhibits some of the most favourable properties for the hydrogen evolution reaction in acidic solutions. Electrolysis at various voltage values and in several electrolytes was performed and the bubble dynamics captured with a high-speed camera. High-speed bubble visualization and data analysis were employed to delve deeply into the intricacies of bubble growth and their mutual interactions on and above the surface. The impact of surface structuring on bubble dynamics was studied on several surfaces with different laser-induced structures. Laser texturing parameters were varied to achieve microstructures with varied distances between adjacent grooves and varied depth of the grooves. Finally, enhancement of the current density versus cell voltage was evaluated in comparison with the untreated surface.

Remaining Useful Life Prognostics During Boiling-Induced Surface Degradation

Jure Berce, Klara Arhar, Matic Može, Matevž Zupančič, Iztok Golobič

University of Ljubljana

Email: jure.berce@fs.uni-lj.si

After sustaining long-term nucleate boiling, any surface is prone to changes over time, which can be the result of various factors, such as surface degassing, oxidation, corrosion, adsorption of volatile organic compounds (VOC) and other carbon-based impurities or the appearance of a deposit layer (i.e., fouling). Additionally, in the case of structured and functionalized surfaces, the degradation of the fabricated surface micro- and nanostructures and/or coating can drastically affect boiling operation. Evidently, long-term boiling stability marks an important operational characteristic in reliable thermal management; however, it is difficult to attain in practical applications. Hence, to minimize energy losses and avoid system failures, one can adopt a prognostic approach to predict the remaining time until the performance of the system no longer meets the chosen performance criteria, i.e., the remaining useful life (RUL). In this work, we first present long-term pool boiling experiments on various samples of different surface topography and morphology in DI water and in an aqueous foulant solution. We show the changes induced by vigorous bubble nucleation over the course of several hundred hours of operation, discussing the underlying degradation phenomena. Following this, we employ a novel prognostic approach to predict the RUL of each sample. The method consists of (i) a Kalman filter to identify the degradation drift model and (ii) a Monte Carlo simulation to propagate the drift to the terminal threshold and obtain a RUL distribution. Its practical applicability is reinforced by advantageous features of: (i) non-intrusive prognostic performance irrespective of material, system size, and characteristics of the boiling surface, as well as of the severity of degradation, (ii) low computational and memory requirements and (iii) numerous options for individual tuning and extensions.

Study of Nucleate Boiling Growth Regime on Thin Surfaces.

Yohann Jaunet, Mattia Bucci, Matevž Zupančič, Julien Sebilleau, Catherine Colin, Iztok Golobič

Centre National de la Recherche Scientifique
Université de Toulouse
University of Ljubljana

Email: yohann.jaunet@imft.fr

In a context of energy transition, heat exchanges are a key for energy sobriety. Among those exchanges, nucleate boiling is the preferred heat transfer method for high heat flux. Vaporisation plays an important role in nuclear industry, but also in thermal machine such as cooling systems in data centres. In spite of decades studying boiling thermodynamics, modelling heat transfers in the nucleate boiling is still a major difficulty because of its numerous parameters. In the 1960s, scientists developed promising models called heat flux partitioning models [Bowring, 1962]. These models have been improved over the years, but they need closure laws about bubbles dynamics and nucleation site density [Kurul and Podowski, 1990; Basu, 2005]. For the purpose of improving those models, we took part of an international collaboration (ANR TraThI) to study interface heat transfer. We focused our study on boiling with a controlled nucleation site density. To do so, we began by studying isolated bubble in water pool boiling, and we aim to address multiple bubble interactions, in pool and flow boiling. In this presentation, we will focus on the bubble growth regimes observed in a water pool boiling experiment, with a distinction between microlayer and contact line growth regime. Our boiling surface is a textured thin metallic foil. It allows us to control nucleation location, creating patches with high probability of nucleation by means of nanosecond pulsed fibre laser. Moreover, we used high-speed infrared thermography and shadowgraphy to study heat transfer at the wall below the bubble and the bubble dynamics. This work shows that water nucleate boiling have two different growth regimes, depending on the imposed heat flux and the recent past of the nucleation site.

Online and Real-Time Measurement of Boiling Acoustic Emissions in Flow Boiling

Matthew Hughes, Matteo Bucci

Massachusetts Institute of Technology

Email: mthughes@mit.edu

Precise knowledge of boiling processes, including the identification of the boiling crisis, is essential for any thermal hydraulic design. In nuclear reactors for example, safely increasing the boiling heat flux results in higher output powers and even thermal efficiency. However, without precisely knowing when a boiling crisis may occur, the highest allowable heat flux must be very conservatively set to minimize the risk of boiling crises during steady-state operation, normal transients, and anticipated operational occurrences. Advances that may help with elucidating complex boiling phenomena include the development of high-quality, high-resolution measurement techniques such as high-speed video liquid-vapor phase detection and infrared thermometry. However, these methods can only feasibly “image” small areas, making it unlikely for such approaches to receive widespread adoption in systems with large volumes and fluid inventory. Moreover, there may be local, nonequilibrium effects that take place on regions of the heating surface that cannot be directly measured, which could lead to boiling crises before it would be observed by the aforementioned techniques or by semi-empirical models. An intriguing alternative to these high-resolution measurements is the detection of the boiling crisis and heat flux by measuring the acoustics emitted from vapor generation. Essentially, a microphone can track the relative change in harmonics and intensity over time across a large volume, which can then be related to the boiling regime and boiling heat flux. We develop a diagnostic approach that relies solely on acoustic emissions to uncover distinct acoustic signatures associated with different boiling regimes, advancing the field of non-intrusive thermal-hydraulic diagnostics. The insights gained from this research will offer invaluable contributions to the monitoring and control of boiling processes, paving the way for safer and more efficient thermal hydraulic designs in critical applications.

Superbiphilic Hierarchical Aluminum Surfaces for Exceptional Pool Boiling Performance

Armin Hadžić, Matic Može, Matevž Zupančič, Iztok Golobič

University of Ljubljana

Email: armin.hadzic@fs.uni-lj.si

The advancement in high-power electronic devices coupled with the need to ensure reliable and efficient heat dissipation within two-phase cooling systems incorporating boiling phenomena underscores the urgent necessity for breakthroughs in enhancing boiling performance and especially critical heat flux in order to minimize the risk of system failure. Thus, surfaces with tailored wettability have already demonstrated their potential to enhance boiling heat transfer intensity, while surfaces featuring permeable structures like micropillar arrays have shown significant improvements in CHF. In this study, we investigate the use of aluminum micropillar surfaces with tailored wettability to simultaneously enhance nucleate boiling heat transfer performance and increase the CHF. We fabricated the micropillar surfaces using a combination of nanosecond laser texturing and chemical etching in hydrochloric acid, while the wettability of selected surfaces was further tailored by application of a fluoroalkyl phosphonic acid and an additional laser texturing step. We tested three micropillar patterns under pool boiling condition using saturated twice-distilled water at atmospheric pressure. Importantly, our results revealed that the bottom part of the boiling interface (i.e., the superhydrophilic area) ensured increased liquid supply, while the top part (i.e., the superhydrophobic area) tended to provide much lower temperatures of transition into the high-heat-transfer-coefficient nucleate boiling regime. When combined, these two effects allowed us to simultaneously improve CHF and the heat transfer coefficient, resulting in enhancements of up to 113% (2343 kW m^{-2}) and 450% (205 kW m^{-2} K^{-1}), respectively, compared to the benchmark untreated surface. This research provides a practical and reliable approach to enhancing heat transfer by fabricating hierarchical surfaces, offering potential applications in ultrahigh heat flux thermal technologies.

The Role of Superhydrophobic Surface Stability in Anti-Icing Applications

Samo Jereb, Matic Možе, Matevž Zupančič, Iztok Golobič

University of Ljubljana

Email: samo.jereb@fs.uni-lj.si

Superhydrophobic surfaces are known for their low affinity towards water which is achieved through entrapment of air inside the micro/nanoscale surface features beneath the droplet, as described by the Cassie-Baxter wetting model. Very promising applications of poorly wettable interfaces were demonstrated in prevention of icing with the potential of substituting currently typical energy-consuming approaches, yet such superhydrophobic surfaces are rarely seen outside laboratory research. Excluding surface contamination and damage caused by environmental factors, this can mainly be attributed to the unstable nature of the air pockets. The latter are flooded by water under increased pressure, causing homogeneous wetting of the surface, also known as the Wenzel wetting state, impeding their future use. In this work, we focus on evaluating the resistance of superhydrophobic surfaces to pressure-induced Cassie-to-Wenzel wetting state transition on samples pretreated through laser texturing and hydrophobized via immersion in environmentally friendly hydrophobization media, namely saturated fatty acid (e.g., lauric, myristic, palmitic and stearic acid) solutions. The stability of resulting superhydrophobicity is evaluated by compressing a water droplet between two identically-treated substrates, while monitoring the force exerted by the droplet on the bottom surface. Afterwards, we fit a curve given by the Young-Laplace equation onto the droplet edge in order to determine the capillary pressure at which the wetting regime transition occurs. Furthermore, the anti-icing ability of the fabricated surfaces is evaluated through measuring the freezing delay by depositing droplets onto the samples and cooling the surface below the freezing point. Obtained results correlate the stability of the poorly-wettable interfaces to their performance in the prevention of freezing and thus illustrate the importance of the resistance of superhydrophobic surfaces towards wetting state transition for their practical use. This indicates that apparent contact angle measurements alone might be insufficient when evaluating poorly-wettable interfaces.

Study on the Difference in Flow Boiling Performance of Alloy Substrate Surfaces in a Rectangular Channel

Sihong He, Jiyun Zhao

City University of Hong Kong

Email: sihonghe2-c@my.cityu.edu.hk

Alloy materials with excellent properties are widely used in different industrial fields. Conducting flow boiling studies on alloy surfaces with excellent properties will help explore their applications in flow boiling, especially in heat exchange equipment in high heat flux densities and highly corrosive environments. Seven alloy materials were selected, including Monel, Inconel, Zr-4, 316SS, 304SS, Ti alloy, and FeCrAl. Under the conditions of pressure 1 atm, flow rate 0.5 m3/h and subcooling degree 6 K, a high-speed camera was used to observe the dynamic behavior of bubbles on the surface of alloy sheets of 1 × 1 × 0.1 cm in a 1 × 3 cm rectangular channel. Heat flux and wall superheat are calculated from temperature data collected by thermocouples. The surface roughness and contact angle of different alloys were analyzed through an optical surface profilometer and contact angle measuring instrument, and the boiling curves and bubble behaviors on the surfaces of different alloys were comparatively analyzed, focusing on the differences in critical heat flux (CHF).

Influence of Subcooling Conditions on Pool Boiling in Microgravity Conditions: Single Artificial Site Case

Lounès Tadrist, Fedor Ronshin, Oleg Kabov, Alexey Rednikov

Aix-Marseille Universite
Kutateladze Institute of Thermophysics
Université Libre de Bruxelles, Belgium

Email: lounes.tadrist@univ-amu.fr

Boiling experiments on an artificial nucleation site have been carried out on the International Space Station - ISS. The aim of these experiments is to understand the bubble nucleation-growth mechanisms involved in heat transfer during boiling. This unique experiment provides access to phenomena in the absence of natural convection. In fact the presence of gravity, the fundamental nucleation-growth phenomena are masked by gravity inducing natural convection. The RUBI experiment, installed on the ISS in 2019, has been used to carry out a series of experiments under several operating conditions during various measurement campaigns. After developing image analysis tools in the visible and infrared, we analyze the results in the case of nucleate boiling. In this paper, we will present results relating to the behavior of vapor bubbles in the case where the wall is heated and the surrounding liquid sub-cooled. The aim is to determine the geometrical characteristics of the bubble from nucleation through to large sizes that are inaccessible in the presence of gravity. These experiments are complemented by numerical simulation to gain access to phenomena taking place at microscopic scales.

Prediction of Heat Transfer Coefficient and Pressure Drop of Flow Boiling and Condensation Using Machine Learning

Edgar Santiago Galicia, Andres Hernandez-Matamoros, Akio Miyara

Saga Daigaku
Meiji University

Email: edgar@cc.saga-u.ac.jp

Heat transfer coefficient and pressure drop are two critical parameters in diverse thermal engineering applications, and the prediction of these parameter plays a crucial role in the develop of heat exchangers. Traditional methods often reliant ion empirical correlations based on limited refrigerants and experimental conditions. The present research explores the application of machine learning methods to predict the heat transfer coefficient and pressure drop of a wide range of pure and mixed refrigerants. The methodology involves the compilation over 30, 000 experimental observations encompassing an extensive range of operating conditions, heat exchanger geometries, and refrigerants. A total of 27 machine learning algorithms models, such as regression tree, support vector machines, and deep neural networks, are trained on this diverse dataset to discern intricate patterns and dependencies. After dividing the dataset in 80% for training and the rest for validation, the machine learning prediction shows an ability to predict the heat transfer coefficient and the pressure drop with a high accuracy in most of the refrigerants. Notably, the accuracy achieved by the machine learning models surpassed that of conventional correlations, highlighting their superior predictive capabilities. Furthermore, the accurate prediction of heat transfer enhancement methods added another layer of validation to the model's effectiveness. It is noteworthy that the dataset employed in this study has been made publicly accessible online.

Experimental Investigation of Electrically Enhanced Boiling of FC72 Using High-Resolution Phase-Detection Diagnostics

Marco Graffiedi, Alekos Ioannis Garivalis, Paolo Di Marco, Matteo Bucci

Massachusetts Institute of Technology
Universita degli Studi di Pisa

Email: mgraff@mit.edu

This work features the boiling of FC72 on a transparent substrate. The tests are performed using a thin film ITO (indium tin oxide) as the heat source, a fast speed video camera to capture the boiling process using the phase detection technique, an InfraRed video camera to capture the temperature distribution, and a metallic grid to impose an electric field perpendicular to the boiling surface. The maximum average electric field tested in this work is 3.3kV/mm, which led to an improvement of the critical heat flux of 18%. This study will analyze in detail the phase detection data showing that (1) there is no evidence of microlayer formation, (2) the triple contact line evaporation accounts for approximately 20% of the total heat flux, and (3) that the quenching stage accounts for approximately 80% of the total heat flux. Finally, the image segmentation will be used to corroborate the hypothesis that the boiling crisis can be modeled as a percolation limit phenomenon even in the presence of high electric fields.

Numerical Modeling and Simulation of Near-Contact-Line Evaporation Kinetics

Irina Graur Martin, Alexey Rednikov, Fedor Ronshin, Lounès Tadrist

Université Libre de Bruxelles, Belgium
Kutateladze Institute of Thermophysics
Aix-Marseille Universite

Email: irina.martin@univ-amu.fr

A mathematical model of the liquid-vapor interface based on the Onsager reciprocity relations and the kinetic theory of gases is proposed. In the framework of this model, the heat and mass fluxes through an interface are related to the pressure and temperature jumps across it. The developed model is applied to the wedge-type geometry to simulate near-contact-line behaviors, where the kinetic effects are of paramount importance in view of flux singularities. The results of implementation of this model to the case of FC-72 evaporation are shown in comparison to a very widely used Schrage-type relation.

Meshfree Methods and Applications in Thermal Sciences

Chairs: Boštjan Mavrič, C. S. Chen, Elisabeth Larsson, Andreas Karageorghis

Application of the Localized Method of Fundamental Solutions to Heat Transfer Problems

Csaba Gáspár

Szechenyi Istvan Egyetem

Email: gasparcs@math.sze.hu

Time-independent heat transfer problems can be described by special elliptic partial differential equations. The approximate solution of such problems still requires a large amount of computational effort. To reduce the necessary computational cost, a lot of advanced methods have been developed. One of them is the Method of Fundamental Solutions (MFS), which has become quite popular, since it is a truly meshfree technique; it requires neither domain nor boundary structured discretization. The method can be implemented in an extremely simple way, and the accuracy is often considerably high. However, it leads to a large linear system with a fully populated and possibly ill-conditioned matrix. In addition to this, in its original form, the method is restricted to homogeneous elliptic equations, the fundamental solutions of which are explicitly known. To overcome the above drawbacks, a number of special techniques have been developed, for instance, the localization techniques, which make the resulting linear system sparse. Due to the local character of these methods, the possible applications of the MFS can significantly be extended. In this paper, a special localization technique is described. The domain of the original problem is covered by several smaller subdomains in an overlapping way. In each subdomain, the problem is solved separately by using the Method of Fundamental Solutions. This results in an iterative method, which mimics the overlapping (alternating) Schwarz method. Though the rate of convergence is moderate, the high-frequency error components are reduced much faster. Therefore the iteration can be used as a smoothing procedure of a special multi-level method, which remains meshless at the same time. The approach can be generalized to inhomogeneous problems and also to some more general elliptic equation. Due to its advantageous numerical properties, the technique seems a useful generalization of the MFS. The method is illustrated through numerical examples.

An Adaptive One-Step Fictitious MPS-MFS Using Effective Condition Numbers

James Snead, C.S. Chen, Božidar Šarler

Mississippi Institutions of Higher Learning
University of Southern Mississippi
University of Ljubljana

Email: james.snead@usm.edu

The method of fundamental solutions (MFS) is a well-established numerical method for solving homogeneous boundary value problems (BVPs). Coupling with the method of particular solutions (MPS), the MFS can be extended to solve inhomogeneous BVPs effectively. In recent years, the ghost point method has been introduced to enhance the performance of the evaluation of particular solutions using radial basis functions (RBFs). The main idea of the ghost point method is to distribute the centers outside the domain. As a result, the so-called one-step MPS-MFS has three parameters to be determined; i.e., shape parameter of RBFs, location of sources of the MFS, and ghost circle radius of the ghost point method. In this paper, we propose applying effective condition numbers with an adaptive method for the optimal selection of these three parameters to achieve stable and high accuracy. Furthermore, due to the use of the particular solution, we revisit the idea of distributing the collocation points inside and outside the physical domain, which significantly simplifies the difficulty of selecting the internal collocation points, especially for complicated 3D domains. Five numerical examples are presented to demonstrate the effectiveness of the proposed approach.

An Efficient Localized Collocation Method Based on Nonsingular Fundamental Solutions for Heat Conduction Analysis in Functionally Graded Materials

Shuainan Liu, Zhuojia Fu, Qingguo Liu

Hohai University
University of Ljubljana

Email: shuainan_liu@163.com

This paper further develops a localized nonsingular method of fundamental solutions (LNMFS) for effectively solving transient heat conduction problems in functionally graded materials. The LNMFS approach is introduced for solving the inhomogeneous boundary value problems, where a pseudo-spectral Chebyshev collocation scheme (CCS) is employed for the approximation of the particular solutions. In the proposed LNMFS, the artificial boundary that is present in the existing localized method of fundamental solutions (LMFS)is eliminated. The singularities are replaced by the normalized area integrals of the fundamental solution over small circular disks, which encompass the source nodes that intersect with the collocation nodes. The proposed method demonstrates a high level of accuracy and efficiently eliminates the need to consider the optimal position of the artificial boundary. Additionally, the LNMFS overcomes the constraint of exclusively simulating heat conduction issues in functionally graded materials (FGMs) that exhibit specific spatial variations. Numerical tests demonstrate that the developed computational strategy can be considered as one of the competitive alternatives in the simulation of heat conduction behavior of functionally graded materials.

Improved Linear Notch Mechanics for Lightweight Design Using MFS

Wataru Fujisaki

Gakko Hojin Nakamura Sangyo Gakuen

Email: fujisaki@ip.kyusan-u.ac.jp

This paper is a study of a new prediction method for accurately evaluating the brittle fracture strength of notched materials with multiple holes using the methods of fundamental solutions. Conventional linear notch mechanics (LNM) was proposed in 1980's to clarify the distinction between the fracture and non-fracture zones using a master-curve. It is effective for evaluating the fracture strength of notched materials with a single hole. When multiple holes are adjacent to each other, the conventional LNM is less accurate in predicting the fracture strength. To eliminate the shortcomings, I proposed the improved linear notch mechanics (ILNM) , which uses the stress gradient χ at the bottom of the notch instead of the notch radius ρ. The physical meaning of ILNM for the multiple holes problem is the same as that of cracked problem using stress intensity factor K. The conventional linear notch mechanics is effective for evaluating the fracture strength of brittle materials with a single hole except for the big one. In this study, the applicability of ILNM and its effectiveness are verified for notched materials with multiple holes by some experiments and some calculations. Up to now, we have conducted systematic tensile tests and found this improved LNM can be applied to very big single notches and multiple holes. Improvements to the LNM will greatly expand its range of application and efficiently achieve both product weight reduction and mechanical safety at room temperature or high temperature.

Optimal Design in 2D Forced Convection Problem Using Generalized Finite Difference Method and Particle Swarm Optimization

Chiung-Lin Chu, Chia-Ming Fan

National Taiwan Ocean University

Email: 20952002@email.ntou.edu.tw

In this study, we develop a simulation-optimization model for forced convection problems to determine the optimum location of the outlet port in a two-dimensional square cavity with inlet and outlet ports. In the model, a numerical approach based on the localized meshless method, the generalized finite difference method (GFDM), and the Newton-Raphson method is used for efficient and accurate simulation and coupled with particle swarm optimization (PSO) to acquire the optimum location of the outlet port for the most efficient cooling design suggestions. The fluids in forced convection are forced to move by external sources, such as fans, pumps, air conditioning, etc., in order to increase the rate of heat transfer. When the velocity-vorticity formulation of the Navier-Stokes equations and the erengy equation can describe forced convection problems, a simulation model, combines the GFDM for spatial discretization and the Newton-Raphson method for solving the system of nonlinear algebraic equations, is adopted to obtain efficient and accurate numerical solution. Based on the merit that GFDM does not require mesh generation, numerical quadrature, parameter determination, and the rapid convergence speed of the Newton-Raphson method, we coupled them with PSO, one of the metaheuristic algorithms, to search the optimal location of the outlet port. In this study, several examples are provided to show the advantages of using the proposed simulation-optimization model.

Application of the Generalized Finite Difference Method for Solving Three-Dimensional High-Order Boundary Value Problems

Tsung-Han Li, Chia-Ming Fan, Po-Wei Li

National Taiwan Ocean University
Qingdao University

Email: y1024501@gmail.com

In this research, a combination of generalized finite difference method (GFDM) and fictitious-nodes technique is proposed to accurately solve three-dimensional high-order boundary value problems (BVPs). Traditional numerical methods such as FDM, FVM, and FEM could be difficult and rarely discussed to deal with high-order partial derivatives. Therefore, the GFDM, a localized meshless method, is proposed to solve BVPs governed by high-order partial differential equations (PDEs). The GFDM was developed by adopting Taylor series expression and moving-least square method. By applying the GFDM, the partial derivatives can be approximated as the linear accumulation of functional values and weighting coefficients at each node and the nearby nodes on computational domain. In addition, to enforce the satisfaction of the governing equation at interior nodes and the boundary conditions at boundary nodes will result in the system of algebraic equations. After that, the numerical solution can be efficiently obtained by solving the sparse system of linear algebraic equations. However, an overdetermined system is formed due to the multiple boundary conditions for solving high-order PDEs, which might cause numerical instability and increased numerical errors. As such, the fictitious-nodes technique was adopted, and the resultant square matrix can be solved stably and accurately. In this research, several numerical examples of three-dimensional BVPs governed by high-order PDEs will be presented to validate the feasibility and accuracy of the proposed technique of the GFDM. The stability and consistency of proposed method will be tested and verified through various parameters.

Phase-Field Formulated Meshless Simulation of Rayleigh-Taylor Instability Problem

Khush Bakhat Rana, Boštjan Mavrič, Božidar Šarler

University of Ljubljana

Institute of Metals and Technology, Ljubljana, Slovenia

Email: khush.bakhat.rana@fs.uni-lj.si

An immiscible Newtonian two-liquid system with different densities and the same viscosity, influenced by gravity, is based on the phase-field method (PFM) formulation. The solution of the related governing coupled Navier-Stokes (NS) and Cahn-Hillard equations is structured by the meshless Diffuse Approximate Method (DAM and Pressure Implicit with Splitting of Operators (PISO). The variable density is involved in the inertial and buoyancy terms. The related moving boundary problem is handled through single-domain, irregular, fixed node arrangement in two-dimensional Cartesian coordinates. The meshless DAM uses weighted least squares approximation on overlapping subdomains, polynomial shape functions of second-order and Gaussian weights. Implicit time discretization is performed for the NS and CH equations in the momentum predictor and PF variable corrector steps of PISO, while the momentum corrector steps solve the NS equation explicitly. This solution procedure has improved stability compared to Chorin’s pressure-velocity coupling, previously used in meshless solutions of related problems. The Rayleigh-Taylor instability problem simulations are performed for an Atwood number of 0.76. The DAM parameters (shape parameter of the Gaussian weight function and number of nodes in a local subdomain) are the same as in the author’s previous studies on single-phase flows. The simulations did not need any upwinding in the range of the simulations. The results compare well with the mesh-based finite volume method studies performed with the open-source code Gerris.

Numerical Solutions of Direct and Inverse Three-Dimensional Boundary Value Problems by Using the Method of Fundamental Solutions and the Particle Swarm Optimization

Chia-Ming Fan, Fu-Li Chang, Chiung-Lin Chu

National Taiwan Ocean University

Email: cmfan@ntou.edu.tw

In this study, a novel meshless numerical scheme, which is the combination of the method of fundamental solutions (MFS) and the particle swarm optimization (PSO), is proposed to efficiently, accurately and stably deal with direct and inverse boundary value problems in three-dimensional computational domain. The MFS, truly free from mesh generation and numerical quadrature, is one of the most powerful boundary-type meshless method, since only boundary nodes are required for numerical simulation. In the MFS, the numerical solutions are expressed as a linear combination of fundamental solutions, which should be placed out of the computational domain. In this study, the optimal locations of fundamental solutions are determined by the PSO, which is one of the metaheuristic evolutionary algorithms. Several numerical examples are provided in this presentation to verify the merits of the proposed meshless numerical schemes, while three-dimensional direct and inverse boundary value problems are considered.

Keynote Lecture:
Multi–Level Method of Fundamental Solutions for Solving Polyharmonic Problems

Andreas Karageorghis, C.S. Chen, Pihua Wen

Panepistemio Kyprou
University of Southern Mississippi
Nanchang University

Email: andreask@ucy.ac.cy

We consider a multi-level method of fundamental solutions (MFS) for solving polyharmonic problems governed by $\Delta^{\mathcal{N}} u = 0, \mathcal{N} \in \backslash\{1\}$ in both two and three dimensions. Instead of approximating the solution with linear combinations of $\mathcal{N}$ fundamental solutions, we show that, with appropriate deployments of the source points, it is possible to employ an approximation with a linear combination of only the fundamental solution of the operator $\Delta^{\mathcal{N}}$. The results of several numerical tests are presented and analyzed. In addition, we show that the proposed technique, when applied to boundary value problems in circular or axisymmetric domains, lends itself to the application of matrix decomposition algorithms.

Deep Residual Network for Interpolation and Inverse Problems

Amir Noorizadegan, C-S David Chen, Der Liang Young

National Taiwan University

Email: a.noorizadegan@gmail.com

In this paper, we introduce a groundbreaking neural network architecture known as the Power-Enhancing Residual Network, tailored to significantly enhance the interpolation capabilities of neural networks in both 2D and 3D environments for both smooth and non-smooth functions. By incorporating power terms within the residual elements, this architecture amplifies the network's expressive capacity, unlocking a realm of new possibilities in deep learning. Our study delves into critical aspects of network design, including network depth, width, and optimization techniques. It systematically showcases the adaptability and performance advantages of the Power-Enhancing Residual Network. The results consistently underscore the remarkable precision achieved by this innovative architecture, particularly when handling non-smooth functions. Real-world examples validate its superiority over conventional neural networks in terms of accuracy, convergence speed, and computational efficiency. Furthermore, our investigation extends to exploring the effects of deeper network configurations. In a practical application, we deploy the Power-Enhancing Residual Network to tackle the inverse Burgers' equation, demonstrating its superior performance in solving complex real-world problems. In conclusion, the Power-Enhancing Residual Network stands as a versatile and transformative solution, greatly elevating the capabilities of neural networks and fostering new horizons in machine learning.

Superconvergent Behaviour of the RBF-FD Method

Andrej Kolar-Požun, Mitja Jančič, Miha Rot, Gregor Kosec

Slovenska akademija znanosti in umetnosti

Email: andrej.pozun@ijs.si

One of meshless methods that has been gaining popularity in the context of solving Partial Differential Equations (PDEs) is the Radial Basis Function-generated Finite Difference (RBF-FD) method. In RBF-FD, we associate to each discretisation point a neighbourhood known as a stencil. We can define an interpolant on it as a linear combination of chosen Radial Basis Functions (RBFs) and possibly monomials. In this paper we opt for a common, simple choice of a RBF – radial cubics. To have a well-defined interpolation problem, it turns out that we must also include a set minimum amount of monomials in our basis. We can then form a linear system for the unknown basis expansion coefficients, which is uniquely solvable as long as our stencil is unisolvent. In order to use this method to solve PDEs, we can then apply a linear differential operator to the afromentioned interpolant and obtain the differentiation weights for each stencil. It is then a standard procedure to build a sparse system, discretising the PDE and solving it. It is easy to see that the convergence order of such an operator approximation is determined by the augmentation degree. In this paper we present a more interesting observation – convergence rate of the solution error is of an order higher than expected for even augmentation degrees. After presenting this »superconvergence« result we perform some further tests of its behaviour. We then proceed by studying it more systematically using an explicit formula for the operator approximation error. This allows us to analyse the error term by term and notice a different behaviour for terms containing odd or even powers of fill distance. We conclude by listing some possible ways of proceeding in our analysis.

Comparing Stabilisation Methods for RBF-FD Method Applied to Pure Advection

Miha Rot, Gregor Kosec

Slovenska akademija znanosti in umetnosti

Email: miha.rot@ijs.si

The numerical approximation of advection is usually one of the main sources of instabilities in the simulation of transport problems. For example, the simulation of fluid flow problems at higher flow velocities breaks down due to numerical instabilities in the advection term. In this work, we focus on the problem at hand in the context of the popular meshless radial basis function-generated finite difference (RBF-FD) method by analysing the pure advection equation in a fully periodic, i.e., infinite domain. First, we evaluate the behaviour of the method in terms of its parameters, namely the radial basis order, the augmentation order and the support size. We continue the analysis by comparing the two most popular methods for stabilising the numerical treatment of the advection term, namely the upwind and the hyperviscosity.

RBF WENO Reconstructions with Adaptive Order Using a Simplified Smoothness Indicator and Applications to Conservation Laws

Chiehsen Huang

National Sun Yat-sen University

Email: huangcs@math.nsysu.edu.tw

A two-level RBF based weighted essentially non-oscillatory (WENO) reconstruction with adaptive order (RBF-WENO-AO) was developed by the author for solving the hyperbolic conservation laws. WENO-AO reconstructions use arbitrary linear weights, and so they can be developed easily for RBF approximations, even on nonuniform meshes in multiple dimensions. However, WENO schemes require the computation of a smoothness indicator. This can be expensive, especially in multiple space dimensions. To circumvent the problems, we proposed a modification of WENO-Z weighting that gives a reliable and accurate reconstruction of adaptive order using of the simple smoothness indicator $\sigma^{SS=\frac{1}{N_{SS-1}\sum_j (j-m)^2}}$, where N_{SS} is the number of mesh elements in the stencil, $\square_j$ is the local function average over mesh element j, and index m gives the target element. We denoted the scheme as SWENOZ-AO. We propose to utilize the SWENOZ-AO weighting while using RBF based stencil functions for reconstructions (RBF-SWENOZ-AO). We show RBF-SWENOZ-AO is about 10 times faster in two space dimensions than RBF-WENO-AO reconstructions which use the classic smoothness indicator. The computational efficiency (CPU time versus error) is expected to be noticeably improved when solving the hyperbolic conservation laws.

An Improved Two–Step MPS–MFS Ghost Point Method with Effective Condition Number

C.S. Chen, Lionel Amuzu, James Snead

Mississippi Institutions of Higher Learning
University of Southern Mississippi

Email: cschen.math@gmail.com

Effective condition number (ECN) has been adopted to determine the optimal shape parameter of radial basis functions (RBFs), the location of the fictitious centers of RBF, and the source location of the method of fundamental solutions in a two-step MPS-MFS method for solving elliptic partial differential equations. The method ECN is primary based on the Uncertainty Principle of RBFs which implies the worse the condition number, the better the accuracy. In this presentation, motivated by Uncertainty Principle of RBFs, we propose to apply the effective contion number as a numerical tool to determine a number of parameters in the two-step approach using method of particular solutions (MPS) and the method of fundamental solutions (MFS).

On Different Implementations of Boundary Conditions in the Meshless RBF-FD Method for Phase-Field Modelling of Dendritic Solidification

Tadej Dobravec, Boštjan Mavrič, Božidar Šarler

University of Ljubljana, Faculty of Mechanical Engineering
Institute of Metals and Technology, Ljubljana, Slovenia

Email: tadej.dobravec@fs.uni-lj.si

Dendritic morphology is one of the most common microstructures in solidifying metallic materials. The phase-field method represents the most suitable approach for modelling the evolution of dendritic morphology. The meshless RBF-FD method and forward Euler scheme are used to solve the partial differential equations arising from the phase-field model. We consider the growth of a single dendrite into a supercooled pure melt. The computational domain is square (2-D) or box (3-D). On the computational domain's surfaces, we apply symmetric boundary conditions. Special care has to be taken in the RBF-FD method to satisfy boundary conditions accurately. In this paper, we test different implementations of boundary conditions. In the first implementation, the boundary conditions are incorporated when constructing an interpolation problem in local support domains containing boundary nodes. In the second implementation, the interpolation problem in each local support domain does not consider boundary conditions, which are satisfied by solving a system of linear equations for values in all boundary nodes at each time step. The second implementation can be upgraded with the use of ghost nodes. We show that using ghost nodes is very suitable for implementing boundary conditions in the RBF-FD method. Alternatively, the values in ghost nodes can be determined by using finite-difference stencils in the boundary nodes. The latter implementation is computationally very effective; however, it is less accurate. The accuracy and computational efficiency of different implementations are compared. We discuss the advantages and disadvantages of each implementation.

Solving Nonlinear Time-Dependent PDEs Using Polyharmonic Splines and Polynomials

Guangming Yao, Shawn Murphy, Trevor Francisco, Olaoluwa Ogunleye

Clarkson University

Email: gyao@clarkson.edu

In this paper, the improved localized method of approximated particular solutions (ILMAPS) using polyharmonic splines (PHS) together with a low-degree of polynomial basis is used to approximate solutions of various nonlinear time-dependent Partial Differential Equations (PDEs). The method uses shape parameter-free radial basis function (RBF). The discretization process is done through a simple collocation technique on a set of points in the local domain of influence. Resulted system of nonlinear algebraic equations is solved by the Newton's method. The performance of the proposed method is tested on various nonlinear PDEs. The performance of the method is compared with other bases such as multiquadrics (MQ) basis functions, and with finite difference method. The numerical experiments suggest that ILMAPS with polyharmonic splines yields considerably superior accuracy than other RBFs as well as other approaches reported in the literature for solving nonlinear elliptic PDEs.

An Effective Algorithm for Computing the Effective Condition Number with Application to RBFs

Tsung-Lin Lee, Amir Noorizadegan

National Sun Yat-sen University
National Taiwan University

Email: leetsung@math.nsysu.edu.tw

Radial basis functions have been one of the most accurate ways of solving interpolation and partial differential equations in a mesh-free manner. Two main difficulties for RBFs are the selection of the scale or shape parameter and the problem of ill-conditioning. We develop an accurate and efficient algorithm to obtain the effective condition number for finding a good shape parameter of radial basis functions.

Meshfree Boundary Integral Equation Method for Calculating the Conduction Shape Factor of Exchanger Tubes Containing Slits

Jia-Wei Lee, Yang Homg Wen, Jeng-Tzong Chen

Tamkang University
National Taiwan Ocean University

Email: jwlee@mail.tku.edu.tw

Following the successful experience of applying the meshfree boundary integral equation method (BIEM) to determine the conduction shape factor of heat exchanger tubes, this paper extends to those containing slits. The main difference between the present method and the conventional boundary element method (BEM) is that the adaptive exact solution and Gaussian quadrature are simultaneously employed to technically calculate the singular integral free of the sense of Cauchy principal value in numerical implementation. When dealing with the boundary value problem containing a slit or so-called degenerate boundary, a rank-deficient influence matrix due to a degenerate boundary may occur. To overcome the rank-deficiency problem, we introduce the dual BIEM with the hypersingular boundary integral equation to obtain independent equations for collocation points on the slit. A feasible adaptive exact solution is also required for the problem with a degenerate boundary. Since the jump behaviour cannot be described by the previous adaptive exact solution using the Cartesian coordinates for the corresponding collocation point on the slit, we adopt the harmonic basis function in the elliptical coordinates to construct the new adaptive exact solution. After comparing available exact solutions of conduction shape factor, the obtained data of meshfree BIEM isconsistent with those in the literature. However, the numerical instability due to the degenerate scale of an outer boundary is also observed. No matter what kind of slit is considered, the degenerate scale is dependent only on the size of outer boundary. Moreover, the boundary layer effect is also treated in the present method. In order to avoid the appearance of numerical instability due to a degenerate scale, regularized techniques are employed. Stable conduction shape factors for any size are obtained by using the proposed approach with regularized techniques.

Functional Gradation of Material Coefficients and Size –Effects in Heat Conduction: Numerical Simulations

Vladimir Sladek, Jan Sladek

Slovenska akademia vied

Email: vladimir.sladek@savba.sk

It is well known that the classical theory of heat conduction is scale invariant. On the other hand, it is experimentally evident that size-dependent effects are observable in small samples of micro/nano-scale dimensions. Incorporation of higher-order gradients of primary field variables into constitutive relationships yields a qualitative explanation of size-effects. Form the mathematical point of view, the governing equations are given by the partial differential equations (PDE) with higher-order derivatives in higher-grade continuum theories. As compared with classical theory of continua, the other complication of governing equations occurs in case of continuous media with functional gradation of material coefficients, when the problem is described by the PDE with variable coefficients. The traditional weak formulations are considered in global sense, hence the whole analysed domain/boundary is to be discretized into finite size elements. On the other hand, the strong formulations bring better computational efficiency because of elimination of integrations, but the price which should be paid is the need to approximate higher order derivatives of field variables. One of the most criticized shortcoming of the finite element (FE) approximation is its limited continuity on element intersections. The C0 continuity is insufficient for calculation of gradients of field variables on element intersections as well as for numerical solution of problems with governing equations of higher than 2nd order partial differential equations (PDE). Recently widely spread and elaborated mesh-free approximations utilize the higher order continuous shape functions. Another advantage is elimination of discretization of the analysed domain into finite elements, since only the nodes are scattered on the domain and its boundary. Both the strong and weak formulations are applicable in combination with mesh-free approximations. The Moving Finite Element Approximation (MFEM) and its utilization in mesh-free formulations for heat conduction problems is presented in this paper.

Joule Heating Analyses in Electrically Conductive Micro/Nano-Sized Structures

Jan Sladek, Vladimir Sladek, Miroslav Repka

Slovak Academy of Sciences

Email: jan.sladek@savba.sk

The Joule heating in electrically conductive materials creates a high temperature around the crack tip. For that temperature distribution one can observe a compressive stress state mainly at the crack tip vicinity. These compressive stresses can be utilized for the crack arrest. One can see that Joule heating is a multi-physical problem with the coupling of thermo-electric-mechanical fields. The thermal effects studies on the stress distribution around the crack tip in electrically conductive materials are investigated for the crack arrest in macro-sized structures only. Cracks are also observed in microelectronic structures, where the structure size is comparable to the internal characteristic lengths (e.g., lattice spacing, grain size) of the material and the size effect has been confirmed experimentally. Due to the size effect the application of the classical continuum mechanics is not justified in micro/nano-sized structures. In the present paper the size-effects are considered for both mechanical equation and the thermal transport. The thermal transport can't be well described by classical Fourier's law here. Therefore, a novel gradient theory is developed for the heat conduction. Similarly, for the mechanical field description the strain gradients are included into the constitutive equations for the double stress tensor. Due to occurrence of higher order of derivatives in our multi-physical problem the classical finite element method with C0 continuous approximation cannot be applied. Therefore, the collocation mixed FEM with large strain gradients and higher order flux are applied to our multi-physical problem.

Numerical Modelling of Continuous Casting of Round Billets with Turbulence in the Melt

Katarina Mramor, Zlatko Rek, Robert Vertnik, Božidar Šarler

University of Ljubljana, Faculty of Mechanical Engineering
Štore Steel d.o.o.
Institute of Metals and Technology, Ljubljana, Slovenia

Email: katarina.mramor@fs.uni-lj.si

The present work's purpose is to solve the continuous casting benchmark problem in axisymmetry with turbulence in the melt by the meshless method. The physical model of the liquid-solid system that involves mass, momentum and energy conservation is formulated in the mixture continuum approximation. The melt is assumed Newtonian and incompressible. A k-epsilon Reynolds averaged Navier-Stokes turbulence model is implemented with Abe-Kondoh-Nagano closures. The mushy region is modelled as a Darcy porous media with the Kozeny-Carman permeability model. The solution of partial differential equations is implemented locally using collocation with radial basis functions and explicit time-stepping. The velocity pressure coupling is performed by the fractional step method. The validation of the model is assessed by comparison with the finite volume method results. A sensitivity study of the process parameters of the model is performed on the velocity, temperature, and solid fraction fields.

The Method of Fundamental Solutions and Radial Basis Functions for Direct and Inverse Bioheat Transfer Problems

Jakub Grabski

Politechnika Poznanska

Email: jakub.grabski@put.poznan.pl

The heat transfer in living tissues is commonly known in the literature as bioheat transfer. For example, the Pennes equation can mathematically describe this phenomenon. Furthermore, in the direct problem, all parameters, boundary, and initial conditions are known, and the temperature inside the tissue is to be determined. In the inverse problem, some parameters, boundaries, or initial conditions are also unknown. An example of the inverse bioheat problem is the location of the tumor inside the tissue based on the surface temperature. In this work, some numerical procedures are proposed with the method of fundamental solutions and radial basis functions, based on which the direct and inverse bioheat transfer problems can be solved.

Keynote Lecture: Meshfree RBF-FD Methods for Numerical Simulation of PDE Problems

Elisabeth Larsson, Boštjan Mavrič, Andreas Michael, Fatemeh Pooladi, Igor Tominec

Uppsala University
University of Ljubljana
Persian Gulf University
Stockholm University

Email: elisabeth.rbf@gmail.com

The meshfree RBF-FD method is becoming an increasingly popular discretization method for challenging PDE problems due to its flexibility with respect to geometry and its ease of implementation. Using a combination of poly harmonic spline basis functions and polynomial basis functions has proven to be particularly successful with a convergence rate depending on the polynomial degree and good behavior at boundaries due to the contribution of the spline part. A challenge that has been observed is that derivative boundary conditions can give rise to large errors. A known remedy for this problem is to introduce one or more layers of ghost points, which improves the boundary approximations. In a recent paper, Tominec, Larsson, Heryudono (2021), it was shown that using oversampling is another effective way to handle this problem. Oversampling also allows for theoretical analysis of the method, which can then be viewed as a discretization of a continuous least-squares projection. By looking deeper into the error estimates we gain an understanding about how to best implement the method. The two main errors that occur are an integration error and a smoothing error. Both will be discussed in detail both from a theoretical and practical perspective.

Simulation of Steel Billets Moving in the Reheating Furnace by a Meshless Method

Qingguo Liu, Umut Hanoglu, Božidar Šarler

Institute of Metals and Technology, Ljubljana, Slovenia

Email: qingguo.liu@imt.si

The purpose of this work is a thermal analysis of multiple steel billets inside a continuous pusher-type reheating furnace from the entry to the exit by using a novel meshless solution procedure. In the reheating procedure, the moving steel billets in the furnace are heated from room temperature up to the required 1200 ˚C. The suitably heated billets are then processed at the reversing rolling mill. The heat diffusion equation is solved by the meshless local radial basis function collocation method (LRBFCM) in a strong form with explicit time-stepping. The radiative and convective heat fluxes are applied at the boundaries. The temperature of the moving billets is solved for the corresponding time and position inside the furnace. Ray tracing procedure is employed to determine the radiative heat flux. The meshless simulation system is verified on a benchmark with a reference solution obtained by commercial FVM-based software. A comprehensive sensitivity study is performed on the influence of the distance between the billets and the total time of the billet in the reheating furnace. An optimal solution in terms of energy consumption may be obtained from these simulations. For the first time, LRBFCM is successfully employed to solve such a complex industrial problem.

Lessons From Accelerating an RBF-FD Phase-Field Model of Dendritic Growth on GPUs

Boštjan Mavrič, Tadej Dobravec, Božidar Šarler

University of Ljubljana, Faculty of Mechanical Engineering
Institute of Metals and Technology, Ljubljana, Slovenia

Email: bostjan.mavric@fs.uni-lj.si

Phase-field modeling of dendritic growth presents the state of the art in the field of solidification modeling and are usually implemented using finite difference models combined with explicit time marching and accelerated by using GPUs. They are a prime candidate for such acceleration, since they require many arithmetic operations on relatively low ammount of data. We present an attempt at porting an existing RBF-FD code optimized for CPU execution to use GPU acceleration while keeping the resulting implementation portable between architectures. We discuss the acceleration achieved, scaling and implementation issues and critically discuss current landscape of GPGPU offerings.

Thermo-Elasto-Plastic Simulation of Hot Shape Rolling of Steel by a Meshless Method

Umut Hanoglu, Božidar Šarler

University of Ljubljana

Email: umut.hanoglu@fs.uni-lj.si

The aim of this research is to simulate large deformation of steel in a continuous hot rolling mill by using a novel meshless solution procedure. During the rolling process, steel billets at high temperatures go through multiple roll passes, each with a specific groove shape and roll gap. The major outcome of this simulation system is to test and design the roll pass design; i.e. if the final shape is inside the acceptable range or not. For this purpose, a 2D slice model approach is used to obtain fast and accurate simulation results. An elasto-plastic material and heat transfer model is used to obtain displacements, strains and stresses based on a return mapping algorithm together with a strong form force equilibrium equation, coupled to the heat equation. A novel local radial basis function collocation method (LRBFCM) is used for calculating unknowns through local interpolation in the differential equations. Predefined slice positions are entered into the system, and for each position, the slice contact line and amount of reduction is predicted and used as boundary conditions. Thermal and mechanical models are run in a sequential way and this is repeated until the final slice position after the exit from the last roll is reached. Several comparisons have been made with an established FEM software and industrial experiments. For the first time, LRBFCM is shown to be successful in solving complex thermo-elasto-plastic large deformation problems of hot rolling.

Enhancing Meshfree Methods with RBFs: An Approach Using Approximate Least Squares Solutions

Leihsin Kuo, C.S. Chen

State University System of Florida
University of Southern Mississippi

Email: lkuo@uwf.edu

This study presents an innovative solution to address rank-deficient linear least-squares problems. Our proposed method employs advanced techniques to overcome challenges without resorting to time-consuming matrix decomposition. Numerical experiments showcase the ease and efficiency of our approach. In addition, our study further apply the technique to secures a unique solution for the Kansa mesh-free method with RBF. Furthermore, our approach mitigates the oscillation of approximation errors, particularly when confronted with ill-conditioned interpolation matrices, similar to regularization.

Hyperviscosity Stabilisation of the RBF-FD Solution to Natural Convection

Miha Rot, Žiga Vaupotič, Gregor Kosec

Slovenska akademija znanosti in umetnosti

Email: miha.rot@ijs.si

The numerical stability of fluid flow is an important topic in computational fluid dynamics as fluid flow simulations usually become numerically unstable in the turbulent regime. Many mesh-based methods have already established numerical dissipation procedures that dampen the effects of the unstable advection term. When it comes to meshless methods, the prominent stabilization scheme is hyperviscosity. It introduces numerical dissipation in the form of a higher order Laplacian operator. Many papers have already discussed the general effects of hyperviscosity and its parameters. However, hyperviscosity in flow problems has not yet been analysed in depth. In this paper, we discuss the effects of hyperviscosity on natural convection flow problems in the turbulent regime.

Fourth-Order Phase-Field Simulation of Cracks Using Strong Form Meshless Method

Izaz Ali, Gašper Vuga, Boštjan Mavrič, Umut Hanoglu, Božidar Šarler

University of Ljubljana

Institute of Metals and Technology, Ljubljana, Slovenia

Email: izaz.ali@fs.uni-lj.si

The prediction of crack propagation in fracture mechanics is still a fundamental challenge. In recent decades, phase-field formulation has emerged as a powerful tool for the simulation of crack evolution, which offers a continuous representation of cracks from intact to broken material. For the first time, the present study combines a fourth-order phase field and a strong-form meshless method to investigate crack propagation in brittle elastic material. The local radial basis function collocation method, structured with the augmented polyharmonic splines, is used to solve the coupled mechanical and phase-field models. Due to the smooth nature of the fourth-order phase-field model, the sharp transition at the crack surface can be avoided. It is found that the presented method successfully predicts the crack propagation for a material subjected to tensile loading. The obtained results agree well with the benchmark case. Further study will include elastoplastic material behaviour and applications to crack formation in different steel processing steps.

Solutions to Full Very Ill-Conditioned Equation Systems

Edward Kansa

Convergent Solutions LLC

Email: edwardjkansa@gmail.com

Transcendental radial basis functions (C¥ RBFs), although being exponentially convergent, are notoriously ill-conditioned. Similarly, Hilbert and Vandermonde systems are also very ill-conditioned and can serve as proxies for C¥RBFs applications such as in solutions of integral, integto partial differential, and partial differential equations. The hypothesis employed her is that it is the deficiency of finite precision computers that is the root cause of ill-conditioning, and its unreliable answers is the finite and small precision of present computers, not of the mathematics represented per se. The algorithm chosen to solve such full, very ill-conditioned equation systems is the Block Gaussian Elimination (BGE) and the Arbitrary Precision Arithmetic (APA) algorithm presented here. Each separate algorithm addresses ill-conditioning and APA also addresses the problem with Schur component matrices used in the block elimination process. Amazingly accurate results were obtained with solutions to the Hilbert and Vandermonde systems. The system of ill-conditioned C¥ RBFs arising from the solution of Ñ2U = exp(5x1+5x2) on a unit square exhibits a complex topography of local maxima and minima due to the variation of the degrees of freedom of the data and evaluation centers as will as the shape parameters and the multiquadric exponential.

Morphology Optimised Design of Heat Exchangers

Chair: Sara Rainieri,

CFD Simulation and Geometric Optimization of a Novel Baffle Heat Sink for Power Electronics Cooling

Ilya T'Jollyn, Jana Rogiers, Jasper Nonneman, Michel De Paepe

Universiteit Antwerpen
Universiteit Gent

Email: ilya.tjollyn@uantwerpen.be

The electric drivetrain of an electric vehicle typically has two fluid circuits: an oil circuit for lubrication and a water-glycol circuit for cooling. Eliminating the water-glycol circuit can result in a more compact and lighter drivetrain. However, this requires the drivetrain components to be cooled with oil as coolant, which has inferior heat transfer properties compared to water-glycol mixtures. This is particularly challenging for the power modules in the power electronics unit, which exhibit some of the highest heat fluxes in the drivetrain. This study focusses on the design and optimization of a novel type of heat sink for laminar flows with baffles to guide the flow in four different directions. This has two main advantages: the baffles act as fins and increase the heat transfer area and they introduce a swirling motion in the oil thereby disturbing the boundary layers continuously. Computational fluid dynamics simulations using ANSYS Fluent are performed to evaluate the thermohydraulic performance of the heat sink. The influence of design parameters such as fin height, spacing and thickness and baffle length, height, thickness and pitch was evaluated by simulations with several combinations of the design space, considering manufacturing constraints. The optimization of the design showed that the fin thickness and spacing and baffle length should be as small as possible, while there were optimal values for fin height and baffle height, thickness and pitch. The simulations show that the thermal resistance of the baffle heat sink can be 20% lower than that of a benchmark pin fin heat sink at equal pumping power.

A Novel Type of Additively Manufactured High Pressure Mini-Channel Heat Exchanger for Precooling in Hydrogen Refueling Stations

Robin Kahlfeld, Felix Müller, Henrik Müntefering, Paul Gembarski, Ruben Steinhoff, Stephan Kabelac

Leibniz Universitat Hannover
FUNKE Wärmeaustauscher Apparatebau GmbH

Email: kahlfeld@ift.uni-hannover.de

During refueling supercritical hydrogen in high pressure storage tanks, the fluid has to be cooled down to temperatures between -33°C and -40°C before entering a vehicle fuel tank. This cooling takes place while the hydrogen is at a pressure of up to 875 bar. The requirements for the heat exchanger performing this task are extremely high. It has to be pressure resistant, compact enough to fit in the dispenser column and provide a high thermal performance to ensure a fast refueling with high mass flow rates. Very few conventional manufactured heat exchangers are able to fulfil these requirements. With the rise of additive manufacturing technology, especially laser powder bed fusion, new heat exchangers produced without use of conventional joining technologies can be realized. This manuscript presents a new type additively manufactured of mini-channel heat exchanger. It is developed in a joint research project involving the Leibniz University Hanover and an industrial heat exchanger manufacturer. The Apparatus has a design pressure of 1034 bar and will be suited to be used in hydrogen refueling stations. The thermal requirements and the design of the apparatus are presented. Thermal power and pressure drop for the full-size heat exchanger are calculated via a cell model. Scaled smaller heat exchangers made of 1.4404 stainless steel are additively manufactured via laser powder bed fusion. The thermofluiddynamical performance of the scaled apparatuses are measured in a testbench to verify the applicability of the used correlations. Deviations in hydraulic diameter and surface roughness are taken into account. Existing correlations are fitted to the new geometry.

Thermal Management of Multi-Chip Electronic Module with Wavy Fin Heat Sink-Assisted Immersion Cooling

Pratheek Suresh, Kasavajhula Naga Vasista, Shailesh Kushwaha, Balaji C

Indian Institute of Technology Madras

Email: me19d404@smail.iitm.ac.in

Single-phase immersion cooling technology has emerged as a promising solution to effectively manage the thermal needs of electronics in modern high-heat-flux-density data centers. This approach involves submerging electronic components, such as heat-generating chips, into a thermally conductive dielectric liquid. The present study investigates the thermal performance of single-phase immersion cooling applied to a third-generation open compute server using NOVEC 7500 as dielectric fluid. A wavy fin heat sink design is proposed to overcome the poor thermal performance of the dielectric fluid. The study focuses on exploring the effects of amplitude and wavelength of the wavy fin, the number of fins for each heat sink, as well as the Reynolds number at the inlet of the fluid flow, on the thermal performance of heat sink subjected to heat flux of 5 to 15W/cm^2. The governing equations and their corresponding boundary conditions are solved numerically using the finite volume method-based commercial software ANSYS 2021R1. The preliminary results showed an average temperature reduction of 7.6% for three processors aligned in the direction of fluid flow for the wavy fin heat sink compared to the plate-fin heat sink design. The study provides insights into the use of optimal heat sink with novel wavy fin for thermal management of high-heat-flux-density data centers.

Numerical Study of Solar Evaporation in 3D Printed Structures

Romain Fillet, Vincent Nicolas, Alain Celzard, Vanessa Fierro

Universite de Lorraine
Institut Universitaire de France

Email: romain.fillet@univ-lorraine.fr

Water scarcity poses a significant threat to human health. Europe has experienced numerous droughts in recent years, resulting in severe consequences for agricultural production. For example, in Hungary between January and October 2020, 4 million hectares of crops were adversely affected. France is also projected to experience a decline in the annual flow of rivers by 10% to 40% by 2050. Globally, the water stress situation is alarming, with a prediction that by 2050, 3.9 billion people—more than 40% of the world's population—will reside in river basins facing high water stress. One potential solution to mitigate this threat is the purification of polluted water. Solar evaporation has attracted significant interest as an effective method. In essence, this technique involves evaporating water from a material exposed to solar radiation at the water's surface and then condensing it over a surface, facilitating distillation and the recovery of clean water. The evaporation material is typically placed on an insulating layer and must be capable of transporting water and absorbing solar radiation. To enhance evaporation, 3D materials are employed, which expose more surface area to the air compared to 2D materials, thus augmenting heat and mass transfers. Building on a prior study, three-dimensional concave geometries were 3D-printed and filled with activated carbon, creating a unique 3D solar evaporator. The evaporation process was modelled considering air flux and solar rays to investigate the advantages of concave 3D geometries. This work encompasses a series of studies, focusing on mass transfer within the concavity and around the structure to comprehend the specific exchange points. Additionally, the research delves into studying the effects of increased air velocity and introducing additional holes to the structure, which are anticipated to enhance evaporation. Finally, shape optimization will be applied to the surface to fine-tune the concavity for optimal airflow.

Stratified Liquid-Liquid Flow Through Microchannels with Grooved Walls

Mainendra Dewangan, Tim Persoons

The University of Dublin Trinity College

Email: dewangam@tcd.ie

Patterned surfaces have numerous applications in the microscale flow regime, such as heat transfer enhancement, mixing, and microfluidics devices. The present study analytically examines the pressure-driven flow of two immiscible Newtonian fluids through a grooved microchannel. The orientation of the channel is defined as the top and bottom walls being flat and wavy surfaces, respectively. A no-slip boundary condition is assumed at both walls. The present problem is investigated by invoking the Fourier theory for a flow along streamwise grooves at the Stokes flow limit. Flow rates, shear stress and velocities of both fluids are determined analytically and numerically. A finite-element-based numerical study is conducted to understand the accuracy of the theoretical models. Results are generated to show the effects of viscosity ratio, wall undulation, aspect ratio and wavelength of the patterned channel. The spectral model can predict previously described models from the literature based on small amplitude theory and lubrication theory. Using the domain perturbation theory, the literature model is accurate only at small amplitude. For a wide range of wavelengths, the spectral model achieves good accuracy, even at large pattern amplitude. Roughness and confinement effects are captured with increasing pattern amplitude at different wavelengths. The Fourier theory only observes the transition phenomenon between roughness to confinement effect, whereas the flow rate may increase or decrease with pattern amplitude, as per the literature model. The present model agrees well with numerical values. This new understanding is condensed into a set of guidelines for selecting microchannel shapes and dimensions. The present study provides a deeper understanding of the stratified flow through microchannels with undulating surfaces, with potential applications in electronic cooling, skin-friction drag, interfacial fluid dynamics, enhancement and reduction in heat transfer.

Enhancing Fluid Distribution in Additively Manufactured Heat Exchanger Using Autogenerated Geometries

Joel Kimmich

Universitat Stuttgart

Email: joel.kimmich@igte.uni-stuttgart.de

Compared to conventional heat exchangers, such as tubes and plates, additively manufactured heat exchangers are not limited in design. Novel free-shape geometries enable the generation of more complex internal heat transfer geometries. The distributor, which connects the medium-carrying tube to the heat transfer geometry, received less attention in previous research studies than heat transfer geometries. This paper presents an approach to automate the generation and optimization of an additively manufactured distributor, that aims to distribute the fluid uniformly in the subsequent heat transfer geometry. This is demonstrated using a single-phase incompressible flow, but can be extended to multi-phase flows likewise. A fully parameterizable distributor is automatically generated by an iterative but deterministic algorithm. In this process, a flow cross-section is quasi-fractally split to generate the subsequent iteration layer of the distributor. The proposed algorithm is implemented object-oriented in Python. This allows the individual evaluation and modification of every geometric part. Through the quasi-fractal design high surface-to-volume densities are generated with few iterations. This yields increased heat transfer rates. The distributor and all connected geometries (pipe connections, heat transfer geometry and collector) are meshed and simulated in OpenFOAM using steady-state computational fluid dynamics. This is done by parsing all files and parameters to an OpenFOAM case. The volumetric flow rates determined from the simulation are assigned to the respective flow cross-sections during the distribution process. The flow cross-sections of the geometric model are modified within an optimization process, under consideration of prevailing volumetric flows. Through repetition of the process the quality of the results is improved. The printable geometry can be extracted directly out of Python using common file formats. The domain of the simulation is limited to the fluid region to prevent high computational costs.

Enhancing the Degrees of Freedom of Topology Optimization via Variable-Porosity Metal Foams: Design of Heat Conduction Paths in a Volume-to-Point Problem

Gerardo Maria Mauro, Nicola Bianco, Andrea Fragnito, Marcello Iasiello

University of Sannio
University of Naples Federico II

Email: germauro@unisannio.it

Multi-material topology optimization determines the optimal distribution of different materials within a design domain in order to achieve specific performance goals, e.g., minimizing average temperature, compliance or entropy generation. This work takes advantage of such technique to enhance the thermal performance of a benchmark heat conduction problem in the field of thermal management, ensuring a consistent basis for comparison. It consists in the design of heat conduction paths in a volume-to-point problem, considering a square domain with a uniform heat source and a Dirichlet condition at a point on the boundary. The aim is to investigate the impact of increasing the degrees of freedom of topology optimization introducing different materials, i.e., metal foams of variable porosities, in the design of heat sinks under a constant weight constraint. Interpolation functions via the solid isotropic material penalization (SIMP) method allow the properties – e.g., thermal conductivity – of materials to be correctly assigned. The distinction between materials is made by means of different thresholds on the projection function. By varying the foam porosity, the trade-offs between weight and heat dissipation efficiency are investigated. The objective is to find the ideal combination of materials that maximizes heat transfer – i.e., minimizing the average domain temperature – while complying with the weight constraints. Findings reveal that multi-material topology optimization – when applied to heat conduction problems – can outperform other methods at equal weight, such as the standard topology optimization and the constructal dichotomic tree with perpendicular branches, paving the way for innovative heat sink designs.

Measurement of Effective Thermal Conductivity of Composite Powders of 2D Materials and Metals for Additive Manufacturing

Hyunjong Lee, Apostolos Koutsioukis, Davoud Jafari, Bernard J. Geurts, Wessel W. Wits

University of Twente

Email: h.lee@utwente.nl

The ever-increasing demand for excellent thermal, mechanical, and electrical properties has driven research on new composite materials, such as the mixture of 2D materials and metals. Especially, graphene, one particular type of 2D material, exhibits a very high thermal conductivity of ~ 2,000 W/mK at room temperature, which originates from in-plane strong covalent bonding and reduced phonon scattering compared to 3-dimensional crystal structures. However, implementing the excellent properties of 2D materials in 3D configurations involving metals has not shown much progress. A promising solution for successful processing of 2D materials is additive manufacturing in which a part is fabricated layer-by-layer. To facilitate additive manufacturing processes, such as powder bed fusion, using composite powders that consist of 2D materials and metals, the effective thermal conductivity of the composite powder bed needs to be accurately determined in order to predict optimal processing conditions. For example, the effective thermal conductivity of the composite powder bed has an impact on the melt pool behavior during powder bed fusion. Also, the thermal history of 3D-printed parts, responsible for the part's microstructure and mechanical properties, is dependent on the powder bed's thermal conductivity. Hence, to design 3D-printed parts with optimal properties, the thermal properties of the composite powder bed are key parameters to investigate. In this study, we measure the effective thermal conductivity of both graphene-metal composite powders and pure metal powders as a function of the packing density. The packing density of the powder bed is set to the general range of 40-60% typically found in powder bed fusion processes. Furthermore, we vary the types of 2D material and metals, including graphene, h-BN, Cu, AlSi10Mg, and SS316L. Finally, microstructure observations and numerical modelling are applied to understand heat transfer through composite powder beds.

Effects of Morphological Features of Butterfly-Shaped Inserts on Thermal Performance Enhancement in Tubular Heat Exchangers: A Numerical Study

Luca Pagliarini, Fabio Bozzoli, Luca Cattani, Sara Rainieri

University of Parma

Email: luca.pagliarini@unipr.it

Single-phase heat transfer enhancement in heat exchangers represents a crucial issue for effective thermal treatment in many industrial applications, especially when highly viscous fluids or gases are processed. Increase of heat transfer capabilities might be obtained by either active or passive techniques. While the former guarantee a constant recirculation of the fluid flow through mechanical actuators, passive methods, such as tube corrugations or static inserts, promote boundary layers disruption at the fluid-wall interface without the use of any electrical power sources. Despite different insert solutions have been proposed over the past decades, including twisted tapes, wired coils and conical rings, the literature lack of optimal geometries definitions. The aim of the present work is to design a numerical model to be effectively used for extensive studies on the effects of morphological features of insert devices on heat transfer enhancement and pressure losses in heat exchangers. In the present investigation, novel butterfly-shaped inserts are investigated by solving conjugate conduction-convection heat transfer in the computational domain through a steady RANS approach. Turbulence is treated by means of the Realizable k-ε model with enhanced wall function. The results, compared with previously collected experimental data, highlight a very good agreement between the computed physical quantities and the ones experimentally assessed.

A New Test Section Made via Additive Manufacturing to Perform Local Heat Flux Measurements

Gianluca Cattelan, Marco Azzolin, Stefano Bortolin, Massimiliano Bonesso, Adriano Pepato, Davide Del Col

University of Padua

Email: gianluca.cattelan@studenti.unipd.it

Recent advancements in the field of additive manufacturing have enabled the realization of products that were previously almost unattainable and in very short timeframes. In the field of heat transfer, the production of heat exchangers through additive manufacturing is highly intriguing because it allows for the creation of compact heat exchangers that can be tailored to specific requirements and with specific geometries, given the freedom to design and the ability to produce optimized complex geometries. However, heat exchangers, especially for research applications, require appropriate design considerations with respect to the type of study that needs to be realized and the specific physical problems that need to be addressed before being manufactured through additive manufacturing. This paper outlines the design process of an innovative test section created through Additive Manufacturing technology to investigate two-phase heat transfer processes. A CFD analysis has been performed to optimize the geometry with the final goal of precisely determining the local heat transfer coefficients. The test section has been realized with the DMLS (Direct Metal Laser Sintering) technology using the AlSi10Mg alloy. Despite the benefits of this technology, the possible anisotropy of SLM-produced parts can have a significant effect on their thermal properties and thus it is important to investigate them, in particular the thermal conductivity. Thus, cylindrical samples have been produced by SLM with several build orientations to perform specific measurements of the thermal conductivity with the Hot Disk technique. To conclude, the paper also presents a preliminary assessment of the experimental technique, to determine the local heat flux, comparing some experimental data with existing correlations available in the literature.

Forced Air Flow Through a Rectangular Channel With 3D Turbulence Enhancers: Visualization of Flow Structures by Laser Sheet Scattering

Luigi Vitali, Maria Corti, Pasqualino Gramazio, Damiano Fustinoni , Alfonso Niro

Politecnico di Milano
Polytechnic University of Milan
Polytechnic Institute of Milan

Email: luigi.vitali@polimi.it

In recent times, the design of heat transfer devices is strongly evolving due to both the diffusion of additive manufacturing techniques, that use a wide variety of materials, and the CFD-based shape optimization techniques. Such technological advancements can be applied to enhance heat transfer inside channels for various applications. Considering a 1:10 AR rectangular channel, that reproduces the geometry of heat transfer devices used for batteries cooling and compact automotive HEs, the first step of the project towards the design of optimized turbulence enhancers is the validation of the CFD methodology. As for this project it's important to have a fast-running 3D simulation with physical meaning, several techniques of increasing complexity and computational requirements are currently employed, including RANS and LES. The validation is mainly carried out by experiments that measure both local and global heat transfer enhancement; however flow visualization is a powerful tool to get insight into physical mechanisms that generate convective heat transfer enhancement. In this paper, the setup of a laser sheet scattering method to visualize turbulent structures in a Plexiglas 1:10 AR rectangular channel ribbed on one of the main surfaces is presented, and results on a plane perpendicular to the ribbed surface and oriented along the flow direction are presented for V-down squared ribs for Re= 7500, and are compared with the results of LES simulations of the same flow in the same channel geometry.

Numerical Simulation-Based Design of Optimized Surfaces for Condensation Heat Transfer

Arianna Berto, Mario Junio Gabellone, Nicolò Mattiuzzo, Stefano Bortolin, Davide Del Col

University of Padua

Email: arianna.berto@unipd.it

Heat transfer enhancement is a well-established research topic due to its industrial relevance, mostly relying on the design of the morphology of heat exchangers. The advent of 3D printing, also known as additive manufacturing, has opened up the pathway for the design of compact heat exchangers with innovative shapes, complex geometries and smaller size. In particular, additive manufacturing technologies enable the direct translation of CAD/CFD models into physical prototypes, thus the multi-scale simulation of fluid flow and heat transfer is expected to play a key role for the design of heat exchanger components and the accurate selection of materials. With regard to filmwise condensation, design tools capable to precisely account for all the main parameters involved (such as gravity, surface tension, shear stress at the interface, waviness and turbulence in the liquid film) must be considered for the optimization of the condensing surfaces. The present study will focus on the preliminary design of innovative shapes for enhancing the condensation heat transfer in a grooved wick heat pipe. Most of the condensation heat transfer studies available in the literature deal with vapor condensing inside tubes or over cylindrical/plain surfaces, while the condensation process occurring inside heat pipes has been less investigated. The grooved wick surfaces must be designed with the aim to promote the drainage of the condensate and minimize the thickness of the liquid film forming over the wick structures, therefore maximizing the heat transfer coefficient due to surface tension effects. In the present work, such surfaces will be designed and studied during condensation of refrigerants by performing Volume-of-Fluid (VOF) numerical simulations on Ansys Fluent. The simulated condensing surfaces will be compared in terms of liquid film thickness and heat flux distribution along the grooves to identify the best heat transfer configuration.

Forced Air Flow Through a Rectangular Channel With 3D Turbulence Enhancers: Fluid-Dynamics and Thermal Analysis by LES

Maria Corti, Luigi Vitali, Pasqualino Gramazio, Damiano Fustinoni , Alfonso Niro

Polytechnic University of Milan
Polytechnic Institute of Milan

Email: maria.corti@polimi.it

Many newer engineering applications, like the cooling of automotive batteries and high-performance CPUs, require heat exchangers with large contact areas and very small heights. Performances of this type of HEs can be significantly improved by using surfaces enhanced with optimized 3D structures. The project Mood4hex, funded by the Italian Ministry of Universities and Research, aims at designing new geometries with optimized morphology, starting from a large database of experimental data on ribbed surfaces. In order to perform an optimization algorithm on such surfaces, analyses are carried out on CFD models with increasing complexity and computational time, to find a fast-running code that preserves the physical meaning. Authors have previously used RANS models, and now are investigating the use of Large Eddy Simulations (LES) as a higher order validation tool for simpler models. In this paper, we present the results of LES along with experimental data of a forced air flow inside a streamwise periodic rectangular channel of high aspect ratio (1:10) equipped with standard V-shaped ribs (α= 60°). Finally, ribs point the channel exit and are placed only on the lower wall that is operated at constant heat flux, while the operating Reynolds number is 7500. Validation of LES results vs experimental data shows promising agreement.

A Combined Experimental and Numerical Investigation of the Potential for Heat Transfer Enhancement in a Rectangular Minichannel Using Half Rectified Pulsating Laminar Flows

Parth S Kumavat, Sajad Alimohammadi, Seamus O'Shaughnessy

The University of Dublin Trinity College
Technological University Dublin

Email: kumavatp@tcd.ie

The continued miniaturization of electronic circuits and their associated high-power density requires more innovative cooling solutions. Pulsating flows in single-phase cooling systems are viewed as a potential solution due to their distinct features such as unsteady fluid mixing, flow reversal effects, and periodic disruption of thermal boundary layer. This study aims to bridge a knowledge gap by using experimental and computational methods to investigate the complex flow and thermal characteristics of laminar pulsating flow in a heated rectangular minichannel. Experimental analysis involves a uniformly heated thin foil approximating a constant heat flux bottom wall. Local, non-intrusive temperature measurements of the heated surface are recorded using a high spatial resolution infrared thermography system. Analogous to the experimental conditions, a three-dimensional conjugate heat transfer computational model is developed using ANSYS CFX, in which a volumetric heat generation source is imposed on the solid domain. In both experimental and computational approaches, positive and negative half rectified sinusoidal pulsating flow waveforms are studied at a fixed flow rate amplitude A0=3 and pulsation frequencies in the range 0.02 Hz < f < 25Hz, corresponding to a Womersley number range of 0.5 < Wo < 18.3. Results show the rectified waveforms exhibit peak values of axial pressure gradient and shear stress magnitudes resulting from intense fluid mixing promoted by the co-existence of steady and oscillating flow stages within one pulsating cycle. A thermal-hydraulic performance parameter is defined to contextualise the improvement in heat transfer. For the positive half rectified waveform, a marginal enhancement in heat transfer of 2.2% is obtained for Wo = 2.5, A0=3, with a marginal reduction in thermal-hydraulic performance due to the presence of high shear stresses. Whereas for the negative half rectified case an enhancement of 9% and 6% exists, and over twice the thermal-hydraulic performance compared to a steady flow.

Effect of the Plates Geometry on the Performance of a Cross-Flow Recuperator for Indirect Evaporative Cooling Systems

Roberta Caruana, Luca Marocco, Stefano De Antonellis, Manfredo Guilizzoni

Polytechnic University of Milan
Polytechnic Institute of Milan

Email: roberta.caruana@polimi.it

Air-to-air Indirect Evaporative Cooling (IEC) systems use the latent heat of vaporization of water to cool down an air stream, thus guaranteeing adequate thermohygrometric conditions with a lower energy consumption than traditional air conditioning solutions. IEC heat exchangers are usually composed of alternating primary and secondary channels, separated by thin conductive plates which often present protrusions and / or dimples. The primary (product) air flows in the primary channels, and it is cooled while maintaining a constant humidity ratio. The secondary (working) air flows in the secondary channels, either in “wet” conditions, namely in contact with mist flow and a liquid film, or in “dry” conditions, namely entering the recuperator after a pre-humidification and a subsequent removal of the excess water, to avoid problems of aging and corrosion of the plates. The design of the plates geometry for these recuperators should have a threefold aim: firstly, granting mechanical stiffness and easy manufacturability; secondly, enhancing the heat transfer whilst avoiding an excessive increase of the pressure drop along the channels; thirdly, preventing the presence of preferential pathways for water, which may lead to non-uniform wetting of the plates and rivulet flow during “wet” operation. The fulfillment of these tasks requires a detailed analysis of the thermal-fluid dynamics behavior of the air streams inside the channels. Therefore, in this work an experimentally validated numerical Computational Fluid Dynamics (CFD) model has been used to evaluate the effect of the plates geometry on the performance of a cross-flow recuperator for IEC systems. In particular, the study was conducted in “dry” conditions, analyzing various types of protrusions and dimples (with different shape, size, and distribution along the plate), in order to assess one or more configurations able to guarantee the cited improvements.

Experimental Investigation of the Cooling Performance of an Additively Manufactured Prototype for Nuclear Fusion Energy Application

Giacomo Favero, Massimiliano Bonesso, Razvan Dima, Luca Doretti, Adriano Pepato, Giulia Righetti, Simone Mancin

National Institute of Nuclear Physics Italy
University of Padua

Email: giacomo.favero.1@phd.unipd.it

Metal Additive Manufacturing (MAM) is a non-traditional technology recently introduced to manufacture multifunctional mechanical components. In fact, recent developments in the Laser Powder Bed Fusion (LPBF) process have enabled the production of materials characterized by high density and high thermal conductivity properties, such as copper alloys and pure copper, making the technology attractive for thermal science. In nuclear fusion energy applications, mechanical components often encounter extremely high heat fluxes. An innovative solution using unconventional integrated cooling channels is therefore required to safely manage the components. However, the high surface roughness in 3D-printed parts represents an intrinsic limitation of the LPBF technology: the cooling channels show high-pressure drops due to high viscous dissipation generated by the rough surface. To address this challenge, a lab-scale prototype of an e-beam extraction grid for a fusion experiment with an original integrated cooling system was designed and manufactured. Additionally, a novel heating mask was designed and manufactured to reproduce the realistic heat load distribution on the grid during the experimental tests. The prototype was built using additive manufacturing with a CuCrZr copper alloy. The grid underwent heat treatment via solution annealing and age hardening, to increase thermal conductivity from about 100 W m^{-1} K^{-1} to almost 300 W m^{-1} K^{-1}. The prototype was tested at three different constant heat fluxes by varying the water flow rate while measuring the pressure and the maximum temperatures of the grid. A CFD numerical model was also calibrated to estimate the thermo-hydraulic performance of the prototype under test conditions. The experimental and numerical results are presented in terms of overall thermal performance, maximum temperature, and temperature distribution.

Numerical Investigation of Gyroid Heat Exchanger

Arul Prakash Karaiyan, Bhagaban Jena, G. Saravana Kumar, Vipul Dharmendra Choudhari

Indian Institute of Technology Madras

Email: arulk@iitm.ac.in

A numerical analysis of novel cylindrical Gyroid surface-based heat exchanger tubes is carried out to investigate fluid flow and heat transfer characteristics. It is crucial to employ fundamental governing equations like the continuity equation, Navier-Stokes momentum equation, and energy equation to accurately model and analyze the fluid dynamics and heat transfer phenomenon in numerical simulations of heat exchangers. The simulations are carried out using commercially available finite volume based CFD software Ansys Fluent. The numerical validation and grid independence tests are also carried out for the better quality of numerical results. The heat exchanger tubes with gyroid surfaces are manufactured using metal additive manufacturing and a cross-flow heat exchanger is fabricated with a bank of Gyroid tubes. An in-house experimental configuration for testing the gas to gas cross- flow heat exchanger is fabricated and experimental trials were conducted. The numerical results of the Realizable k-ε turbulence model with enhanced wall treatment are validated with experiment results. The results are found be reasonably accurate. The thermal performance of Gyroid tube heat exchanger is also compared with conventional circular tube (CCT) in turbulent regime based on the average Nusselt number, friction factor and thermal performance factor. Parametric analysis is also carried out by varying the gyroid design parameter and an optimum heat exchanger design is proposed which shows a maximum Nusselt number distribution when compared with CCT at the cost of higher internal pressure drop. The results show that due to larger surface area and enhanced fluid mixing, the Gyroid tube based heat exchangers shows better thermal performance compared to conventional circular tube based Heat exchangers. It is concluded that Gyroid tube based heat exchanger will find widespread use in various industrial and scientifc applications where effective heat transfer holds paramount significance.

Multiscale Computational Methods in Bioengineering

Chairs: Alain Kassab, Ryszard Białecki

Assessment of Local Temperature Elevation at the Surface of Tissue Exposed to Radiation of Millimeter Waves Using Simplified Analytical Approach

Dragan Poljak, Josipa Šarić

University of Split

Email: dpoljak@fesb.hr

Dominant effect of human exposure to electromagnetic fields in GHz frequency range corresponding to operation of 5G mobile communications systems is tissue heating, i.e. local temperature elevation at the body surface, i.e. skin, ear and eyes. For the frequencies below transition frequency of 6GHz specific absorption rate (SAR) is used to quantify the volume heating. On the other hand, the surface heating above 6GHz is quantified via absorbed power density (Sab). Once these quantities are determined by means of internal field dosimetry procedures it is possible to determine the local surface temperature increase by solving the bio-heat transfer equation. The present paper deals with the calculation of tissue surface heating due to exposure to millimeter waves. Internal dosimetry is based on the planar tissue model exposed to dipole antenna radiation. Electromagnetic model is based on the Pocklington integro-differential equation numerically handles via Galerkin-Bubnv variant of the boundary element method (GB-IBEM). The assessment of tissue surface heating in planar tissue model has been carried by analytically solving a simplified variant of the bio-heat transfer equation. Some illustrative results for power density and temperature elevations will be presented for different antenna parameters.

On Some Computational Aspects for Electromagnetic-Thermal Dosimetry of MM Waves

Mario Cvetković, Hrvoje Dodig, Dragan Poljak

University of Split

Email: mcvetkov@fesb.hr

The computational dosimetry models are considered important tools in the exposure assessment of humans to high-frequency (HF) electromagnetic fields (EMFs). The improvement of computational methods is very important for the development of exposure standards related to the safe use of EMFs as the experimental validation is very difficult to perform. However, at frequencies in the GHz range, integral equation based formulations are limited due to computational requirements related to problem frequency. The fully populated matrices becomes prohibitively large to solve on typical desktop computers, unless GPU acceleration or parallel processing is employed. At the same time, the penetration depth, i.e. the measure of the depth at which an incident electromagnetic wave penetrates the medium, is limited to several millimetres at frequencies up to 10 GHz, and to micrometers above 30 GHz. Reducing the computational domain of the employed EM model, without necessarily sacrificing the accuracy of the approach, could facilitate the use of integral equation based methods. This work is on the use of a state-of-the-art hybrid boundary element method/finite element method (BEM/FEM) for electromagnetic (EM) dosimetry and the coupled thermal dosimetry model based on the Pennes' heat transfer equation (PHE) for biological tissue solved by means of FEM. The distribution of the induced electric field obtained in both homogeneous and non-homogeneous human head models using EM model is used as a distributed heat source in the piecewise homogeneous human head thermal dosimetry model. As the penetration depth is inversely proportional to the frequency of incident EM wave, we consider the heating depth in several human head models, to illuminate whether homogeneous models are pertinent in the EM part of the model. If confirmed, the results could be found useful in standardisation efforts related to the assessment of human exposure to EM fields in the high frequency range.

Thermal Management System for Cell-Biology Experimental Setup Studying Calcium Signaling in Microgravity Settings

Adina Hochuli, Patrick Estermann, Raphael Kummer, Simon Wuest, Sebastian Ammann, Anastasia Stamatiou, David Schiffmann, Anabel Palacios, Christoph Zumbühl, Maik Böhmer

Luzern University

Goethe-Universitat Frankfurt am Main

Email: adina.hochuli@hslu.ch

The American NASA and the European Space Agency (ESA) have committed to send humans to the Moon and beyond to Mars within the upcoming decades. Such endeavors will only be possible if good health of the crew can be ensured. In this context, calcium, which is an evolutionary agent and important cellular signaling molecule, is thought to play a crucial role. The artificially engineered protein CaMPARI2 is developed to identify elevated intracellular Ca^{2+} concentrations in mammalian and plant cells. When elevated Ca^{2+} concentrations and violet light provided by LEDs coincide the fluorescent protein CaMPARI2 goes through an irreversible photoconversion from green to red. The project in question has the goal to validate the independence from gravity levels of such experiments. This work focuses on a temperature control hardware that ensures uninterrupted temperature stability of the experimental setup in microgravity settings (parabolic flights, drop towers, sounding rockets and centrifuges). A novel, phase change material (PCM)-based system is developed to achieve high temperature stability with high portability, simplicity, and reliability. During the entire experimentation process the sample temperatures need to be stabilized between 30 and 37.5 °C for mammalian cells and between 18 and 22 °C for plant cells. The PCM based thermal management system acts as a heat source when heat supply is interrupted during aircraft loading and unloading times as well as during take-off and landing when the heating element and the LEDs are off. It acts as a heat sink when the LEDs are switched on during the experiment. The thermal management is assessed theoretically by thermal simulations as well as experimentally by the investigation of various materials and structures in detailed lab experiments. By providing constant temperature levels throughout the whole experiment the cells reaction to different irradiation timespans and patterns can be investigated.

Non-Invasive Thermal Measurements During Newborn's Therapeutic Hypothermia and Processing of Their Results

Andrzej J. Nowak, Jakub Tumidajski, Daniel Wagstyl, Dominika Bandola, Mateusz Bojdol, Ziemowit Ostrowski, Marek Rojczyk, Wojciech Walas, Zenon Halaba

Silesian University of Technology
Silesian University of Technology, Poland
University of Opole, Poland

Email: andrzej.j.nowak@polsl.pl

This work reports on the heat transfer processes occurring during the treatment of neonates suffering from hypoxic-ischemic, which is a type of brain dysfunction (brain injury). It happens (mainly during delivery) when the brain experiences a decrease in blood flow (because of abnormal blood circulation) and as a consequence, a decrease in the delivery of oxygen into the cells. It frequently results in an increase in brain temperature to the danger level and hypoxia in some other organs. Treatment requires the application of a specialized cooling device to decrease and control the temperature of the neonate's brain. One of the cooling methods is so-called "selective brain cooling", which consists of direct cooling of the head by using a cooling cap. In this work results of non-invasive measurements for 11 patients suffering from hypoxic-ischemic encephalopathy and treated at the University Clinical Hospital in Opole, Poland. Collected data were processed in a novel way to formulate a whole series of cumulative energy balances for all patients. It is found that cumulative balancing allows one to determine the metabolic heat rate and results in a more stable history of the therapy than balancing step by step. Obtain results were also used to find values of the Thermal Index for 8 patients and demonstrate its correlation with Magnetic Resonance Imaging (with the scale proposed by Weekee) results.

Blood Flow in Deforming Vessels

Ryszard Białecki, Wojciech.Adamczyk@Polsl.Pl, Ziemowit Ostrowski, Bartlomiej Melka, Maria Gracka, Krzysztof Psiuk-Maksymowicz, Damian Borys, Aleksander Sinek, Mateusz Mesek, Jacob Sturdy, Adam Golda

Silesian University of Technology, Poland
Norwegian University of Science and Technology
Gliwice Municipal Hospital no 4

Email: ryszard.bialecki@polsl.pl

Blood flow is most often modeled in vessels of unchanged shape, significantly simplifying calculations and shortening their time. Vascular deformations affect the flow patterns in the arteries, the deposition of atherosclerotic plaques, the formation of aneurysms,and many other phenomena. There are two methods to consider blood vessel deformations: vascular geometry registration by diagnostic imaging such as CT, MRI, or ultrasound or the use of the fluid-structure interaction (FSI) procedure. The first method requires additional procedures for processing image data, and the second requires knowledge of the material properties of the vessel walls, which is a very challenging task for in vivo experiments. The article will show the method to obtain a time-varying geometry of the coronary artery based on images obtained by angioCT. Emerging difficulties result from using images of different resolutions and the need to generate numerical grids of the same topology at every time step. This problem was solved by using image segmentation, smoothing the resulting 3D geometry, and projecting the numerical grid on subsequent shapes of the vessel using diffeomorphism. The resulting velocity fields show the locations of intensive oscillations of the wall shear stress where the atherosclerotic plaque is deposited. The FSI method was used to reproduce the material properties of the carotid artery based on measuring the deformations of the artery wall with ultrasound and blood pressure measured with an applanation tonometer. In both cases, the medical simulations were preceded by the validation of numerical methods, using fantoms created for this purpose, in which cyclic deformations of flexible vessels were measured, and pressures and flows were recorded

Keynote Lecture: In-Vitro Analysis of the Novel Self-Powered Fontan Circulation via a POD-Trained RBF Interpolation Network

Arka Das, Ray Prather , Purdy Clayton, Keyu Vadaliya, Anthony Damon, Levi Blumer, Martin Cinelli, Eduardo Divo, Alain Kassab, William Decampli

Embry-Riddle Aeronautical University
State University System of Florida
Orlando Health Arnold Palmer Hospital for Children

Email: dasa@erau.edu

Around 8% of all newborns with a congenital heart defect have only a single functioning ventricle (SV). The Fontan circulation is a result of the third stage surgical procedure to correct the SV anatomy in these patients. Despite successful implementation over the years, this altered circulation is prone to failure with survival rates of only 50-80% to adulthood. Increased inferior vena caval (IVC) pressure plays a significant role in “Fontan failure.” We propose to augment energy in the Fontan circulation with an injection jet shunt (IJS) drawing flow directly from the aortic arch, balanced by a conduit-to-atrial fenestration to approximately preserve the ratio of pulmonary flow (Qp) to systemic flow (Qs). The basic concept involves the injection of a high-velocity jet in the direction of flow causing flow entrainment, leading to a significant reduction of upstream (i.e., IVC) pressure and enhancement of downstream flow.In this study, we have utilized proper orthogonal decomposition (POD) trained Radial Basis Function (RBF) interpolation network to develop a supervisory control algorithm to dynamically control a benchtop Mock-Flow loop (MFL) to validate our hypothesis by cross-validating the encouraging multiscale computational results. The “Hardware-in-the-Loop” (HIL) technique has been utilized to embed the POD-RBF framework based supervisory controller into the MFL setup on a real-time basis. This automated controller has been developed to maintain the Qs constant by modulating the ventricular preload and the systemic vascular resistance to mimic circulatory response in the presence of shunting IJS flow. The MFL replicates a reduced four-compartmental lumped parameter model (LPM) of the Fontan circulation, and it is integrated with a patient generic 3D phantom of the IJS-assisted Fontan with average dimensions matching those of a 2–4-year-old patient. Experimental results are presented and compared to the computational findings on the hemodynamic results and oxygen saturations for various total cavopulmonary configurations.

A Needle-Form 3-Omega Sensor for Thermal Characterization of Cryopreserved Biological Tissues

Spencer Alliston, Chris Dames

State of California
UC Berkeley

Email: spencer.alliston@berkeley.edu

Thermal properties of cryopreserved tissues are of critical importance to the biopreservation community, which continues to seek more effective ways to store biological samples for improved outcomes in organ transplants as well as to facilitate the preservation of a record of biodiversity. Here, we present a reusable thermal needle-type 3-omega method designed for in situ characterization of such tissues, as well as other soft materials. The 3-omega method is a classic thermal materials characterization technique, which has been integrated into a modified microfabricated neural probe. This enables the measurement to be robust to environmental and experimental factors in cryopreservation. We demonstrate the viability of such a sensor to concurrently measure heat capacity and thermal conductivity for amorphous and crystalline solid samples of water and tissues, with parallel phase verification by polarized light macroscopy. These measurements can also be used for differentiation of solid samples, which is of particular interest for studies involving the kinetic limits of amorphous solidification (vitrification). We also explore the potential for detection of 1st and 2nd order phase transitions, as well as other metastable properties of interest. In doing so, we demonstrate the value of a packaged thermal sensor to advancing the thermal understanding of cryopreserved biological systems and other solid-liquid phase change systems.

Solidification Science and Technology

Chairs: Božidar Šarler, Miha Založnik

Nucleation Study of Xylitol as PCM in Two Different Crystallization Systems: A Rheometer and a Lab-Scale Prototype.

Miguel Navarro, Ana Lazaro, Monica Delgado, Gonzalo Diarce Belloso

University of Zaragoza
University of the Basque Country-ENEDI

Email: miguel.navarro@unizar.es

Sugar alcohols are promising materials for latent thermal energy storage. Among them, xylitol stands out for its large energy storage density; however, like other sugar alcohols, it shows supercooling and a low crystallization rate, which hinder its potential as a phase change material. This work studies the effect of the seeding and shearing technique to promote the crystallization and, therefore, the latent heat release of the material. In addition, the effect of the operation conditions such as temperature or shear is assessed. The work is based on two different set-ups: a rheometer and a stirred tank. In the results, the induction time (time elapsed until re-crystallization occurred) is calculated from the temperature and viscosity curves, while a crystallization model is employed to analyze the influence of the operating conditions in the crystallization behavior. Results show that both undercooling and stirring velocity in the lab-scale prototype have an influence on crystallization.

Intelligent Casting Based on Hollow Mold and Closed Loop Cooling Control

Jinwu Kang, Baolin Liu, Jingying Xu, Xiaolong Wang, Jiwu Wang

Tsinghua University
University of Science and Technology Beijing
Beijing Jiaotong University

Email: kangjw@tsinghua.edu.cn

Casting is a main manufacturing method. The solidification process is the key procedure, so, the intelligence of casting should focus on the solidification process. Instead of using robots and automatic control during the molding, core shooting, pouring and fettling procedures, in this paper, an intelligent casting concept was proposed by the closed loop control of the solidification process. A new design of mold structure—hollow mold was presented, so as to increase the response of casting to the external cooling and provide space for embedding sensors and actuators in mold. Hollow mold structure includes a layer of shell to form the cavity to hold the liquid metal for casting forming, support structure, and other functional structures, such as multi shells to realize heat insulation effect. Meanwhile, a hollow mold design method was established with the coupling of surface discretization, spatial discretization and numerical simulation. The cooling control system was designed to integrate with the hollow mold to form the intelligent mold. It consists of sensors, such as temperature, displacement, movement sensors, actuators such as cooling nozzles, and central control unit to dispose the feedback from sensors and then give signals to actuators. By the intelligent mold system, the cooling of casting can be controlled at full scope and during the whole process, for example, the forced air cooling nozzle can start, suspend and stop at any required time, and a series of cooling nozzles can be placed around the mold corresponding to interest cooling points of the casting. Cases were provided to demonstrate and validate this intelligent casting idea.

On the Occurrence of Rapid Solidification in Additive Manufacturing of Metallic Alloys

Damien Tourret

IMDEA Materials

Email: damien.tourret@imdea.org

The emergence of fusion-based additive manufacturing (AM) processes for metals and alloys – often involving high solidification rates – has renewed the general interest in rapid solidification. This regime is broadly defined by a significant kinetics-induced departure of the solid-liquid interface from thermodynamic equilibrium – as dictated, for instance, by the alloy phase diagram. Rapid solidification phenomena include solute trapping, kinetic undercooling, and morphological microstructural changes (e.g., planar interface restabilization or oscillatory "banding" instability). Yet, while cooling rates in metal AM are often high, it is usually ambiguous whether solidification actually occurs in a rapid solidification regime, as definite proofs of solute trapping, kinetic undercooling, or morphological transitions (if any) are scarce. In this talk, we will briefly review the underlying mechanisms of rapid solidification, and question whether and when metal AM can be considered a rapid solidification process. We will base our discussion on classical theories, as well as computational (e.g., phase-field) simulations, compared to experimental data from various collaborations. These include (1) laser spot-melting experiments of ternary Ni alloys imaged in-situ using synchrotron X-ray radiography, which do not show any definitive trace of rapid solidification despite the high measured solidification velocities (Acta Materialia 250, 118858, 2023), as well as (2) microstructures observed in biomedical-grade Mg alloys printed using laser powder-bed fusion, which exhibit banded microstructures that clearly indicate a rapid solidification regime (Materials Science & Engineering C 119, 111623, 2021).

Long-Distance Settling Simulation of Equiaxed Dendrite by Moving-Frame Method: Phase-Field Lattice Boltzmann Study with Parallel-GPU AMR

Shinji Sakane, Tomohiro Takaki

Kyoto Institute of Technology

Email: sakane@kit.ac.jp

In large ingot castings, a distinct cone-shaped negative segregation often arises in the lower region due to movement of equiaxed dendrites influenced by buoyancy from the solid-liquid density ratio. Understanding kinetic behavior of equiaxed dendrites is crucial for accurately predicting such macro-segregation. The phase-field (PF) method is one of the most accurate approach for quantitatively simulating dendrite growth. Although coupling PF method with Navier-Stokes (NS)/lattice Boltzmann (LB) equations enables simulations of dendrite growth with melt convection and solid motion, mostly limited to two-dimensional setups due to huge computational costs. Our group developed a high-performance computing method, parallel-GPU AMR, combining adaptive mesh refinement (AMR) with multiple GPU implementations. This novel approach allows quantitative evaluation of three-dimensional equiaxed dendrite settling due to the solid-liquid density ratio using the PF-LB method. However, settling distances are limited to a few millimeters corresponding to the domain size, and the dendrite contacts the domain bottom before rotation begins. In this study, we introduce a moving frame method to PF-LB simulation with parallel-GPU AMR, tracking dendrite movement by shifting the computational domain. This enables simulating long-distance settling of equiaxed dendrites without computational domain size constraints. Using this method, we assess the impact of motion behavior, including dendrite rotation, on growth behavior.

Multi-Phase-Field Lattice Boltzmann Simulations of Semi-Solid Simple Shear Deformation in Thin Film

Namito Yamanaka, Shinji Sakane, Tomohiro Takaki

Kyoto Institute of Technology

Email: d2821004@edu.kit.ac.jp

During the casting process, semi-solid deformation due to external forces and/or melt flow induces grain rearrangement. This results the formation of localized regions with low solid fraction. These regions have a direct impact on macrosegregation or band segregation. Therefore, understanding the semi-solid deformation behaviour is crucial. According to experiments, semi-solid deformation significantly differs depending on the solid fraction and grain morphology. Moreover, a thin semi-solid material was observed in situ, and it was confirmed that local dilatancy caused by grain–grain contact and rearrangement results in changes in the local solid fraction of the semi-solid material. Recently, 3D observation of semi-solid deformation became possible, and it was confirmed that grain rearrangement behaviour due to the compression of semi-solid material leads to localized solid fraction changes. Numerical simulations were performed to gain more detailed understandings of semi-solid deformation by using discrete element method. It has been demonstrated through simulations that the solid fraction, grain morphology, and deformation rate have a significant impact on semi-solid deformation behaviour. Also, in a previous study, we successfully simulated semi-solid deformation using the multi-phase-field lattice Boltzmann method and performed a continuous simulation from polycrystalline solidification to semi-solid deformation. However, these simulations are limited in 2D, thus the fluid flow cannot express accurately. In this study, to investigate grain rearrangement behaviour, the simulation method was expanded to 3D.

Phase Field Simulation of Solidification in High Entropy Alloys

Alexandre Viardin, Markus Apel, Thomas Bähr, Anna Schönell

Access e.V.

Email: a.viardin@access-technology.de

High-entropy alloys are promising candidates for producing coatings with competitive performance while making little or no use of toxic or critical materials. To achieve this goal, investigating the solidification behavior of Cantor alloys is crucial for understanding the influence of alloy composition on microstructure evolution.Cantor alloys are composed of iron, cobalt, chrome, manganese, and nickel. We have conducted phase field simulations to comprehend the effect of concentration on solidification behavior and final morphologies. The results provide information on the solidification range and microsegregation. The distribution of elements in the microstructure has an impact on corrosion behavior.

Towards a Macroscopic Model of Diffusive Grain Interactions During Equiaxed Dendritic Solidification of Metal Alloys

Miha Založnik, Abdelhalim Chirouf, Ba, Alphonse Finel, Yann Le Bouar

CNRS
Jean Lamour Institute/ONERA
National Centre for Scientific Research

Email: miha.zaloznik@univ-lorraine.fr

Solidification is a key step in the processing chain of most metal alloys ; it occurs during casting, additive manufacturing, welding, etc. Dendritic crystal grains are the most common growth form in solidification of metal alloys. The growth of equiaxed dendritic grains during the solidification of a liquid alloy is accompanied by the ejection of chemical species (solutes) into their environment. Nearby grains interact via the solute diffusion field in the liquid phase. These interactions depend to a large extent on the spatial arrangement of the grains. The diffusive interactions do not only affect the shape and the size of grains but they also control the overall kinetics of the phase transformation. In current macroscopic mean-field models the description of the spatial arrangement is entirely neglected. The diffusive interactions are thus overly simplified and the power of these models to predict of the solidification kinetics is impaired. In this study we investigate diffusive grain interactions during equiaxed solidification by full-field simulations of a representative elementary volume (REV) containing an ensemble of grains. The simulations are done with the Grain Envelope Model (GEM) for different periodic and random arrangements of grains. We compare the full-field results to state-of-the-art mean-field models and we point out the limitations of the latter. We propose an analysis of the interactions of individual grains with their local neighborhood, defined by a Voronoi tessellation. Based on this analysis we propose an improved macroscopic model that includes a description of the spatial arrangement of grains.

Effect of Marangoni Flow During the Solidification of a Fe-0.82wt.%C Steel Alloy

Ibrahim Sari, Menghuai Wu, Kharicha Abdellah

Montanuniversitat Leoben
University of Leoben
CDLAB MHD Montanuniversität

Email: ibrahim.sari@unileoben.ac.at

A two-phase Mixture model is proposed to simulate the liquid-solid phase transition of a Fe-0.82wt%C steel alloy under the effect of Marangoni flow. This model simplifies computations by solving a single momentum and enthalpy equation for the mixture phase using a three-dimensional finite volume method. The simulation involves solidifying a rectangular ingot (100 × 10 × 100 mm) from the cold bottom surface towards the hot-free surface at the top. To facilitate heat exchange with the surrounding environment, a high heat transfer coefficient of h = 600 $W/m^2/K$ was applied on the bottom surface to establish an upward solidification direction. However, a lower heat transfer coefficient of 20 $W/m^2/K$ was applied on the top free surface, which was considered flat. This study aims to examine the effect of Marangoni flow generated by surface tension on flow and segregation patterns. The results show that the Marangoni flow emerges at the free surface and penetrates into the liquid depth, leading to the formation of hexagonal patterns along the liquid thickness. Upon full solidification, macro-segregation also exhibits hexagonal structures, mirroring the stationary hexagonal shapes generated by Marangoni flow.

Influence of the Coriolis Force on the Solidification of a Fe-0.82wt.%C Alloy

Ibrahim Sari, Menghuai Wu, Kharicha Abdellah

Montanuniversitat Leoben
University of Leoben
CDLAB MHD montanuniversität

Email: ibrahim.sari@unileoben.ac.at

A three-dimensional multiphase numerical simulation model of the solidification of Fe-0.82wt%.C steel alloy under the effect of thermosolutal convection during Horizontal Centrifugal Casting (HCC), will be presented. The studied system is in a rotating frame of reference, rotating with constant angular speed in a counter-clockwise direction around the x-axis of the original frame with r = 1 m radius of rotation. However, the location of the rotating reference frame must be accurately determined. This suggests solving the flow in the rotating frame of reference i.e. taking into account fictitious forces such as the Centrifugal and Coriolis forces. The purpose of this work is to study the effect of the Coriolis force on centrifugal buoyancy-driven convection in a rotating geometry. Indeed, a steel molten metal is contained in a rectangular cavity initially at a temperature above the liquidus, with an inner hot wall and an outer cold wall. Considering cooling from the bottom wall by applying a slow cooling rate of CR = 0.1 K/s which allows the occurrence of the phase transformation as the temperature drops below the liquidus. However, the liquid-free surface (molten metal and air interface) on top. Nevertheless, the free surface was considered flat. The activities include convective heat transfers and solidification in rotating geometry layer of liquid. The results show that the Coriolis force breaks the thermosolutal convection, having an important impact on the final macro segregation pattern.

Effects of the Asymmetric and Oscillating Turbulent Melt Flow on the Heat Transfer and Solidification Inside the Thin Slab Continuous Casting Mold Under the Applied Electromagnetic Brake

Alexander Vakhrushev, Kharicha Abdellah, Menghuai Wu, Yong Tang , Gernot Hackl , Josef Watzinger , Ebrahim Karimi Sibaki

University of Leoben
CDLAB MHD Montanuniversität
RHI Magnesita GmbH Austria
Primetals Technologies Austria

Email: alexander.vakhrushev@unileoben.ac.at

Continuous casting (CC) is nowadays the world-leading technology for steel production. The thin slab casting (TSC) is featured by a slab shape close to the final products, which are casted at a high speed with the fast solidification rate. The quality of the thin slabs strongly depends on the uniformity of the turbulent flow, the super-heat distribution, and the growth of the solidified shell. The electromagnetic brake (EMBr) is applied to control the highly turbulent flow after the fresh melt is fed through the ports of a submerged entry nozzle (SEN). Numerical modelling is a perfect tool to investigate the multiphase phenomena of the turbulent flow in the CC mold, heat transfer and solidification coupled with the effects of magnetohydrodynamics (MHD). Traditionally the heat transfer in the CC mold simulations is predefined by the heat flux profile. In this case the heat extraction is restricted, and the influence of the transient flow behavior is excluded. The presented study considers the coupled heat transfer through the solid shell into the water-cooled copper mold, including the averaged thermal resistance of the slag skin and the air gap with an aim to incorporate the flow effects from the liquid pool side. The turbulent flow and heat transfer are coupled with the MHD model using an in-house code developed inside the open-source CFD package OpenFOAM®. The model is applied to investigate different undesired solidification issues related to the asymmetric melt flow with the (i) partially blocked or (ii) misaligned SEN or caused by the mean jet oscillations (iii) for the specific SEN designs or (iv) related to the natural flow frequencies. The variation of the flow pattern and the solid shell thickness is studied for different scenarios both with and without applied EMBr.

On the Conditions for the Occurrence of Crystal Avalanches During Alloy Solidification

Golshan Shayesteh, Andreas Ludwig, Mihaela Stefan-Kharicha, Menghuai Wu, Abdellah Kharicha

Montanuniversität Leoben
CDLAB MHD Montanuniversität
University of Leoben

Email: ludwig@unileoben.ac.at

Experimental studies on the solidification of ammonium-chloride-water alloys in relatively large containments reveal conditions that lead to the formation of numerous crystal avalanches. Columnar segments that occasionally slide downwards along the vertical mushy zone further fragmentate and so crystal multiplication occurs. As a condition for this phenomena solidification-induced solutal buoyancy that leads to a rising interdendritic flow was identified. The interaction with sedimentation-induced downward flow ahead of a vertical columnar region results in a redirection of the interdendritic flow and thus, to local conditions that slow down further solidification or even lead to remelting. Gravity is then pulling loose segments downwards. In larger containment, the flow in the bulk melt is generally unsteady and even turbulent. Thus, the outlined flow-solidification/melting interplay happens frequently at numerous positions but in an unpredictable way.

Determination of Transient Heat Transfer by Cooling Channel in High-Pressure Die Casting Using Inverse Method

Jan Bohacek, Krystof Mraz, Jiri Hvozda, Alexander Vakhrushev, Ebrahim Karimi Sibaki

Brno University of Technology
University of Leoben

Email: xmbohac02@vutbr.cz

Complex shapes of aluminum castings are typically manufactured during the short cycle process known as the high-pressure die casting (HPDC). High productivity is ensured by introducing die cooling through a system of channels, die inserts or jet coolers. Die cooling can also effectively help in reducing internal porosity in cast components. Accurate simulations based on sophisticated numerical models require accurate input data such as material properties, initial and boundary conditions. Although the heat is dominantly dissipated through die cooling, indicating the importance of knowing precise thermal boundary conditions, open literature lacks a detailed information about the spatial distribution of heat transfer coefficient. This study presents an inverse method to determine accurate heat transfer coefficients of a die insert based on temperature measurements in multiple points by 0.5 mm K-type thermocouples and a subsequent solution of the two-dimensional inverse heat conduction problem. The solver was built in the open-source CFD code OpenFOAM and the free library for nonlinear optimization NLopt. The results are presented for the commonly used 10 mm die insert with a hemispherical tip and coolant flow rates ranging from 100 l/h to 200 l/h. Heat transfer coefficients reach values well above 50 kW/m^{2}K in the hemispherical tip, which followed by a secondary peak gradually drop to values around 1 kW/m^{2}K further downstream.

A New Hybrid Local Radial Basis Function Collocation Method for 2.5D Thermo-Mechanical Modelling of Continuous Casting of Steel

Gašper Vuga, Tadej Dobravec, Boštjan Mavrič, Božidar Šarler

Faculty of Mechanical Engineering, University of Ljubljana
Institute of Metals and Technology, Ljubljana, Slovenia

Email: gasper.vuga@fs.uni-lj.si

This study presents a new strong-form meshless method to solve the thermomechanical problem of the solidification process in the continuous casting of steel. A two-dimensional slice with zero thickness that travels in the casting direction is modelled in the Lagrangian system. The developed mechanical model is one-way coupled to the thermal model, where the heat fluxes from the mould, sprays, and rollers are imposed to solve heat conduction in the strand, where the convective effects in the melt are heuristically included in the enhancement of the thermal conductivity in the melt. The resulting temperature and metallostatic pressure govern the Norton-Hoff visco-plastic model used for computing shrinkage of the solid shell and induced residual stresses. The results are used to estimate critical areas susceptible to hot-tearing formation. The mechanical model uses a generalised plane strain assumption that includes linear strains perpendicular to the slice and enables the computation of the bending of the strand. The thermo-mechanical model is spatially discretised with a local radial basis function collocation method (LRBFCM). The mechanical part includes a new hybrid method that combines LRBFCM with classical finite differences and increases the method's stability. The presented work investigates the impact of process parameters such as casting velocity on the solid shell shrinkage and the probability of hot-tearing occurrence in the continuous casting of square billets.

Thermal Control Devices and Thermal Circuits

Chairs: Andrej Kitanovski, Miguel Muñoz Rojo

Experimental and Pore Scale Numerical Characterization of Multi-Mode Heat Transfer in Porous Ceramics Exposed to a Transverse Heat Flux

Alexandre Briclot, Pierre Lea, Clemens Suter, Jean-François Henry, S H, Jaona Harifidy Randrianalisoa

University of de Reims Champagne-Ardenne
ETH-Rat

Email: briclotalexandre@gmail.com

This communication presents experimental and computational approaches to characterize the multimode heat transfer in an open-cell Silicon-Silicon Carbide foam exposed to a transverse heat flux. The experimental setup consists of a 5cmx5cmx10cm parallelepipedal 10ppi foam (80% porosity) placed within a Ca_2SiO_4 rectangular channel, in which airflow was established. To mimic the heat source for waste heat recovery in industrial systems, an electrically heated aluminum plate was mounted in contact with one side of the foam so that the heat flux is perpendicular to the flow direction. The pressure drop across the foam was measured and infrared cameras were used to obtain the temperature of the heater plate and the thermal maps of the foam at the outlet and on the top. Air temperature at 81 uniformly distributed points over a 5cmx5cm plane at the channel outlet was measured with K-type thermocouples. The computational approach is a 3D pore scale model of flow and multi-mode heat transfer in the channel subjected to the same experimental conditions. A X-ray tomography technique (50µm resolution) was used to scan the foam structure. The whole volume was then reconstructed and meshed using Avizo and GMSH software. The computational geometry consists of 1000x1000x2000 voxels and about 101 million cells after meshing. The computations were performed using ANSYS Fluent. Laminar or k-ε model with enhanced wall treatment were used depending on the airflow rate. Firstly, comparing the experimental and the numerical temperature distributions at the foam outlet and the temperature jump at the foam/heater interface at different heating powers and when no flow was applied, the foam thermal conductivity and the interfacial thermal resistance were retrieved. Then, the numerical model was employed to determine the heat exchange efficiency, the pressure drop properties and local velocity and temperature fields. Finally, numerical results were compared with experimental data.

Thermal Control of Thin Films with Nanoscale Structure

Moeka Sato, Yoshiya Takahara, Mitsuhiro Matsumoto, Nobuhiko Kajinami, Misaki Hanaoka, Manabu Iwakawa

Kyoto University

Mitsubisishi Electric Corporation

Email: sato.moeka.37a@st.kyoto-u.ac.jp

Thermoelectric energy conversions have been attracting much attention, which directly generate electric energy from thermal one owing to the Seebeck effect. Among various efforts to improve the conversion efficiency, control of phonon propagation with nano-scale structures are very popular, such as super-lattices and nano-dots, which utilize phonon scatterings on interfaces. The basic concept is the difference of mean free path between phonons and electrons (i.e., charge carriers). In typical cases with silicon-base devices, the mean free path of phonons is in an order of 100 nm while that of electrons is 1-10 nm. Thus structures of 10-100 nm size are expected to effective for suppressing the phonon heat transfer without much reducing the electric transport, leading to thermoelectric conversion efficiency. We have developed a numerical scheme to investigate phonon transport based on the Boltzmann transport equation (BTE) for time development of the phonon distribution function in a reciprocal space with the conventional relaxation time approximation. The point of our scheme is utilization of a VOF (volume of fluid) like boundary conditions in order to treat an arbitrary shape of nano-scale structures. In the presentation, we will show results of several test systems of nano-structured Si thin films, combined with evaluation of effective electric conductivity, and discuss how such nano-scale structures improve the conversion efficiency.

Fast Thermodynamic Simulation of Electrical Control Cabinets via Thermal Resistor Networks

Daniel Haag, Prof. Dr.-Ing. Konstantinos Stergiaropoulos

University of Stuttgart

Email: daniel.haag@igte.uni-stuttgart.de

To prevent temperature-related failure or malfunctioning of electronics in control cabinets and simultaneously decrease the energy demand of the cooling system, thermodynamic modeling and simulation are crucial. A fine-grid Computational Fluid Dynamics (CFD) simulation featuring heat transfer and buoyancy provides an accurate and detailed prediction of the temperature and velocity field inside a control cabinet. But due to the high computational demand of a single calculation, this approach is not eligible for a widespread use in design and optimization processes. This paper presents a novel temperature node-based approach for the thermal and energetic modeling of control cabinets that enables fast predictions of zonal and component temperatures and the energy demand of the cooling system. Heat transfer between nodes is modeled by means of the thermal-electrical analogy. Depending on the layout of the control cabinet and the setup of the cooling system, an individual thermal resistor network is created. Due to the network design algorithm, at most one connecting resistor exists between two nodes. Therefore, the enforcement of energy conservation over all nodes always yields a computationally inexpensive solvable system of linear equations. Temperature dependence can be considered through an iterative solving procedure. Resistors inside the network are classified into convective (surface-bound) and flow elements. Convective resistances are calculated with Nusselt-Number-Correlations. To approximate Nusselt-Numbers and flow resistances between nodes, an isothermal and incompressible coarse-grid CFD simulation is performed and mapped to the network. The method is demonstrated in a two-dimensional domain that encompasses the internal air volume and surfaces. Its capabilities are assessed by comparison to fine-grid CFD simulations. Through the parametrization of a block-structured background mesh, the geometric resolution of the resistance network and its associated coarse-grid CFD model are adaptable. The achievable speedup and level of accuracy are investigated for a variety of randomly generated control cabinet configurations.

Conductive Thermal Transistor with Insulator to Metal Phase Transition Materials

Karl Joulain, Younès Ezzahri

National Center for Scientific Research
University of Poitiers

Email: karl.joulain@univ-poitiers.fr

We show in this article that it is possible to make a simple conductive thermal transistor with the help of materials exhibiting an insulator to metal transition at a certain temperature. Our transistor is constituted of a collector and an emitter made of a good thermal conductor, each of them being linked to a thermal reservoir at a constant temperature. The third part of the transistor is the base also constituted of a good thermal conductor. Two very thin layers made of a material exhibiting an insulator to metal transition separate the collector from the base on the one hand and the base from the emitter on the other hand. By managing the base input thermal current, we show that it is possible to modulate and to amplify the thermal current flowing from the collector to the emitter. In order words, a small change in the base thermal current provokes a change from insulator to metal in the collector-base and base-emitter separating layers leading to a strong increase of their thermal conductance and consequently a large increase in the thermal current crossing the structure. The base thermal current is therefore amplified into the emitter (collector) thermal current. Several configurations are studied: separating layers made of the same materials or of different ones. We show that amplification can be obtained in both configurations and that an amplification ratio of several tens can be achieved. We think that this solid state thermal device could lead to important applications in several thermal management systems.

Keynote Lecture:
Thermal Control Devices and Thermal Circuits

Katja Klinar, Andrej Kitanovski

University of Ljubljana

Email: katja.klinar@fs.uni-lj.si

It is becoming evident that conventional thermal management methods like conventional thermal insulation and conventional thermal storage cannot meet the thermal control requirements of advanced, especially small, systems with higher power densities or potentially transient, fluctuating, or migrating hot or cold spots, and for temperature sensitive devices. This challenge is most evident in electronic components, which experience degradation and loss of efficiency without constant and effective heat dissipation. To overcome these limitations, thermal control devices have emerged in various thermal management fields. These small-scale devices enable nonlinear, switchable, and active control of heat, similar to how electrical counterparts regulate electrical current. Among others, notable thermal control devices include thermal conduits (acting as solid-state heat routers), thermal resistors (providing thermal insulation), thermal switches (actively managing heat transfer through on-off states), and thermal diodes (rectifying heat currents). In this paper, we provide state of the art on the research activities and applications thermal control devices, as well as an overview of various mechanisms that have been applied in different applications; i.e. in caloric refrigeration, space and cryogenic applications, thermal management of buildings, and others. Based on this, we provide advantages and disadvantages and finally, the guidelines and target characteristics for future research activities in order to reduce the response time and/or increase the performance and/or energy efficiency.

Thermal Control Circuit Modelling with TCCbuilder

Katja Vozel, Katja Klinar, Nada Petelin, Kitanovski Andrej.Kitanovski@Fs.Uni-Lj.Si

University of Ljubljana

Email: katja.vozel@fs.uni-lj.si

Thermal control elements (TCEs) and thermal control circuits (TCCs) are a promising complement to conventional thermal management methods because they provide nonlinear, switchable, and active control of heat. However, TCEs and TCCs are still at an early stage of development and are not yet ready for use in commercial applications. Most of the works on TCEs focus on a single TCE and mainly look for suitable materials and geometries to achieve better performance. Some concepts have not yet been properly analysed, e.g., the characteristics of combining TCEs in thermal control circuits (TCCs) and the transient operation of TCEs and TCCs. We took a step forward by making an open-source TCC simulation tool, TCCbuilder, which will be available to a broader community. TCCbuilder enables the design of TCEs and TCCs and offers the possibility to analyse their behaviour in (quasi-)steady-state and in transient operation. This article describes how to design and simulate a TCC using TCCbuilder. TCCbuilder allows the design and testing of custom TCCs by either using various TCEs available in the TCCbuilder's library or by creating new TCEs with specific properties. The tool has an extensive library of materials and TCEs (thermal switches, thermal diodes, etc.) compiled from extensive literature searches and contributions from fellow researchers. TCCbuilder's library of materials provides input data (e.g., temperature-dependent thermal properties) for simulations. However, the user can also enter their own data. The simulation results (system properties and time evolution of the system with respect to temperature) are exported to a text file in a format that can be imported into standard editors for further analysis. The tool and its library can be accessed at www.tccbuilder.org. It is expected that this platform will revolutionize thermal management practices and facilitate innovation in TCC design and optimization.

Riding the Thermal Wave: A Review of Materials for Thermal Switches

Katja Klinar, Katja Vozel, Andrej Kitanovski

University of Ljubljana

Email: katja.klinar@fs.uni-lj.si

Thermal switches are becoming increasingly popular in thermal management applications, especially in systems dealing with cyclic temperature differences. Thermal switches allow heat flow when on and limit it when off. Activation and deactivation are triggered by external stimuli, such as changes in magnetic or electric fields, applied force, pressure, light, or combinations thereof. Factors such as switching ratio, response time, and energy efficiency are usually considered when evaluating thermal switches. This review addresses the ongoing search for best materials for thermal switches used in solid, liquid, and gaseous phases. In addition to the current state of the art, emerging materials are also covered. Besides advantages and disadvantages of the various materials and the problems associated with the design and testing of such thermal switches, insight is also provided into future design guidelines aimed at producing the thermal switches for systems with cyclic temperature differences.

Modeling the Effect of Thermal Contact Resistance on the Conduction of 3D Fibrous Materials in Relation to Fiber Properties

Clémence Gaunand, Yannick De Wilde, Adrien François, Veneta Grigorova-Moutiers, Karl Joulain

National Center for Scientific Research
ESPCI Paris, University PSL, CNRS, France
Saint-Gobain Research Paris - Aubervillers (France)

Email: clemence.gaunand@espci.fr

Reducing energy costs in the building sector drives important efforts to improve the performance of insulation materials such as glasswool. Understanding heat transfers in this fibrous material, particularly conduction, is a major challenge due to its heterogeneous and multiscale nature, and the unknown contribution of fiber-to-fiber contacts. Indeed, in most established numerical models for conductive fibrous media, either only 2D materials are represented, or contacts specificities are ignored. In addition, 3D modeling imposes strong restrictions regarding the number of fibers which can be included in the simulation, and thus restricts the size of the simulated sample. In this work, we describe a nodal method, based on a 1D equation solver, to analyze the steady-state heat conduction through the solid fibrous structure (i.e., the solid conduction) in numerically generated 3D fibrous networks and to compute the corresponding effective thermal conductivity. We study the influence of four fiber parameters, namely, density, orientation, length and diameter, with a particular focus on their joint effect with the thermal contact resistance between fibers. We find that the impact of fiber dimensions on the conductive properties of these networks is quite complex and depends strongly on the value of the thermal contact resistance between fibers. We explain these findings with qualitative arguments based on the evolution of the contact density. When applied to the specific case of glasswool insulation mats, our numerical approach shows that contacts become significant when the individual thermal contact resistance exceeds 107K/W. This indicates an order of magnitude of the threshold at which the contact resistance begins to dominate the thermal resistance of fibers in the fibrous network. This modeling study may enable predictive modeling and optimization of fibrous insulation materials, based on the exploitation of a regime where contacts dominate the thermal resistance.

Oscillating Thermal Switch Using Electrostatic Forces

Nada Petelin, Borut Pečar, Danilo Vrtačnik, Urban Tomc, Kitanovski
Andrej.Kitanovski@Fs.Uni-Lj.Si

University of Ljubljana

Email: nada.petelin@fs.uni-lj.si

Thermal control devices such as thermal switches, thermal diodes, thermal regulators, and thermal capacitors can control the intensity and direction of heat flow, which has proven useful in various thermal management applications. Their operation is analogous to that of their electrical counterparts, where temperature differences correspond to voltage differences and heat flow to electric current. Here, we design and fabricate a thermal switch that controls heat flow from the heat source by changing the ON and OFF thermal conductance. The thermal switch uses electrostatic forces for actuation and makes thermal contact between the heat source and the heat sink when in the ON state, and breaks contact when in the OFF state. The thermal switch is constructed using the commercially available materials. The heat sink and heat source are made of silicon, and the active part of the heat switch is a flexible gadolinium sheet with a thickness of 180 µm. The proof-of-concept device was evaluated for a magnetocaloric application; however, the proposed thermal switch can be used in any thermal circuits for fast thermal regulation and advanced thermal control.

Keynote Lecture:
A Brief Review of Applications of Thermal Diode

Baowen Li, Guimei Zhu

Southern University of Science and Technology, China

Email: libw@sustech.edu.cn

It has been 20 years since we propose the thermal diode in 2004 (Phys. Rev. Lett 93, 184301 (2004). Many important progress have been made both in fundamental research and applications such as heat control and management. In this talk, I will give a brief review of the most important discoveries and applications based on the thermal diode, for example, thermal transistor, thermal logic gate, thermal memory etc. Specially emphasise will be given on the applications of solid state cooling. It is reported both theoretically and experimentally that the application of thermal diode in elastocaloric can enhance significantly the COP of the cooling.

Contact Resistance Effects on Current Path and Thermal Characteristics of Ag-Nanowire Network

Kazuya Tatsumi, Yuta Sugihara, Kanji Tamai, Reiko Kuriyama

Kyoto University

Email: tatsumi@me.kyoto-u.ac.jp

A sheet composed of Ag-nanowires (Ag-NWs) which is randomly dispersed and forms a network pattern is applied to electronic devices and wearable devices as a flexible and transparent conductive film and heater. To evaluate the performance and reliability of the sheet and device, it is necessary to understand the characteristics of the current and temperature distributions of the Ag-NW network. The current and heat transfer paths are based on the dispersion pattern of the Ag-NWs, and the electrical and thermal resistances of the Ag-NWs and the contact point at the Ag-NW junction. These resistances vary depending on the surface properties of the Ag-NW and the contact conditions at the junction. Although, the contact electrical resistance at a single point of two Ag-NWs has been investigated, the variation of the resistance and its effect on the current and temperature distribution of the Ag-NW network are unknown. In this study, we present a novel method which can obtain the distributions of the current and contact resistance of the junction in the Ag-NW network by combining the two-dimensional temperature measurement using the thermoreflectance imaging and numerical simulation. The effects of the variation in the contact resistance and the number density of Ag-NWs on the current and temperature distributions were investigated and analyzed using modified statistical and stochastic models to provide insight into the design and reliability analysis of the devices and sheets using the Ag-NW network. The results showed that the contact electrical resistance varied over the Ag-NW network at certain magnitude, which significantly affected the temperature distribution by defining the current path and producing a non-uniform current distribution. We showed that the restriction of the paths due to the variation of the contact resistance resulted in highly concentrated current flows and low heat dissipation which led to the formation of hotspots.

Opportunities for Thermally Aware Devices and Control in AlGaN/GaN on Si(C) Power Electronic Systems

Karen Dowling, Lex Pardon, Ananth Saran Yalamarthy, Miguel Muñoz Rojo, Debbie Senesky

Delft Technical University

Frore Systems

University of Twente

Email: k.m.dowling@tudelft.nl

Gallium nitride (GaN) is a rising star in power electronic devices. This material has a high critical electric field, high mobility, and harsh environment durability which has led to orders-of-magnitude improvements in energy management circuits compared to existing Silicon devices. In addition, its lateral device structure enables it to be easily integrated in monolithic systems, meaning control circuits, sensors, and energy harvesting could all be implemented alongside high voltage switches. This presents several opportunities in the areas of thermal management control of power electronics. Our previous work includes the realization of harsh environment GaN Hall-devices which operate from 50K to 600C and a GaN n-type thermoelectric platform leveraging the two-dimensional electron gas (2DEG) for its high power factor. In this talk, we also propose future directions and present our preliminary findings. First, we will show the potential of modulation of the GaN 2DEG to create a “thermal-smart” magnetic field device through the use of gating the active area. Gated cryogenic measurements to 50K will be presented. Next, we will discuss the potential of creating p-type thermoelectric elements by future investigation of two dimensional hole gases and present our preliminary simulations of the structure along with finite element estimations of a thermoelectric energy harvester. Finally, we propose another future concept regarding thermal switching with GaN-on-SiC devices that leverage the highly thermal conductive SiC substrate and piezoelectric properties of GaN devices to create hybrid actuators that couple with existing SiC MEMS machining techniques to create thermal routing elements with micro-electro-mechanical systems.

Thermal Photon Driven Transistor

Philippe Ben Abdallah

University of Paris-Saclay

Email: pba@institutoptique.fr

The transistor is a key element of modern computer technology which has revolutionized our everyday life. Due to its unique capabilities to control and amplify an electric current, it is the main building block of any electronic logic gate, and therefore of any information-processing device. Here I introduced a thermal analog of transistor that operates by exchanging heat flux driven by thermal photons, instead of the flow of electrons. I will show that such a device can not only find broad applications in the field of thermal management to switch, modulate, and even amplify heat flux through exchange of thermal photons but also it can be used to design novel logic gates. This three terminal system could pave the way to innovative wireless sensors using heat as a primary source of energy and to a ‘low-electricity’ technology enabling information processing. This could provide a new degree of freedom in the direct interaction of machines and smart systems with their environment using thermal signals.

Thermomagnetic Devices for Heat Flow Switching, Rectification, and Amplification

Lorenzo Castelli, Qing Zhu, Ajay Garg, Kaitlyn Zdrojewski, Trevor Shimokusu, Pooja Sashital, Geoff Wehmeyer

Rice University

Email: lc94@rice.edu

Switchable and nonlinear thermal devices are enabling technologies for applications ranging from waste heat scavenging to solid-state cooling and battery thermal management. There is need for high-performance thermal devices with a convenient temperature range of operation, a simple manufacturing process, and an ability to work in a wide range of vacuum and air environments. Here, we demonstrate devices using a passive magnetic actuation mechanism that enables heat flow switching, rectification, and amplification. Our devices rely on the reversible paramagnetic/ferromagnetic phase transition of the magnetically soft material gadolinium, which has a convenient Curie temperature near 20ºC. We use this magnetically-actuated mechanism to construct three devices: (1) a thermal regulator, a two-terminal device in which the thermal conductance can be passively tuned ON or OFF by the temperature of the control terminal; (2) an oscillating thermal diode, in which an unstable balance between gravitational forces and temperature-dependent magnetic forces allows for thermal rectification; and (3) the first reported thermal transistor, a three-terminal device in which the temperature at the gate terminal controls the source-drain heat flow in a manner that leads to heat flow switching and amplification. The thermal regulator and the thermal transistor display average thermal switch ratios in vacuum of 34+30-13 and 109+62-38 respectively, with switching temperatures near 20ºC and a 5ºC thermal hysteresis. The thermal transistor displays measured heat flow amplifications near 30, meaning that a small gate heat flow can drive a larger source-drain heat flow. Finally, the oscillating thermal diode shows a maximum thermal rectification of 23 in air and of 16 in vacuum. We quantify the durability and performance of all devices over >1000 switching cycles, and construct proof-of-concept thermal circuits to demonstrate that these thermal devices enable thermal logic operations and temperature regulation capabilities for advanced thermal management applications.

Magnetocaloric Cooling with Fluidic Thermal Switches Based on Electrowetting Effect

Urban Tomc, Blaž Velkavrh, Hana Uršič, Lukas Beyer, Jens Freudenberger, Klara Lünser, Enric Stern Taulats, Lluis Manosa, Kitanovski Andrej.Kitanovski@Fs.Uni-Lj.Si

University of Ljubljana
Jožef Stefan Institute
Leibniz Institute for Solid State and Materials Research
University of Barcelona

Email: urban.tomc@fs.uni-lj.si

Magnetocaloric energy conversion is a promising alternative to vapor compression. However, challenges persist in using rare-earth materials, improving energy efficiency, and reducing costs. Current devices use Active Magnetic Regeneration, which is moderately efficient at low frequencies (up to 5 Hz). Therefore, researchers are exploring higher frequency operation (up to 20 Hz) to enhance power density, allowing device miniaturization and reduced material usage. Challenges include developing efficient high-frequency magnetic field sources and implementing new thermal control devices like thermal switches. Microfluidics, particularly ElectroWetting On Dielectric (EWOD), offers a unique approach to thermal switches. Unlike traditional continuous flow microfluidic devices, there's growing interest in manipulating discrete droplets using surface tension effects. EWOD leverages the wettability of liquids on a dielectric surface by varying electrical potential, earning it the term "digital microfluidics" due to its similarity to digital microelectronics. In this contribution, we introduce a novel concept for a magnetocaloric device that combines the magnetocaloric effect with EWOD droplet manipulation as a thermal switch mechanism. We will explore various potential device designs, their operation, and discuss the materials and their properties that constitute the entire device.

Fabrication of Microfluidic Thermal Switches for Magnetocaloric Cooling Applications

Blaž Velkavrh, Hana Uršič, Victor Regis, Matej Šadl, Bianka Colarič, Kitanovski Andrej.Kitanovski@Fs.Uni-Lj.Si, Urban Tomc

University of Ljubljana
Jožef Stefan Institute
University of Ljubljana

Email: blaz.velkavrh@fs.uni-lj.si

Vapour-compression has become a well-developed technology for refrigeration and air conditioning, however it suffers from environmental problems of refrigerants and moderate efficiency. In recent years, the interest in finding alternative technologies has rapidly increased. One alternative is magnetocaloric (MC) energy conversion. Despite showing the greatest potential to replace vapour-compression in the future, today's MC devices have issues with inefficient heat transfer and hydraulic losses. In light of this, we are preparing unique fluidic thermal switches based on the electro-wetting on dielectric (EWOD) effect for MC cooling applications. With EWOD, an external electric potential can manipulate liquid droplets (e.g. change of shape, movement), which can be used to form a thermal bridge for heat transfer. Such a device consists of a multilayer structure on a MC substrate with two electrodes and two dielectric layers. Two electrodes are needed for the EWOD effect, while dielectric layers separate the electrodes from the MC material and liquid droplet. In this work, we prepared multilayer structures with three different dielectric thick-film layers on stainless steel and gold-sputtered glass substrates and investigated their functional properties. SU-8 polymer layers were prepared using the spin-coating method while polyimide P84® NT and Al_2O_3 layers were prepared using aerosol deposition method. In the aerosol deposition, N_2 was used as the carrier gas with two flow rates: 2 L/min for polyimide and 4 L/min for Al_2O_3 layers. In this contribution, the surface and cross-sectional microstructural properties together with dielectric properties of prepared films will be discussed. In addition, droplet dynamics measurements of EWOD effect were performed to determine operating frequency range of thermal switches.

Radiative Thermal Switch

Guillaume Boudan, Étienne Eustache, Philippe Ben-Abdallah, Riccardo Messina, Christophe Lucchesi, Agnès Andrée Delmas

Thales Research & Technology, Palaiseau, France
Optics Institute Graduate School, Paris Saclay, France
Institut National des Sciences Appliquées de Lyon, France
Groupe INSA

Email: guillaume.boudan@thalesgroup.com

The management of heat dissipated in solids is a major issue in many sectors, ranging from microelectronics to space industry. In the vast majority of these applications, radiation represents one of the main channels through which heat can be lost or absorbed in an uncontrolled way, potentially leading to a dramatic reduction of performances. With this respect, the design of thermal switches (i.e. devices where the radiative behavior strongly depends on the temperature) proves to be a promising tool. In our work we introduce a thermal switch based on a multilayer planar structure. Our device employs vanadium dioxide (VO_2), known for its first-order Mott transition from a low-temperature insulating phase to a high-temperature metallic phase close to room temperature. By exploiting an optimization procedure based on the genetic algorithm, we have designed a structure having the opposite thermal behavior, i.e. weakly emitting thermal radiation at lower temperatures and very similar to a blackbody at higher temperatures. Optimized structure taking into account fabrication constraint is characterized with a dedicated experimental setup, allowing direct measurement of spectral and directional emissivity with temperature.

Electronic Cooling Above 1.5 K

Joel Hätinen, Alberto Ronzani, Renan Loreto, Emma Mykkänen, Klaara Viisanen, Tuure Rantanen, Joel Geisor, Antti Kemppinen, Mika Prunila

Ministry of Employment and Economy
VTT Technical Research Centre of Finland

Email: joel.hatinen@vtt.fi

Low temperatures are important enablers in many applications of technological relevance, from superconducting quantum computing to sensitive radiation detection. Cryogenic temperatures below few kelvin are traditionally achieved by pumped helium systems. Alternatively, tunnel junction based on-chip cooling methods have been studied for several decades, yielding proof-of-concept demonstrators at various temperatures below 1 K. We extend the range by first demonstration of absolute electronic cooling from bath temperatures above 1.5 K by using superconducting tunnel junction devices for cooling and thermometry.

Thermal Conductance Tunnability by Means of Block Copolymer Driven Nanostructuration of Si-Ultrathin Membranes

Libertad Abad Muñoz, Alex Rodriguez-Iglesias, Hugo Gómez-Torres, Jordi Tur-Prats, Marta Fernández-Regúlez, Iñigo Martin-Fernandez, Francesc Pérez- Murano, Joaquin Santander , Luis Fonseca , Xavier Alvarez-Calafell, Aitor Lopeandia

Superior Council of Scientific Investigations

ICN2

UAB

IMB-CNM-CSIC

Email: llibertat.abad@imb-cnm.csic.es

Traditionally, the most used thermoelectric (TE) materials have been based on chalcogenides. Nevertheless, these widely used materials are based on alloys of scarce, expensive, and toxic elements that are difficult to integrate within the semiconductor industry. In this contribution, we present the study of suspended Si ultra-thin films as TE material, which are integrated into semiconductor technologies. Although the enhancement in TE performance of thin films is modest when compared to one-dimensional structures, this is compensated and enhanced by introducing surface nanostructuration with the aim to reach ZT similar to the best-reported values in the literature for Si structures. Here, we present a cost-effective and scalable approach for the fabrication of the membranes by block copolymer (BCP) nanopatterning achieving sub-10 nm resolution. Nanostructured Si membranes are fabricated on the device layer of a SOI wafer with ultrathin device layer. The BCP used for surface nanostructuration is polystyrene-block-polymethylmethacrylate (PS-b-PMMA) with cylindrical or lamellar morphology and a period between 28 and 80 nm. The period of such structures is controlled by properly blending BCPs of different molecular weights and the depth and the shape of the walls are tuned by the RIE conditions. The final structures are doped by spin-on-dopant to achieve an optimal doping concentration, 10E19-10E20 cm^{-3}, for TE applications. The thermal conductivity of the membranes will be evaluated using the 3w Völklein approach on an appropriate test structure.

2D Transition Metal Dichalcogenides as Highly Anisotropic and Compact Thermal Barriers for Thermal Management Applications

Rem Elnahas, Jimmy Faria Albanese, Miguel Muñoz Rojo

University of Twente

Email: r.elnahas@utwente.nl

The implementation of smart technologies in the last years have furtherly raised the bar for the development of highly efficient thermal management systems in modern electronic devices. The requested higher processing capacity of these new electronic circuits has led to an increase of dissipated heat, which is the main responsible of reduced performance and lifespan of micro-electronics. Within this context, different thermal insulation strategies are applied to protect sensitive electronics or to limit over-heating effects, like the integration of air gaps or encapsulation of single electronic parts. Unfortunately, the effectiveness of most of these approaches is related to the thickness of the insulating layer, which often leads to the adoption of thick materials or gaps that limit the compactness of the device. Bidimensional (2D) materials like graphene, h-BN and some transition metal dichalcogenides (TMDs) exhibit a great potential as effective thermal barriers. These are highly anisotropic thermal materials which imply that their cross-plane thermal resistances are between two and three orders of magnitude higher than their corresponding in-plane ones. Recently, Vaziri et al. reported that 2D TMDs heterostructures can exhibit cross-plane thermal resistivities that are equivalent to 300 nm SiO2 with less than 2 nanometers of thickness. This property is mainly due to the mass-density and phonon density of states mismatch at the interfaces. Additionally, their flexibility and resistance to thermal degradation make these materials ideal candidates as ultra-thin thermal insulators. To this day, many 2D TMDs thermal properties remain highly unexplored due to the complexity in achieving high-accuracy measurement of the absolute temperature of nanometric-thick layers. In this work, we aim to study the properties that influence phonon transport at the interface boundary in 2D TMDs heterostructures. The cross-plane thermal resistance of few-layers 2D TMDs and the Kapitza resistance among different TMDs-monolayers interface will be evaluated through Raman Thermometry.

Keynote Lecture: Microactuators for Miniature-Scale Thermal Energy Technologies

Manfred Kohl

Hermann von Helmholtz Association of German Research Centers

Email: manfred.kohl@kit.edu

The ongoing miniaturization and increase of functionality have enabled the development and widespread use of smart micro-scale devices and systems. The field of microelectromechanical systems (MEMS) has undergone an exceptionally dynamic evolution from silicon micromechanics to a highly diverse field comprising a large variety of materials and corresponding technologies. However, silicon is not a transducer material. Therefore, additional materials have to be introduced for conversion of energy into mechanical work. Common approaches include passive metal layers for fabrication of capacitor elements to enable electrostatic actuation and passive metal-silicon layers for thermal bimorph actuation. More advanced concepts for actuation consider transducer materials like shape memory alloy (SMA) films, magnetic SMA films and thermo-magnetic layers. This review covers a variety of approaches at KIT on the development of microactuators being useful for miniature-scale thermal energy applications. The various engineering aspects will be discussed including material properties, design, fabrication, experimental and numerical characterization as well as performance optimization. A first category of microactuators are cantilever actuators. Thermal bimorph microactuators have been developed based on SMA / Si and magnetic SMA / Si layer composites, whose shape has been optimized for homogenous temperature profiles. Ferromagnetic Ni-Mn-Ga film cantilever devices show a mixed martensitic and ferromagnetic transition, which enables bidirectional actuation with large stroke. Recent developments include thermal cantilever actuators with thermo-magnetic layers allowing for monostable, bistable or oscillating performance, which has been implemented in novel thermo-magnetic generators. A second category of microactuators are SMA bridge actuators featuring much higher force compared to cantilever-based devices. Also in this case, monostable, bistable and oscillating performances have been demonstrated enabling thermal energy applications like power generation or active control of heat transfer.

Mixed Convection by Buoyancy and Magnetothermal Forces

Masayuki Kaneda, Shunya Yoshimura, Kyosuke Urabe, Kazuhiko Suga

Koritsu Daigaku Hojin Osaka
Osaka Metropolitan University Japan

Email: mkaneda@omu.ac.jp

The magnetic force depends on the local gradient of magnetic flux density. In the paramagnetic fluid, the applied magentic force also depends on the temperature. Thus, this magnetic force is called magnetothermal force. In the gravity field, the magnetothermal force is comparable to the buoyancy force thus the mixed convection is formed. In this study, the magnetothermal convection under the gravity is numerically investigated and valudated experimetally. The use of permanent magnets with various maget array is also examined for the low-cost convection control. It is found that the heat transfer is locally enhanced and suppressed depending on the thermal boundary layer and magentic field. The unsteady convection also can be induced by the combination of them.

An Improved Physics-Informed Neural Networks for Transient Heat Conduction Analysis in Functionally Graded Materials

Zhuojia Fu, Shuainan Liu, Wenzhi Xu, Yuan Guo

Hohai University

Email: 20130010@hhu.edu.cn

This paper develops an improved physics-informed neural network (PINN) for effectively solving transient heat conduction problems in functionally graded materials.In the proposed improved PINN, the physics-informed kernel functions (PIKFs), which are derived according to the governing equations of the considered PDEs, are used to be the activation functions instead of the traditional activation functions. The proposed improved PINN puts the physical information of the considered governing equations in the activation functions, not in the loss function. By using the derived physics-informed kernel functions satisfying the considered governing equations of homogeneous, nonhomogeneous, transient PDEs as the activation functions, only the boundary/initial data are required to train the neural network. Finally, the feasibility and accuracy of the proposed improved PINN are validated by several benchmark examples.

Heat Transport Across Nanometre-Sized Gaps

Oscar Mateos, Pablo M. Martínez, J. G. Vilhena, Juan Cuevas

Autonomous University of Madrid

Email: oscar.mateosl@uam.es

Calorimetry has recently achieved a significant milestone by measuring heat transport through a single-atom junction, positioning us to investigate heat transfer across various distances, from atom-sized contacts to more extensive separations. When metals come into contact, heat is primarily carried by electrons at close proximity, while at greater distances, it is transported by photons in a Planckian or super-Planckian manner. However, when the separation reaches the nanometer scale, a transition from thermal radiation to conduction is expected. Recent experiments have shown that heat flux between gold contacts at sub-10nm distances exceeded conventional theories by several orders of magnitude, challenging our understanding of heat transfer across vacuum gaps. In this presentation, we propose to provide an atomic-level description of heat transport across sub-10nm gap between two metallic surfaces with conductivity values ranging nW/K on gaps as large as 5nm. The all-atom molecular dynamics simulations not only provide results in quantitative agreement with the aforementioned experiments but also settle a nearly decade long paradox.

Ballistic-Diffusive Heat Transport Crossover in Molecular Junctions

Pablo M. Martínez, Oscar Mateos, Juan Cuevas, J. G. Vilhena

Autonomous University of Madrid

Email: pablo.martinezm@uam.es

Understanding and controlling heat transport at the nanoscale is arguably one of the largest pending challenges of Nanoscience. The field has witnessed a renewed interest with recent spectacular experiments including measuring heat transport through a single-atom contacts, then through a single molecule junctions and later the realization of the first molecular-sized nano-refrigerator. However, a fundamental understanding of the novel and exotic heat transport phenomena at these scales remains elusive, ultimately hindering a de-novo design of novel nano-materials with tailored heat transport properties. In this work, we combined advanced all-atom non-equilibrium simulations with dedicated quantum-mechanical force fields to achieve atomically detailed insights into heat transport in benchmark alkane single molecule chains ranging from few atoms up to 4 micrometer long chains. Our simulations, quantitatively reproduce experimental data, but most importantly they provide an atomically detailed understanding on the breakdown of Fourier's classical law for heat transport as new a form of coherent heat transport emerges (ballistic). This surprising coherent for of heat transport is found to persist even in micron-sized contacts and in the presence of major defects. Spectral decomposition of heat flux unveils the non-local nature of this phenomenon thus paving the road for chemically engineering heat transport at the nanoscale. Alternatively, we also explore other strategies (force and mass) for controlling the transport regime (ballistic or diffusive) and overall conductance of alkane chains. By focusing on a relatively simple system, we gain valuable insights into the fundamental factors governing heat transport at the nanoscale.

Manipulating the Thermal Conductance by Applying Reversible Gas-Solid Reactions

Jonina Felbinger, Marc Linder

German Aerospace Center (DLR), Institute of Engineering Thermodynamics
German Aerospace Center Stuttgart

Email: jonina.felbinger@dlr.de

The increasing demand for energy-efficient buildings has prompted the development of innovative solutions, including adaptive building envelopes. One such solution is the controllable insulation layer, which has the potential to significantly reduce building energy consumption. The working principle of the controllable insulation layer is to adjust the gas pressure in open-pored insulation panels to control thermal conductivity along an S-shaped curve (Knudsen effect). This can be achieved by using thermochemical materials by precisely setting the temperature of the reaction material and thus the gas pressure can be adjusted reversibly along the equilibrium line. By coupling the thermochemical reaction with an open-pored panel, an insulation system with adjustable thermal conductance is realized. The approach proposed here is based on a thermochemical reactor that serves as an external component outside the panel as a control unit for the gas pressure and thus for the heat flux. The contribution will present a novel design of the thermochemical reactor, along with experimental proof of function as well as an analysis of power demand and dynamics for gas pressure variation in the range of 5 to 1000 mbar. Furthermore, the reactor is coupled with a vacuum insulation panel, and experimental measurements of the thermal behavior of the overall controllable insulation system – consisting of the porous panel and the connected metal hydride reactor component - are presented.

Optimization of Switching Algorithms for a Thermal Switch-PCM-Switch Stack in a Building Wall for Energy Savings

Sarah Chen, Chris Dames, Sumanjeet (Suman) Kaur

State of California
UC Berkeley
Lawrence Berkeley National Laboratory

Email: sarahmc777@berkeley.edu

Thermally switchable building envelopes can provide energy savings compared to their traditional static counterparts. For example, in the summertime cooling season the envelope can switch into a thermally conductive state at night to exploit the free cooling of the nighttime air, while switching back into a highly insulating state during the warm daytime. Further benefits can be realized by incorporating phase change materials (PCMs) into the same building envelope between two thermal switches, because the PCM can store that nighttime cooling for later use during the day. Here we use analytical and numerical modeling to investigate the advantages of a switch-PCM-switch building envelope and specifically look to optimize the switching algorithms for the two switches. We frame the problem using dimensionless groups in order to identify the key parameter groupings that determine the response. We then build a lightweight lumped element model with runtime speeds of <0.5 seconds per scenario, which enables a brute force search of over 300,000 potential switching algorithms to identify the one with the highest total and net energy savings. We compare the results of the brute force approach with an algorithm that was proposed previously based on physical intuition [Kishore et al., https://doi.org/10.1016/j.apenergy.2020.116306]. We then conduct a similar brute force investigation of switching algorithms optimizing for average daily cost of energy, focusing on scenarios where time-of-day energy pricing is important. Using the resulting optimal switching algorithms, we present cost and energy savings for several real cities based on their realistic temperature data and time-of-day pricing.

Thermal Energy Storage

Chairs: Monica Delgado, Anastasia Stamatiou, Dominic Groulx, Sebastian Gamish

Melting of a Phase Change Material in a Rectangular Cavity in the Presence of Metallic Fins

Claudia Naldi, Giulia Martino, Matteo Dongellini, Cesare Biserni, Gian Luca Morini, Sylvie Lorente

University of Bologna

Email: claudia.naldi2@unibo.it

The melting of the phase change material (PCM) octadecane, confined in a rectangular cavity and heated from the bottom surface, is studied. A variable number of vertical metallic fins is inserted from the enclosure bottom, in order to analyze their impact in the speeding of the melting process. Numerical simulations are performed through a finite element commercial software, modeling the phase change by means of the Apparent Heat Capacity Formulation. The results are analyzed in terms of: time needed for the PCM melting as a function of the fins number, time-dependent average liquid fraction, time-dependent local Rayleigh number and position of the melting front. The obtained outcomes evidence how, with a low number of metallic fins, the initial melting regime dominated by conduction at a certain time gives way to a convective melting regime characterized by Rayleigh–Bénard cells, in full agreement with the results of the theoretical analysis aimed at discovering the main scales of the problem. On the contrary, with a high number of metallic fins, conduction is the only mechanism that governs heat transfer and the rapid melting slows down in the end, when the phase change front reaches the top of the fins. In detail, the addition of the fins within the cavity yields a reduction in the time needed for the complete PCM melting up to 90% in the analyzed cases. The reported results provide new insights regarding the heat transfer mechanisms involved in PCMs melting within bottom-heated enclosures.

Analyzing the Impact of Using Phase Change Materials on Energy Consumption in Buildings: A Case Study

Igor Vušanović, Boris Hrnčić

Univerzitet Crne Gore

Email: igorvus@ucg.ac.me

The use of phase change material (PCM) in building materials has been shown to be a promising method for reducing energy consumption in buildings. This study investigates the effectiveness of using PCM in reducing energy consumption in a typical residential building located in Podgorica, Montenegro, a city with mild winters and hot and dry summers. EnergyPlus software was used to simulate the thermal performance of a building with and without the incorporation of PCM. The simulation was performed over a one-year period to account for seasonal variations in temperature and humidity. The simulation results indicate that the use of PCM in buildings design can significantly reduce energy consumption in buildings located in climates similar to Podgorica. This approach can also lead to a reduction in greenhouse gas emissions associated with energy consumption. These findings provide valuable insights for building designers and policymakers looking to reduce energy consumption and improve the sustainability of buildings. In conclusion, the results of this study support the use of PCM in building design as an effective strategy for reducing energy consumption and greenhouse gas emissions in buildings. Further research is needed to investigate the cost-effectiveness of using PCM in buildings design and to explore the potential for their widespread implementation.

Influence of PCMs on Thermal Performance of Wet Fire Protective Clothing

André Malaquias, João Bernardo Lares Moreira De Campos

Universidade do Porto

Email: up201007676@up.pt

The inclusion of phase change materials (PCMs) in firefighting protective clothing (FPC) has been previously shown to be beneficial during the exposure stage, but detrimental in the post-exposure, mainly due to PCM latent heat release towards the skin, after the exposure. It is unclear whether that is also the case for when the FPC is wet. Hence, in this study, a one-dimensional numerical approach is used to study the effect of PCM parameters (PCM mass, melting temperature) and global heat flux, on the thermal performance of a wet fire protective clothing. It is concluded that under wet conditions, the PCM amount, and its phase change temperature have a significant effect on thermal performance, depending on the heat exposure scenario considered. Great benefits are observed in introducing PCMs to prevent water condensation near the skin, but at the cost of greater PCM re-solidification skin damage in the post-exposure phase.

Thermal Management of High-Powered Short-Duration Electronics Aided by a Phase Change Material Thermal Energy System

Dominic Groulx, Devin Martherleur

Dalhousie University

Email: dominic.groulx@dal.ca

The high thermal energy storage density of phase change materials (PCMs) makes them interesting candidates in temperature management applications of electronics. The electronics to be managed clearly needs to operate in a transient manner, spending some time during its duty cycle at higher power when the additional heat generated can be stored in the PCMs, and some time at low to no power so the energy stored in the PCMs can finally be expelled to the environment. This work presents the design, testing and performance of a thermal management solution using a PCM for high-power short-duration electronic applications Typical of such high-power short-duration electronics, conventional cooling methods are sized to manage the maximum loading the cooling system will be subjected to in its operation, even if this loading occurs for a short period of time. This design ethos is inefficient and results in heavy and expensive equipment being used in an application where it will be underutilized most of its useful life. In this work, a PCM thermal energy storage (TES) device was built using a commercial plate and frame heat exchanger and investigated to determine the suitability of this system to manage the temperature of high-power electronics with a low duty cycle. These experiments showed how a PCM-TES can be used in conjunction with conventional temperature management equipment to create a reliable cooling solution while allowing the conventional cooling components to be decreased in size and capacity. This work shows that, for a given application, a cooling system can be more efficiently designed with the use of a PCM-TES and opens the door to additional research on this topic.

Integrated Computational Materials Engineering Simulations of Direct-Chill Casting

Božidar Šarler, Tadej Dobravec, Viktor Govže, Boštjan Mavrič, Katarina Mramor, Robert Vertnik, Gašper Vuga, Peter Cvahte, Marina Jelen, Marko Petrovič, Aleksandra Robič

University of Ljubljana
Institute of Metals and Technology, Ljubljana, Slovenia
Štore Steel d.o.o
Impol d.o.o.

Email: bozidar.sarler@fs.uni-lj.si

This paper presents an overview of the simulation system for the direct-chill casting of round aluminium billets. The system's structure fits the Integrated Computational Materials Engineering (ICME) framework. It is divided into related macroscopic, mesoscopic, and microscopic solidification models according to the scales. The macroscopic models are divided into fluid mechanics models, which are used to calculate the temperature, velocity, concentration and phase fields, and solid mechanics models, which are used to calculate the stresses and deformation fields. A vertical coupling with the possible flow stirring or breaking with the electromagnetic field is provided. Mesoscopic models are intended for grain size calculation, while microscopic models are devoted to microsegregation and detailed grain shape calculations. The multiscale and multiphysics solution procedures involved rely on meshless concepts. The simulation system can predict defects such as macrosegregation and hot tearing. Verification and validation of models and examples of simulations are presented. Ultimately, we give further directions for the system upgrades for slabs and refinement of the existing models.

Limitations of the Enthalpy-Porosity Method for Numerical Simulation of Close-Contact Melting on Asymmetric Surfaces

Victor Van Riet, Tomer Shockner, Wim Beyne, Gennady Ziskind, Michel De Paepe, Joris Degroote

Universiteit Gent

Ben-Gurion University of the Negev

Email: victor.vanriet@ugent.be

Latent thermal energy storage (LTES) systems are a promising technology to match the supply of renewable energy and thermal energy demands. The solid-liquid phase transitions during the charging and discharging of LTES systems allow these systems to achieve a high stored energy density for a small temperature range compared to sensible storage systems. One of the main challenges for such systems is slow charging due to the low thermal conductivity of most phase change materials (PCM). Close-contact melting (CCM) can accelerate melting times in LTES systems but the current numerical techniques for solid liquid phase change (e.g. the enthalpy-porosity method) have difficulties with accurately predicting this process. In this study, close-contact melting of PCM on an asymmetric surface is simulated using the enthalpy-porosity method in ANSYS Fluent. The geometry is based on the cross-section of a triplex tube heat exchanger filled with PCM. All PCM properties, including density, are temperature-dependent. In this way, phenomena such as natural convection, volume change and buoyancy between the solid and liquid are taken into account. The volume change is compensated by a gaseous expansion volume. Both 2D and 3D simulations expose to what extent the enthalpy-porosity method lacks in cases with CCM and how these shortcomings manifest themselves. The mushy zone constant, which is set to 10^5 to allow motion in the solid bulk, causes the solid phase to deform as a highly viscous fluid instead of moving as a rigid body. The deformation of the solid is up to 15% of its sinking velocity. As a result, the movement of the solid resembles creep behaviour and the obtained CCM patterns are not physically accurate. Furthermore, the density difference between the solid and liquid phases causes an avalanching effect in the mushy zone, which artificially strengthens convection and partially compensates the underestimated melting speeds.

Experimental Thermal Analysis of an Innovative Heat Sink Coupled to a Nanoemulsion

Giulia Righetti, Jorge Burgos, Leonor Hernandez, Rosa Mondragón, Simona Barison, Filippo Agresti, Simone Mancin

Universita degli Studi di Padova

Universitat Jaume I de Castellón

Universitat Jaume I

Consiglio Nazionale delle Ricerche

Email: giulia.righetti@unipd.it

This work presents some new experimental measurements collected on a very innovative system proposed for electronic cooling. This set up combines a classical heat sink with a latent thermal storage. The storage material used is a phase change nanoemulsion made of water and 5 wt. % RT44HC paraffin wax with 1-octadecanol as nucleating agent (weight fraction 1:10 with respect to PCM) to reduce the supercooling up to 32 % respect to samples without nucleating agent, while the heat sink is made of copper via 3D printing. The integration between the two components does not interfere with the external air convective heat transfer, as the emulsion is embedded inside the heat sink, which has an internal cavity. The tests analyse the temperature of the component and the emulsion in different locations during the charging and discharging phases. The results compare the behaviour of this innovative system to a system with the same geometry and only water inside. The presence of the latent thermal storage slightly modifies the temperature field and turns out to be useful in the case of intermittent heat load.

Numerical Modelling of Thermal Hysteresis in Melting and Solidification of Phase Change Materials

Maité Goderis, Adam Buruzs, Fabrizia Giordano, Tilman Barz, Wim Beyne, Michel De Paepe

Universiteit Gent

Republik Osterreich Bundeskanzleramt

Email: maite.goderis@ugent.be

Latent thermal energy storage (LTES) systems using solid-liquid phase change materials (PCMs) have garnered significant attention for their comparatively high energy density during phase transitions. However, investigations into the impact of thermal hysteresis effects on the storage performance remains a crucial challenge. This study employs Computational Fluid Dynamics (CFD) modelling in ANSYS Fluent to investigate and comprehend hysteresis phenomena during the solid-liquid phase change. The well-known enthalpy-porosity method is extended by introducing an alternative relation between solid/liquid fraction and PCM temperature. This alternative relation is implemented in Fluent through User Defined Functions. The new model allows for numerical analysis of supercooling and thermal hysteresis during partial and complete melting and solidification processes, and consequently allows for the analysis of hysteresis effects on the part-load operation of LTES systems. The proposed model is validated by comparing the numerical results to experimental data where the hysteresis loops of PCMs are observed.

Characterization of Hydration Levels of Salt Hydrates Using X-Ray Computed Tomography

Benjamin Fenk, Anastasia Stamatiou, Dario Guarda, Jorge Martinez Garcia, Philipp Schütz, Damian Gwerder, Poppy O'Neill, Rebecca Ravotti, Simone Mancin, Ludger Josef Fischer, Jörg Worlitschek

Hochschule Luzern

University of Padova

Lucerne University of Applied Sciences and Arts

Cowa Thermal Solutions AG

Email: benjamin.fenk@hslu.ch

Salt hydrates have gained increasing popularity as phase change materials (PCMs) in latent thermal energy storage systems due to their remarkable enthalpy of fusion and cost-effectiveness. These properties make them a promising choice for efficient energy storage solutions. Still, salt hydrates present several challenges, amongst which a tendency to segregate. Recently, X-ray computed tomography (XCT) has been proposed as a non-destructive method to distinguish different hydrate forms within a salt hydrate and thus give insights to the mechanisms behind segregation. However, it also presents distinct challenges. This work focuses on the use of XCT to develop advanced characterization methods to explore different hydrate forms of salt hydrates and their phase, with sodium acetate (SA) hydrates as a model system. The primary focus is on comparing grey values in the XCT measurements between pure reference samples and other salt hydrate samples. Specifically, we explore how variations in grey values correspond to changes in hydration level and phase. By elucidating these relationships, we aim to assess the suitability of XCT for the characterization of salt hydrates for PCM applications. The findings presented in this work offer valuable insights into the characterization of SA employing XCT, paving the way for characterization and optimization of other types of salt hydrates. As the demand for efficient and sustainable energy solutions continues to grow, the exploration of salt hydrates holds great promise for meeting these challenges.

Tracking of the Melting Front in Tube-in-Tube Latent Thermal Energy Storage Heat Exchangers

Julie Van Zele, Maité Goderis, Wim Beyne, Kenny Couvreur, Michel De Paepe

Universiteit Gent

Email: julie.vanzele@ugent.be

To achieve the desired reduction in greenhouse gas emissions and address the mismatch between energy and supply, latent thermal energy storage (LTES) systems will play a key role. Despite their great potential, they are not yet widely implemented mainly due to designing difficulties. Classic heat exchanger theory cannot be employed due to the transient behaviour of LTES systems. Analytical models describing the phase change front movement during melting or solidification can potentially provide a framework for designing LTES heat exchangers. Validation of these models require accurate data of the movement of the phase change front. Therefore, a tube-in-tube LTES heat exchanger is experimentally tested. In the setup, water flows through the inner tube as heat transfer fluid (HTF) and the shell contains a paraffin RT35HC as phase change material (PCM). A series of melting experiments with varying operating conditions have been performed to investigate the movement of the phase change front. The movement is represented by plotting the axial position of the phase change front as a function of time. The position of the phase change front is tracked over time using a camera placed next to the PCM tube. It is seen that each front-curve shows an S-shape with a linear part which represents a quasi-steady state regime of the moving speed. These observed outcomes are compared with an analytical model, which states that the time taken by the front to reach a certain axial position is a linear function of the position. The comparison reveals that the model is unable to accurately predict the movement of the phase change front. Too many assumptions are not applicable to the used experimental setup. Consequently, data fitting is performed. An inverse linear relationship is found of HTF inlet temperature and HTF mass flow with the speed of the phase change.

Water Adsorption on Microporous Chabazites as Thermal Energy Storage Materials

Alenka Ristić, Suzana Mal, Natasa Zabukovec Logar

National Institute of Chemistry Slovenia

Email: alenka.ristic@ki.si

Thermochemical energy storage (TCES), one of the three thermal energy storage (TES) technologies, can reduce fossil fuel consumption, which is still dominant in heating of buildings. This technology uses reversible chemical reaction or sorption processes of gases (i.e.water vapor) on solids or liquids. The efficiency of the sorption technology, based on the alternating adsorption (exothermic phenomenon) and desorption (endothermic phenomenon) of the working fluid on the adsorption materials, is determined by the performance of the adsorbent, which should have a high water adsorption capacity and, consequently, a high energy storage density, stability under humid conditions, and fast kinetics of adsorption and desorption. The current trend is focused on the development of water adsorbents with increased adsorption capacity and hydrothermal cycling stability. Increasing the water adsorption capacity of TCM materials usually results in higher thermal energy storage density and thus improved adsorbent performance. The aim of this study is to investigate the effect of different elemental compositions of the commercial and laboratory prepared microporous adsorbents with chabazite (CHA) structure in powder and granular form on water adsorption capacity. The chabazite structure contains 3-dimensional interconnected pore system with pore openings of 0.38 nm and large ellipsoidal cavities of 0.835 nm. The water adsorption behaviour of microporous adsorbents such as different aluminophosphates, a natural zeolite, and a modified synthetic zeolite was explored and compared with the well-known aluminophosphate APO-Tric powder, which exhibited an energy storage density of 320 Wh/kg at an adsorption temperature of 30 ^{o}C and a water vapor pressure of 12.5 mbar and a desorption temperature of 100 ^{o}C.

Numerical Investigation of Phase Change Material Integration in Structured Thermocline Systems for Concentrated Solar Power

Oriol Sanmarti Perona, Jordi Vera Fernandez, Santiago Torras Ortiz, Carlos David Perez Segarra

Universitat Politecnica de Catalunya

Email: oriol.sanmarti@upc.edu

The rapid growth of renewable energies may encounter impediments due to their intermittent nature. Addressing this issue needs the development of novel energy storage methods or enhancements to existing ones. Thermal energy storage has been widely explored in the literature as a potential solution to this challenge. Such systems can be utilized in conjunction with concentrated solar power, wherein molten salt is employed as a heat transfer fluid. Molten salt is the most expensive part of these systems, thus in the last years some alternatives have been studied, where a part of the fluid is substituted by a solid filler or phase change material (PCM) and only one tank is used. There is an interesting alternative presented in the recent years where the solid filler material is built in a structured way using waste materials (e.g., ceramics or concrete). The objective of this paper is to study the coupling of the above mentioned thermocline with PCM located in strategic places of the tank, in the same way is done in packed bed distributions. As the outlet temperature decreases in the thermocline distribution, studying the impact of adding PCM at the outlet can lead to performance enhancements. Similarly, placing PCM at the bottom (Cold side) can extend the cycle time until reaching the cutoff temperature and result in a reduction in thermocline thickness. Furthermore, the implementation of PCM between the filler material will be investigated to observe its effect on thermocline thickness and to study the increase in energy density. These effects will be examined using an unsteady one-dimensional model, with PCM behaviour validated against literature experiments, and considering two PCM geometries: cylindrical tubes or spheres. This study will provide valuable insights for the implementation of PCM in structured thermocline tanks.

Meshless Through-Process Modelling of Steel Production Path

Božidar Šarler, Izaz Ali, Tadej Dobravec, Tin Gošnik, Umut Hanoglu, Qingguo Liu, Miha Kovačič, Boštjan Mavrič, Katarina Mramor, Matjaž Perpar, Robert Vertnik, Gašper Vuga

Faculty of Mechanical Engineering, University of Ljubljana
Institute of Metals and Technology
Štore-Steel, d.o.o.

Email: bozidar.sarler@fs.uni-lj.si

This presentation provides an overview of the laboratory and plant experimental, computational, and artificial intelligence approaches to model macroscopic solid mechanics, fluid mechanics, electromagnetics, and microstructure evolution during billet steel processing. They fit the Integrated Computational Materials Engineering (ICME) framework. ICME is bridging the information from different physical concepts and their computational models. In Horizonal ICME (H-ICME) the simulation codes for different processing or product usage steps are connected with their associated multiscale structures and material properties. In Vertical ICME (V-ICME), the simulation codes at multiple length scales are used to describe the product properties. The H-ICME modules include EAF, continuous casting, controlled cooling, annealing, reheating, reverse and continuous hot rolling, cooling bed, and heat treatment. The V-ICME scales range from the grain size to several tenths of meters. The microstructure is formulated with the phase-field method, the meso-structure with the cellular automaton method and the macroscopic electromagnetic, fluid mechanics and solid mechanics fields with the continuum mechanics concept. The novel solution methods for describing the related multiscale and multiphysics thermomechanical problems are focused on a space-time adaptive meshless solution based on collocation with radial basis functions for solving the microscopic and macroscopic scales and the point automata concept for solving the mesoscopic scale. The phenomena tackled by this novel meshless technique range from the large-eddy simulation of continuous casting to the elastoplastic deformation of the products on the cooling bed. A selection of the models is presented where the computational speed is preferred over the modelling details and vice versa. The validation of the models is shown. A coupling of the physical models with artificial intelligence for quality, energy and productivity optimisation is presented.

Numerical Study on Hybrid Battery Thermal Management System Integrating Water, Phase Change Material and Fins Under New European Driving Cycle

Satyam Singh Thakur, Lalit Kumar

Indian Institute of Technology Bombay

Email: satyamsingh@iitb.ac.in

The present numerical study introduces a novel hybrid cooling method comprised of water, eicosane, and fins, as coolant, phase change material (PCM), and thermal conductivity enhancers, respectively. The system is used for thermal management of 18650-type lithium-ion battery (LIB) pack at a 3C discharge rate. The battery pack consists of twelve LIBs, arranged staggered, with six batteries in each row. An aluminum block with three curved surfaces is placed between every three cells in the battery pack to dissipate the heat. A serpentine cooling duct is designed to send the coolant (water) through the center of each aluminum block. A cavity, adjacent to the coolant flow path, has been made in each aluminum block to accommodate PCM. Fins, made of aluminum, are placed in the PCM volume to increase the effective thermal conductivity of the PCM. Five different BTMS designs are investigated in the present study: (i) only PCM, (ii) only water, (iii) PCM and water, (iv) PCM, fins, and water, and (v) PCM, water, and fins with variable sizes. The numerical simulations with water in the above designs are conducted for a volume flow rate of 0.4 mL/s. From investigations, it is observed that the maximum temperature of LIBs exceeds the permissible limit of 60 ℃ at the end of the 3C discharge rate in designs (i) and (iii). Whereas, the maximum temperature and thermal gradients in the battery module with design (v) are 47.8 and 4.9 ℃, respectively, which are 5.6% and 57.8% lower compared to design (ii), and 2.4% and 48.4% lower compared to design (iv), respectively. Further studies will be conducted on the effect of composite PCM, the variable contact area of the aluminum blocks, and volumetric flow rates of coolant to find the optimal design of BTMS for the safe operation of LIBs.

Modelling a Latent Thermal Energy Storage Prototype with a Diphasic Heat Transfer Fluid

Amandine Da Col, Fabrice Bentivoglio, Benoit Stutz

Commissariat a l'energie atomique et aux energies alternatives
CEA LITEN
Université de Savoie

Email: amandine.dacol@cea.fr

Latent Thermal Energy Storage (LHTES) is a solution for decoupling energy production and consumption. Combined to a thermal solar collectors, it smooths production and extend the duration of the power supply. In this way, an energy source that was intermittent and subject to weather variations can become reliable to supply an industrial process. This paper presents a numerical model to simulate the thermohydraulic behaviour of a shell-and-tubes LHTES with a two-phase heat transfer fluid (HTF). It is based on a 1D homogeneous approach for the HTF (water) and a 1.5D approach for the phase-change material (sodium nitrate). This approach has been validated with experimental data in a previous study for a LHTES with a monophasic HTF. The two highlights of this paper are as follows: On the one hand, computational fluid dynamics has been used to develop equivalent homogeneous materials, integrating the thermal properties of the phase-change material and the helical and longitudinal fins present in the LHTES. On the other hand, the model uses the asymptotic correlation of Steiner and Taborek to describe boiling heat transfers and usual correlations for single phase heat transfers. This work was carried out with the aim of achieving the widest possible range of validity with a computation time significantly lower than that of CFD models, while retaining a sufficient degree of accuracy. The paper focuses on the validation of the model against experimental data from the InPower project. The prototype is a 340 kWh storage that has been tested under different conditions. The model reproduces the total energy retrieved during a system discharge with relative error of 0.4% and a root-mean-square error of 5.68 kW on outlet power.

Conjugate Heat Transfer Analysis of the Transient Thermal Discharge of a Metallic Latent Heat Storage System

Frank Nees, Yogesh Sajikumar Pai

German Aerospace Center (DLR), Institute of Vehicle Concepts
Hermann von Helmholtz-Gemeinschaft Deutscher Forschungszentren

Email: frank.nees@dlr.de

Thermal energy storage systems utilizing metallic phase change materials exhibit great potential as a technology for mobile applications, offering high storage densities and high thermal discharge rates. First experimental investigations show the functionality and performance characteristics of this system. However, there is a lack of detailed numerical studies on transient thermal discharge. Therefore, this work focuses on a transient conjugate heat transfer simulation. For this purpose, a three-dimensional computational fluid dynamics model was developed. For validation of the numerical model, the results of the simulations are compared to experimental measurements. The investigated storage is based on an aluminum silicon alloy and a box-shaped graphite container design. Heat extraction is achieved by forced convection of ambient air utilizing a controllable fan, while charging is realized by electrical heaters. The transient thermal discharge was simulated from 650 °C to 100 °C, and the solidification of the storage material at around 577 °C was simulated using an enthalpy-porosity approach. The discharge time and total heat flow show good agreement with the experimental data, indicating the model's successful validation. An empirical study was carried out to determine the thermal contact resistance at the interface between the storage material and the graphite container. The transition from liquid to solid is associated with a significant rise in thermal contact resistance. Observation of the liquid fraction during the phase change showed that the storage material solidifies faster near to the air inlet due to high temperature gradients along the fluid flow direction. The present study contributes a validated numerical model and substantial physical findings regarding the thermal discharge of a metallic latent heat thermal energy storage system.

Modeling of Phase Change Hysteresis During Partial Phase Change With the “Shift”-Method

Sebastian Gamisch, Stefan Gschwander

Fraunhofer-Gesellschaft zur Forderung der angewandten Forschung eV
Fraunhofer ISE

Email: sebastian.gamisch@ise.fraunhofer.de

For phase change materials (PCM) a difference between melting and crystallization temperature can occur, the so-called supercooling. This hysteresis effect reduces the potential of a high storage density for latent heat storages as the narrow temperature range of the phase change is broadened. For the investigation of PCM for different applications by numerical modelling, like for battery temperature control or latent thermal energy storage, this hysteresis effect as well as partial phase change must be considered as due to operational condition a a partial melting and crystallization is likely to occur. Therefore, phenomenological methods based on the experimentally determined enthalpy-temperature relations for melting and crystallization can be applied. Doing so it is a challenge to model the behavior when the heating is changed to cooling and vice versa during phase transition. The study presents a new phenomenological method which is called “shift”-method. It is derived especially but not only for PCMs with a multistage phase change during crystallization, like it can occur for microencapsulated PCMs. For these materials first a supercooled major phase change followed by a second phase change with larger supercooling associated to rotator phases can be observed. The presented method is verified by experimental results and compared to an existing modelling approach called “curve-scale”-method. Therefore, partial melting and crystallization processes of a PCM composite are experimentally characterized with differential scanning calorimetry (DSC). Additionally, both methods are integrated in a Modelica / Dymola model of a passive battery cooling system with the PCM composite which is validated against experimental results. With both methods the temperature evolution with time can be described with a deviation of less than 2 %. According to the results the benefits and drawbacks of the methods are discussed.

CFD Simulation of Solid/Liquid Phase Change in Commercial PCMs Using the slPCMlib Library

Adam Buruzs, Fabrizia Giordano, Manuel Schieder, Christoph Reichl, Maité Goderis, Wim Beyne, Michel De Paepe, Tilman Barz

Republic of Austria Federal Chancellery
AIT Austrian Institute of Technology GmbH
University of Gent

Email: adam.buruzs@ait.ac.at

Numerical simulations with phase change materials (PCMs) provide a solid basis for the development of Latent Thermal Energy Storage(LTES) systems. In the European research project “HYSTORE” (Hybrid services from advanced thermal energy storage systems), a prototype is developed for a Refrigerant-PCM-Water heat exchanger (RPW-HEX) coupled with a modular heat pump for domestic hot water generation in residential buildings. Modeling the solidification and melting behavior of PCMs presents several challenges, including non-isothermal phase change, multi-step transitions, complex temperature dependence of specific heat capacity between solid and liquid states, and contribution of convection in the heat transfer. To tackle these challenges, this study proposes a novel numerical method based on real experimental data to model complex phase change behaviors of commercial PCMs in Ansys Fluent CFD software. The phase transition of materials is simulated with the apparent heat capacity (AHC) method, which is implemented in Fluent as a User Defined Function. The temperature dependent specific heat capacities are loaded as cubic Hermite spline functions from the slPCMlib database (Solid-Liquid PCM Library), based on the measurements performed by PCM manufacturing companies. The heat convection is simulated with the Boussinesq approximation. Moreover, the method is tested on a selected group of PCMs, with complex phase change behavior, and the results are compared to the piecewise constant heat capacity Solidification and Melting (S&M) model built in Ansys Fluent. The study demonstrates the additional benefit of the adoption of complex temperature dependent material properties on the simulation results.

Ventilation System with Heat Recovery and PCM Thermal Energy Storage for "Free" Cooling in Buildings

Uroš Stritih, Primož Lenassi, Urška Mlakar, Dr. Eneja Osterman

University of Ljubljana

Email: uros.stritih@fs.uni-lj.si

For the conventional cooling of buildings, we are using electrical energy from the network, which increases annually due to the increase in comfort. Alternatives are being sought to reduce electricity consumption. Phase change materials in ventilation systems can provide us with the accumulation of night-time cold in the summer months, resulting in a reduction in the consumption of electricity for cooling. Heat storage devices that take advantage of the phase change are used for short-term storage of thermal energy, as in this case they have the greatest potential. In the introduction to the paper a review of heat storage devices and their use are made which show their connection with the Energy Performance Building Directive (EPBD) and the Energy Efficiency Directive (EED). Based on the measured results of the measurements in the LOSK laboratory a comparison with the model that calculates the storage of heat and cold in a phase change material (PCM - Phase Change Material) developed by a group of researchers from BUT Brno was made. With the TRNYSY software tool and the model simulations of the ventilation gains in the summer season were made and compared with the gains when using the heat exchanger (recuperator) and the PCM heat accumulator for the selected room.

Study of the Phase Change Processes of Encapsulated Phase Change Material for Solar Thermal Energy Storage in Domestic Hot Water Production

Cristina Bianqui, Antonio Viedma, Alberto Egea, Alberto García Pinar

Universidad Politecnica de Cartagena

Email: cristina.bianqui@edu.upct.es

According to the "Renewables 2022 Global Status Report," global energy consumption has increased significantly in the last two decades by 54%. One third of this percentage is allocated to the residential sector, of which a 77% is used for building heating and domestic hot water (DHW). These are applications where solar energy is particularly interesting, as its usage has risen in recent years. An emerging research area to enhance the efficiency of solar systems to obtain DHW involves the use of phase change materials (PCMs) in energy storage systems. This helps to reduce the problem of the dependency that these systems have on solar availability. Specifically, there have been recent works on the design and manufacturing of hybrid accumulators, which contain both water and encapsulated PCM. These are advantageous for their easy integration into existing systems and the quicker DHW acquisition, making them an economical and suitable solution for domestic use. This study aims to examine the melting processes of the PCM in a cylindrical-shaped container, made of stainless steel. It has a diameter of 60 mm and a height of 20 mm. The chosen PCM is the paraffin RT35HC, which has a melting point of 35ºC. Experiments are conducted in a water-filled thermal bath, placing the container vertically and heating the water using a thermostat. Temperature field analysis of the PCM is performed using thermocouples, and simultaneously, the interior of the encapsulated material is observed through a transparent wall. It has been proven that the sensors are capable of detecting the gradual evolution of the liquid phase of the PCM and its accumulation at the top of the container due to its lower density.

Study of the Phase Change Processes of Encapsulated Phase Change Material for Solar Thermal Energy Storage in Domestic Hot Water Production

Cristina Bianqui, Antonio Viedma, Alberto Egea, Alberto García Pinar

Universidad Politecnica de Cartagena

Email: cristina.bianqui@edu.upct.es

According to the "Renewables 2022 Global Status Report," global energy consumption has increased significantly in the last two decades by 54%. One third of this percentage is allocated to the residential sector, of which a 77% is used for building heating and domestic hot water (DHW). These are applications where solar energy is particularly interesting, as its usage has risen in recent years. An emerging research area to enhance the efficiency of solar systems to obtain DHW involves the use of phase change materials (PCMs) in energy storage systems. This helps to reduce the problem of the dependency that these systems have on solar availability. Specifically, there have been recent works on the design and manufacturing of hybrid accumulators, which contain both water and encapsulated PCM. These are advantageous for their easy integration into existing systems and the quicker DHW acquisition, making them an economical and suitable solution for domestic use. This study aims to examine the melting processes of the PCM in a cylindrical-shaped container, made of stainless steel. It has a diameter of 60 mm and a height of 20 mm. The chosen PCM is the paraffin RT35HC, which has a melting point of 35ºC. Experiments are conducted in a water-filled thermal bath, placing the container vertically and heating the water using a thermostat. Temperature field analysis of the PCM is performed using thermocouples, and simultaneously, the interior of the encapsulated material is observed through a transparent wall. It has been proven that the sensors are capable of detecting the gradual evolution of the liquid phase of the PCM and its accumulation at the top of the container due to its lower density.

Optimizing MOF Properties for Seasonal Heat Storage: A Machine Learning Approach

Eliodoro Chiavazzo, Giovanni Trezza, Luca Bergamasco, Matteo Fasano

Politecnico di Torino

Email: eliodoro.chiavazzo@polito.it

In the quest to enhance thermochemical energy storage applications using promising sorbents such as metal organic frameworks MOFs, this work presents a detailed study on the optimization of MOFs properties for gas sorption, with a particular focus on CO2 and H2O adsorption. Through the analysis of crystallographic descriptors, the study aims to streamline the selection process for MOFs that could potentially exceed the performance of existing water sorbent pairs in thermal energy storage applications. A comprehensive comparison of sequential learning (SL) algorithms reveals a method for identifying the minimal yet crucial set of descriptors that significantly influence adsorption properties of MOFs. The two step protocol involves constructing and traini ng machine learning (ML) models to determine the number of influential descriptors and utilizing SHAP analysis to evaluate the importance of each variable. Findings suggest that including only these critical descriptors in the exploration space not only accelerates convergence performance but also reduces computational load. Notably, the COMBO algorithm consistently outshines random guessing, validating its efficacy in materials optimization. The challenge of accessing full adsorption properties across the entire coverage range is addressed by proposing a computational screening procedure that requires minimal input data yet offers reliable predictions of important performance metrics. This method notably suggests that certain vanadium based MOFs, originally designed for different purposes, could surpass the current leading compounds for thermal energy storage, primarily due to their optimal Henry coefficient values for water adsorption. The significance of this research extends beyond identifying intrinsic material properties; it may serve as a significant advancement in efficient MOF (and other screening and optimization. The implications are substantial for other engineering applications such as thermally driven water harvesting, water sorption thermal energy storage, and solar cooling, underscoring the potential of SL algorithms in material discovery.

Assessing the Impact of Copper Wools on the Discharging Process of a Phase Change Material-Based Heat Exchanger Prototype

Alessandro Ribezzo, Sara Risco Amigó, Saranprabhu Mani Kala, Emiliano Borri, Matteo Morciano, Luca Bergamasco, Gabriel Zsembinszki, Matteo Fasano, Eliodoro Chiavazzo, Luisa F. Cabeza

Politecnico di Torino
Universitat de Lleida

Email: alessandro.ribezzo@polito.it

Phase Change Materials (PCMs) stand out as a promising solution within the current array of Thermal Energy Storage (TES) technologies, thanks to their superior energy storage capacities (compared to sensible solutions) and technological readiness. Nonetheless, the limited thermal conductivity of these materials may lead to incomplete phase transitions during use, resulting in a decrease in their effective energy storage capabilities. The major solutions to mitigate this issue that are present in literature either require a significant modification in the heat exchanger design (such as fins) or are costly and still far from reaching predicted performance levels (as PCM nanocomposites). In this work, the use of copper wools is proposed as fillers within a PCM-based heat exchanger prototype, and the assessment of its impact on the heat transfer behavior of the material is evaluated by performing charging and discharging processes. This type of inclusion has been chosen as it is relatively cheap, it can be implemented within an already existing heat exchanger, and it does not suffer from segregation. Two different wools have been tested in two configurations, thus resulting in five test cases (four containing the wools and one containing solely PCM). Both experimental measurements and numerical simulations have been exploited to investigate the figures of merit of the comparison between the different cases, i.e. the charging and discharging periods of the PCM-based heat exchanger, the power as a function of time, and the total energy absorbed or released during the tests. The promising results, especially the remarkable decrease in the time needed for the complete solidification of the PCM within the heat exchanger, may lead to other analyses regarding different configurations and/or materials, in order to obtain an optimized compromise between the decrease in the specific energy density, and the increase in the specific power density.

Enhanced Latent Thermal Energy Battery with Additive Manufacturing

Matteo Morciano, Matteo Alberghini, Flaviana Calignano, Diego Giovanni Manfredi, Matteo Fasano, Pietro Asinari, Eliodoro Chiavazzo

Polytechnic Institute of Turin

Polytechnic University of Turin

Politenicnico di Torino and Istituto Nazionale di Ricerca Metrologica

Email: matteo.morciano@polito.it

This study addresses the challenge of low thermal conductivity in Phase Change Materials (PCMs), like paraffin waxes, which hinders the efficiency of latent heat storage systems during rapid charge and discharge cycles. To overcome this, we explored the use of metal additive manufacturing to create a device-scale latent heat storage system. This design allows for optimal heat conduction paths within the PCM, targeting applications requiring medium temperatures around 90°C. The system's performance was investigated through both numerical simulations and experimental procedures. Results showed the system's ability to achieve significant specific power densities (> 700 W/kg) during charging and an even higher (> 1300 W/kg) during discharging. Importantly, these figures remained consistent over multiple cycles, showcasing the system's durability and reliability. The validated finite-element model, aligned with the experimental data, provided a basis for general design guidelines to boost the system's performance further. Potential applications of this technology are highlighted in the automotive industry, where such systems could efficiently manage thermal energy, for instance, by capturing excess heat from an engine's cooling radiator to expedite the warm-up process during a cold start, which is a critical phase for reducing pollutant emissions. The study concludes by emphasizing the transformative potential of metal additive manufacturing in the thermal energy storage sector. It allows for rapid optimization and production of complex, thermally-conductive structures, which could revolutionize the design and efficiency of energy storage systems. However, for larger-scale applications, the research suggests a redesign of thermal insulation layers, such as using vacuum containers instead of the wood and wool covers used in the study for testing and model validation purposes.

Advancing Thermochemical Storage: Synthesis and Characterization of Cement-Based Composite Materials.

Alessio Mondello, Matteo Morciano, Luca Lavagna, Matteo Pavese, Eliodoro Chiavazzo

Polytechnic Institute of Turin

Email: alessio.mondello@polito.it

Thermal Energy Storage (TES) is crucial for sustainability of the energy sector, yet the development of cost-effective, robust materials remains a significant challenge. This study aims at exploring the synthesis and thermal characterization of cement-based composites for seasonal thermo-chemical energy storage, with the goal to harness the high energy density of hygroscopic salts while mitigating their limitations. In particular, we investigate composites with several cement matrices (e.g. Portland Cement and others) to improve the salt-cement compatibility. Furthermore, we investigate the possible incorporation of other porous low-cost compounds to enhance porosity and improve the economic aspects. As far as the characterization aspects are concerned, we show experimental adsorption isotherms at different temperatures to estimate key material properties like isosteric heat and water uptake, along with the expected relevant figures of merit such as energy density. Our research leverages on the adjustable porosity and affordability of cement as a host matrix for the ‘active phase’ (e.g. magnesium sulfate). We experimented with two synthesis approaches: traditional dry impregnation and an in-situ technique suitable for cements. The in-situ method, being straightforward and reproducible, permits greater control over salt content. We show that preliminary cost analysis positions these composites competitively in the market. Although we are still at a sub-optimal stage, the potential for cost reduction of some less popular cement matrices - through scale economies - suggests an opportunity for improvement. This research paves the way for further advancements in TES materials, presenting a viable alternative for energy conservation through improved thermochemical storage composites.

Low-Order Design Models for Latent Thermal Energy Storage

Wim Beyne, Joris Degroote, Michel De Paepe

Universiteit Gent

Email: wim.beyne@ugent.be

Latent thermal energy storage can be a key technology for a green energy transition by matching fluctuating heat demand and supply. In order to implement a storage system, it needs to be designed which requires estimating the outlet temperature of a system for a given geometry and time history of the heat transfer fluid's mass flow rate and inlet temperature. Currently, design methods are either overly simplistic, focusing solely on e.g. the phase change time or requiring the solution of partial differential equations which can be computationally expensive. The present paper proposes a novel approach where a latent thermal energy storage system is decomposed into a heat transfer fluid vessel, a sensible storage system and a storage system with only latent heat. Computationally inexpensive models are available for all three of these sub heat exchangers. A heat exchanger model is obtained by connecting the sub heat exchangers in parallel. This novel approach is used to model an industrial scale shell and tube latent thermal energy storage heat exchanger. The predicted outlet temperature is compared to the measured outlet temperature and the design model obtains good agreement.

Exploring the Potential and Challenges of Phase Change Materials in Future Heat Storage Systems via Computations and Experiments

Jada-Tiana Carnie, Dr. Antonis Sergis, Yannis Hardalupas, Professor Maria Charalambides

Imperial College London

Email: jada-tiana.carnie21@imperial.ac.uk

The current work describes computational and experimental efforts to enable new heat storage technologies. Approximately 50% of the energy consumed globally is used to generate and manage heat. This is still predominantly acquired from the combustion of fossil fuels, consequently generating just over 40% of the total global CO2 emissions. To achieve a sustainable future, there is a need for sustainable, decarbonised and dependable energy sources. Heat storage is a key technology that could help achieving this goal. Recently published work by the current authors has shown the importance of heat storage in decarbonising heating (and cooling) of buildings using phase change materials (PCMs). The black-box approach followed evaluated the energetic, cost, and environmental impacts of a novel concept PCM storage scheme with microgeneration. The results obtained showed that this system had advantages over current and future competing heating technologies. However, PCM technology remains at low technological readiness levels because of the lack of understanding of the complex heat transfer physics associated with their phase change transition. For example, understanding and characterizing the propagation of the solid-liquid front during charging and discharging in such heat storage modules will be important in predicting and optimising their performance. In the current work, computations have been performed to understand and optimise the effects of container geometry and size on the charging and discharging heat flux profiles of a simple heat storage module. Furthermore, fundamental experiments have also been implemented to identify the complex governing mechanisms of heat transfer and storage in such materials during transient operation.

Session: Application

Chairs: Nicola Massarotti, Yann Bartosiewicz, Matjaž Hriberšek

Carbon Capture with Renewable Heating Based on the Technical Lime Cycle

Eva Klockow, Marc Linder

Hermann von Helmholtz-Gemeinschaft Deutscher Forschungszentren
Deutsches Zentrum für Luft- und Raumfahrt, Stuttgart

Email: eva.klockow@dlr.de

About 2/3 of the final energy demand in German households is required for heat supply, with only 17 % provided from renewable energies. As a result, heating accounts for a large share of CO_2 emissions in Germany and the world. At DLR, an energy storage technology was developed that can shift the supply of renewable heat – mainly available in summer – to times of heat demand – mainly in winter – with almost no losses. Basis of this thermochemical energy storage is the reversible, exothermic reaction from CaO to $Ca(OH)_2$, with lime and water as abundant and globally available storage materials. By energetically closing the technical lime-cycle this renewable heating could be extended to additionally capture CO_2 from the atmosphere – hence local heating demand would actually be carbon-negative. Thereby, operation of the technical lime-cycle would proceed in circular mode: After heat has been provided, the passive storage phase of $Ca(OH)_2$ is used to bind atmospheric CO_2 through carbonation. The final calcination process separates the captured CO_2 and charges the thermal energy storage. Backbone for the technical implementation of the concept is a community-based transport system that transfers the different lime-based compounds between the central charging and decentral discharging sites. The rate at which the lime can pass through the system primarily depends on the kinetically slow carbonation. Experimental results will be presented, that show how the reaction is influenced by surface and atmospheric conditions as relative humidity and temperature. Optimal reaction conditions are derived, as they are essential for the overall efficiency. The contribution will describe the concept in detail, present the current status of the technology and give a brief outlook on currently addressed scientific and technological questions at the DLR.

Investigation of the Operating Characteristic of a Demand-Controlled 368 m Deep CO2 Thermosyphon Geothermal Borehole Heat Exchanger for Building Heating

Janina Hagedorn, Robin Kahlfeld, Malte Nageler, Stephan Kabelac

Leibniz Universitat Hannover

Email: hagedorn@ift.uni-hannover.de

Geothermal energy from a 368 m deep closed vertical pipe filled with CO_2 operating on the thermosiphon principle is used as the heat source for a heat pump. This heat pump heats the water for the buffer tank to a temperature of 40 – 55°C to heat a 1600 m² office building. A thermosyphon of this design with such a depth is unique in Germany. The heat pipe extracts the thermal energy by natural circulation of the evaporating and condensing working fluid in the pipe and does not require an additional pumping power as is the case in convention borehole heat exchanger. It delivers a heat output of up to 22 kW to the heat pump at a ground temperature up to 14°C. The working fluid used, CO_2, is non-toxic and therefore does not pose a risk to the environment or water even if a leakage occurs. The consumption-controlled use of the geothermal probe is being tested and the control of the heat pump optimized for one of these probes. The first heating season showed that this heat source in combination with a conventional compression heat pump reliably and quickly provides the required heat energy at any time, whereby the efficiency or source temperature of the probe depends strongly on the frequency of operation. The average COP of the last heating season was between 2.3 and 4.6. For the second heating season the temperature of the soil around the probe is also measured via a distributed temperature sensing system, that covers the whole depth of the probe. The aim is to investigate the down cooling and regeneration of the soil over the heating season, both of which can be observed.

Experiment and Model Results of a First Bidirectional Active Temperature Control Module with Metal Hydrides for a Fuel Cell

Hanna Lösch, Eva Fensterle, Inga Bürger

Hermann von Helmholtz-Gemeinschaft Deutscher Forschungszentren
DLR

Email: hanna.loesch@dlr.de

There exist plenty approaches to temperature control for various applications. However, many common temperature control techniques have a lack of active controllability and bidirectionality. One example is liquid cooling with the ambient as heat sink, which is only able to cool not to heat (not bidirectional) and uses parasitic power for a pump and a radiator. In the present contribution we would like to present a novel bidirectional and active temperature control module (TCM) which is suitable for hydrogen applications and especially for fuel cells (FC). It is based on the thermochemical reaction between metal hydrides and hydrogen depending on temperature and pressure. Therefore, it operates as reversible thermal energy storage. The metal reacts exo- and endothermically with the hydrogen while ab- and desorbing, respectively. While the absorption releases heat, the desorption takes up heat of a source, here the FC, and thereby cools it down. In the overall process of the TCM, hydrogen is absorbed by the metal hydride and then released to a hydrogen system while desorbing. Therefore, no energy is consumed. The reaction and thus the resulting reaction temperature can be controlled using the reactor pressure, so the system temperature is bidirectional and actively controllable. In order to address required thermal powers, the reactor design needs to consider the main aspects like heat transfer and mass transfer. Reaction and heat transfer are simulated in a 0D model. Results of the according model for the outlet temperature, loading status and pressure will be presented and compared to experimental data. Furthermore, a temperature controller is implemented in the model as well as in the hardware components of the experimental setup. The results demonstrate that the controller changes the pressure in the reactor depending on the interfering temperature of the fluid resulting in a controlled and thus constant outlet temperature.

State of the Art of Heat Pumps Above 150°C

Matevž Cimermančič, Katja Klinar, Andrej Kitanovski

University of Ljubljana

Email: matevz.cimermancic6@gmail.com

In current aim to catch long-term energy and climate policies, the role of the heating and cooling sector cannot be overstated. Heating, accounting for 50 % of final global energy consumption, stands as a huge target for transformation. In this context, heat pumps are emerging as a potential decarbonization tool that promises to significantly reduce emissions through the use of renewable or waste energy, as they can utilize waste heat to elevate temperature to higher levels more efficiently compared to any other existing technology. Particularly, the utilization of high-temperature heat pumps (HTTPs) operating at temperatures exceeding 150 °C holds the potential to enhance overall energy efficiency. This paper offers an insightful exploration into the current state of the art in HTTPs with sink temperatures above 150 °C. We delve into a comparative analysis of various kinds of HTTPs, examining factors such as heating capacity, efficiency, Energy Return on Investment (EROI), and partial exploration of Life Cycle Assessment (LCA). Our primary focus centers on the examination of vapor compression systems, but also other types of HTTPs, such as mechanical vapour recompression, sorption, thermal vapour recompression and hybrid systems, are being investigated as well. Additionally, we provide valuable recommendations and guidelines for the future integration of industrial HTTPs.

State of the Art of Micro Vapour Compression Systems

Mihael Blatnik, Katja Klinar, Andrej Kitanovski

University of Ljubljana

Email: blatnik.mihael@gmail.com

Thermal management is becoming more challenging due to the increasing compactness of appliances. Affordable and efficient solutions include active liquid cooling methods, thermoelectric cooling, and the use of heat pipes. In comparison, vapour compression technologies achieve the highest power density, however, these types of devices are not yet developed enough for small systems such as computers and workstations. To increase low efficiency of small-scale vapour compression systems (with an average of 20 % based on the Carnot cycle), technology trends are moving not only towards the development of efficient small compressors, but also towards efficient miniature evaporators and condensers, which have not yet been adequately researched and developed to date. In this review, we present several vapour compression refrigeration systems that are based in some way of the miniaturisation of systems or individual components – evaporators, condensers, and compressors. They will be compared in terms of cooling power density, COP, specific mass, and price. Lastly, guidelines for future implementation of miniature vapour compression systems are presented.

Estimation of Temperature Hotspots on Li-Ion Cell Under Different Operating and Cooling Conditions: A Machine-Learning Approach

Rajesh Akula, Lalit Kumar

Indian Institute of Technology Bombay

Email: rajesh95.akula@gmail.com

The energy conversion efficiency of Lithium-ion (Li-ion) batteries used in electric vehicles is a vital function of their operating temperatures and thermal gradients. Hence, it is imperative to identify thermal hotspots in the battery module to design an appropriate cooling system for mitigating their impact on reliability. In the present study, an attempt has been made to estimate the location and strength of temperature hotspots on the surface of the AMP20M1HD-A0 Li-ion pouch cell using a machine learning algorithm named Gaussian Process Regression (GPR). Preliminary simulations are conducted on the considered cell, using the NTGK electrochemical model available in commercial ANSYS Fluent software, to validate its electrical and thermal behaviour against literature for 1, 2, and 3C discharge rates. The maximum discrepancies between the present simulations and experiments reported in literature are less than 1% and 5% for voltage and temperatures, respectively. The Latin Hyper Cube sampling technique is used to generate 80 samples for estimating the strength and location of hotspots for any discharge rates in the range of 0.5 to 5C, at any depth of discharge in the field of 0 to 1, and with any ambient convection heat loss condition between 0 to 80 W/m^{2}K. Numerical simulations for these 80 samples will be conducted using the above validated numerical model. The obtained location and strength of hotspot temperatures from the numerical simulations for the 80 samples will be used to train the GPR. The Leave One Out Cross-Validation is planned to be implemented to test the accuracy of the prediction. This study helps identify the location and strength of hotspots within no time and without the intervention of electrochemical models in which multiple cumbersome equations need to be solved simultaneously.

Physically Based Heat Exchanger Sizing Method for the Thermal Management System of All-Electric Regional Aircraft

Marius Nozinski, Carlo De Servi, Behnam Parizad Benam, Stephan Kabelac, Chiara Falsetti

Leibniz Universitat Hannover

Technische Universiteit Delft

Email: nozinski@ift.uni-hannover.de

Fully electric propulsion systems or hybrid solutions integrating hydrogen fuel cells and batteries are promising options to reduce the overall climate impact of regional and mid-range aircraft. However, the production of entropy within the power drive system calls for advanced thermal management system solutions since heat rejection through the hot exhaust gas as for conventional propulsion systems is no longer possible. To address such a challenge, this work investigates the best radiator geometry embedded in a ram-air duct system for the cooling loop of an all-electric regional aircraft based on the ATR-72 platform. In particular, two types of finned heat exchanger geometry are assessed for the radiator. The first features flat tubes with internal microchannels, while the second adopts a plate-fin configuration. To identify a suitable heat exchanger sizing method, the accuracy and simulation time of different discretization schemes of geometry are compared with those obtained with a simple 0D model. Each heat exchanger and the ram-air duct is sized for a hot-day take-off. The geometrical design variables of the overall ram-air system are optimized with respect to weight and drag minimization and the optimal designs form a Pareto front. The performance of specific system solutions on the Pareto front is investigated for cruise and descent to ensure a safe operation of the thermal management system over the entire flight mission. Finally, the ram-air system performance at cruise conditions is compared to that obtained by a thermal management system adopting a skin heat exchanger to highlight a potential second heat sink on all-electric aircraft.

Experimental Characterization and Numerical Modelling of an Air-Water CO2 Heat Pump Under Different Outdoor and Operating Conditions for Domestic Hot Water Generation

F.J.S. Velasco, Fernando Illán-Gómez, Rufino Esono Biyogo Obono, Jose R. Garcia-Cascales, Ramón A. Otón Martínez

Universidad Politecnica de Cartagena

Email: fjavier.sanchez@upct.es

This work assesses the influence of different outdoor and operating conditions on the performance of an air-to-water CO_2 heat pump for domestic hot water generation For that purpose, an air-to-water CO_2 heat pump with an internal heat exchanger was developed and characterized in an experimental campaign under well-controlled lab conditions. The system includes an in-house control system implemented to operate at the optimal gas cooler pressure. The heat pump and the control system are tested for domestic hot water generation. The impact of the internal heat exchanger is also assessed. The experimental data obtained are used to validate a numerical model of the heat pump under different air velocity, air temperature and air humidity at the evaporator. The heat pump uses a finned-and-tubes heat exchanger as evaporator and plate heat exchangers as gas-cooler and internal heat exchanger. The numerical model used a 1-dimensional, cell-by-cell discretization method to characterize the performance of both, the gas-cooler and the evaporator under a wide range of values of their key variables. The internal heat exchanger is also modeled with the same strategy. ARHI equations are considered to describe the compressor performance. The effectiveness of the control system proposed is experimentally confirmed during the experimental tests. Results also show a good agreement between the numerical simulations and the experimental data which permits to validate the model and the assessment of the performance of the transcritical heat pump under different climate conditions.

Seasonal Energy Performance Assessment of a Hybrid HVAC System Driven by Solar and Biomass Energy for Space Heating and Cooling in Residential Buildings

Matteo Dongellini, Christoph Moser, Szabolcs Varga, Joao Soares, Claudia Naldi, Gian Luca Morini

Graz University of Technology
University of Porto
University of Bologna

Email: matteo.dongellini@unibo.it

The H2020-funded project Hybrid-BioVGE aims to develop an efficient and sustainable HVAC system concept for buildings' climatization entirely based on renewable energy. The Hybrid-BioVGE system provides for space heating/cooling and DHW production of small- and medium-scale residential buildings and integrates solar and biomass heat as energy inputs. During the heating period, thermal energy is provided directly by solar thermal collectors and a biomass boiler. On the other hand, the cooling energy demand is met by a thermally-driven chiller based on a Variable Geometry Ejector (VGE), which adjusts its geometry according to two degrees of freedom. In this work, the simulation model of the Hybrid-BioVGE system demonstrator installed in Porto (Portugal) has been developed in TRNSYS environment. Numerical simulations have been performed to evaluate the key performance indicators and optimize the system energy performance. Obtained results put in evidence that up to 98% of the building energy demand for space heating is provided by renewables, with a solar fraction higher than 60%. Conversely, simulation results point out that the system energy performance is lower than expected during the cooling season. In this case, the solar fraction is lower than 50% due to the limited performance of the VGE. The findings of this work state that the critical feature of the system is the condenser loop, based on two heat dissipators by means of which the condensation heat is rejected to the ambient air. A substantial increase in the system seasonal performance, up to +400%, can be achieved if the condenser loop is revisited and water is used to reject heat from the VGE. For example, condensation heat could be used to pre-heat DHW or pool water in gyms or wellness centres.

Assessment of Internal and External Disturbances on the Fuzzy-Based Thermal Control of a Sub-Scaled Building Testbed

Arturo Pacheco-Vega, Anayely Saguilan

The California State University System

Email: apacheco@calstatela.edu

This experimental study presents an extention of our previous work on the development of robust fuzzy-based thermal control strategies of a multi-room sub-scaled building testbed. In the present case, the focus is placed on testing the robustness of the fuzzy controller under internal and external disturbances, as it deals with maintaitning a single setpoint value for all room temperatures. The system has eight rooms that are distributed on two floors. A cooling unit provides cool air to each room whereas 40W light bulbs that serve as heat sources. T-type thermocouples are used to gather temperature data, and eight dampers are used to deliver the airflow. The controller uses information about the difference between setpoint and actual temperatures, their derivative, and their cumulative integral. Both the fuzzy sets and the if-then rules are built from the experimental data, with a Mamdani inference method providing the corresponding values to the actuators. Temperature readings and control actions are done with LabVIEW, while the fuzzy controller is implemented in MATLAB. Results from experimental tests show that the fuzzy control strategy can handle the different types of disturbances while maintaining the corresponding room setpoint.

Influence of Supply Temperature and Booster Technology on the Energetic Performance of a District Heating Network

Alixe Degelin, Robin Tassenoy, Elias Vieren, Toon Demeester, Michel De Paepe

Universiteit Gent

Email: alixe.degelin@ugent.be

In 2019, 70% of the households in Flanders still used natural gas as a fuel for space heating. To accelerate the decarbonisation of the building stock in urban areas, district heating networks supplied by a central heat pump in combination with geothermal energy, were suggested. While central heat pumps are more energy efficient at low condensing temperatures, existing buildings often require high supply temperatures for space heating and domestic hot water. In addition, heat losses decrease when lowering the supply temperature in the distribution network, while pumping losses can simultaneously increase due to increased mass flow rates. This article investigates the influence of low and ultra-low supply temperatures on the primary energy demand and energy efficiency of a district heating network supplied by a central heat pump, taking into account the need for booster heat pumps or booster electric heaters. The simulations consider network temperatures ranging from 10°C to 75°C and distinguish between refurbished and non-refurbished buildings. The dynamic simulation of the network is performed using the IDEAS and Buildings libraries in Dymola (Modelica). The results of the study show that lowering the supply temperature to 45°/55°C for non-refurbished buildings and to 45°C for refurbished buildings in combination with a booster heat pump results in the lowest total primary energy use. In the case of refurbished buildings, the tradeoff between a network at 55°C without booster and a network at 10°C with individual heat pumps is highly sensitive to the performance of the central heat pump. Implementing measures to decrease the temperature needed for space heating offers significant benefits, as it can improve the overall energy efficiency from 291% to 449%.

Parametric Study for Determination of Nighttime Solidification Characteristics of Building Active-Passive System with Nighttime Ventilation for Free Cooling

Eva Zavrl, Urban Tomc, Mohamed El Mankibi, Mateja Dovjak, Uroš Stritih

University of Ljubljana
Ecole nationale des Travaux Publics de l'Etat

Email: eva.zavrl@fs.uni-lj.si

Due to the consequences of global warming, cooling of lightweight buildings with large window area is one of the main future challenges. Despite the fact, that the buildings provide sufficient thermal insulation during the winter, they have low in thermal mass and cannot accumulate enough heat to prevent the overheating of buildings' interior. The present study introduces an active-passive system (APS) with phase change material (PCM) plates integrated into the internal wall and ceiling substructure which during the day cools the indoor space passively. To ensure sufficient cooling effect of the system during the daytime cycle, the system must solidify completely during the nighttime cycle at operation costs comparable to the ones of conventional cooling systems. To improve the heat transfer of the PCM, an air gap was formed between the primary wall and ceiling and the PCM substructure and ventilated with the cool outdoor air during the nighttime cycle actively. To find the required solidification times of the PCM under South-Eastern European climate conditions, the parametric study was obtained with ANSYS Fluent model which was later extended for the energy performance and operation cost analysis. During the nighttime cycle, six different constant outdoor inlet air temperature scenarios (15-20 °C) were modelled. The numerical model was validated against the experimental results obtained at the Hybcell test facility. The results show that with the current configuration, the PCM plates can be sufficiently solidified at supply air temperatures of 17 °C and below before the end of the nighttime cycle. If the system is ventilated with inlet air temperatures above 17 °C, additional cooling options must be considered. The study also determines the energy performance of the system under the chosen conditions and compares its feasibility to the conventional cooling systems (energy efficiency class A+++ and G air-conditioning device).

Comparative Analysis of the Behaviour of the Hot Water Production System of an Indoor Swimming Pool

F.J.S. Velasco, J. Giménez-Villa, Jose R. Garcia-Cascales, Fernando Illán-Gómez, Ramón A. Otón Martínez, J.P. Delgado Marín

Universidad Politecnica de Cartagena

Email: fjavier.sanchez@upct.es

In this paper, we analise the thermal behaviour and the environmental impact of an indoor swimming pool placed in Pozo Estrecho, Cartagena, in the Southeast of Spain. The existing installation is formed by a group of solar thermal collectors supported by two propane heaters which heat the water of the pool and satisfy the needs of hot water in the changing room of the pool. Here we propose an installation based on the use of a heat pump supported by solar thermal collectors. In this work, we first describe the installations. They are modelled with transys and the models employed briefly described. Later on, we compare their primary energy consumption and its environmental impact in terms of their CO2 emmisions. Some conclusions are finally drawn.

Computational Investigation of the Effect of Geometry and Fuel Composition on the Performance of a Solide Oxide Fuel Cell

Pedro Coelho, João Paulo Cardoso Peixeiro De Freitas, Ana Filipa Ferreira

Universidade de Lisboa
Instituto Superior Técnico, Universidade de Lisboa

Email: pedro.coelho@tecnico.ulisboa.pt

The need to reduce the consumption of fossil fuels due to their negative impact on the quality of the air and on the greenhouse effect is causing major changes in the energy sector, with a progressively greater contribution of renewable energy sources to satisfy the global energy demand. Fuel cell technology is emerging as a key player in the energy transition since fuel cells convert directly chemical energy to electricity and heat without pollutant emissions. Among the different types available, solid oxide fuel cells can operate at high temperatures (up to 1000ºC), leading to high efficiency and stability. This work is concerned with a computational investigation of solid oxide fuel cells. The governing conservation equations for mass, momentum, energy, species transport and electric charge are solved for a three-dimensional planar fuel cell. The predictions are firstly validated for a planar solid oxide fuel cell using experimental data available in the literature. Then, different fuel and oxidizer compositions are considered and their influence on the performance of the fuel cell is investigated. The results show that when the hydrogen molar fraction in the fuel decreases, the maximum power and temperature decrease, while the efficiency firstly increases and reaches a maximum for a hydrogen molar fraction of 40%, and then starts to decrease. The power, the efficiency and the maximum temperature are greater when the oxidizer is oxygen rather than air. The influence of the geometry of the air and fuel channels is also assessed, namely the dimensions of the channels and the effect of steps and obstacles in those channels.

Free Piston Linear Generator Efficiency Evaluation Using 0D Modeling Approach

Raffaele Saviano, Carlo Beatrice, Armando Maiello

Consiglio Nazionale delle Ricerche
CNR-Stems
Università degli studi di Napoli Federico II

Email: raffaele.saviano@stems.cnr.it

Exploring Free Piston Linear Generators (FPLGs) technology could be useful in filling the gap between the incoming emission legislations (Euro 7) and the current performance of commercial vehicles. The main application's fields for this technology are powering operating machinery, as an alternative to BEVs or ICEs, or heavy vehicles dedicated to people or materials transport over medium and long distances powering. This technology involves the use of a linear electric generator coupled to a thermal engine. Mechanical losses are significantly lower than a traditional ICE due to the absence of a crankshaft. Thanks to the linear motion developed, the electric generation useful for powering a battery pack is guaranteed. Due to the absence of mechanical constraints, it is possible to achieve high compression ratio values and thus significantly improve combustion efficiency. In contrast to the advantages just discussed, it should not be forgotten that a control system capable of guaranteeing system performance and stability is required otherwise malfunctions could occur such as piston hitting the head, cycle to cycle variation, misfiring. In this paper a zero-dimensional model of an Opposed Free Piston Linear Generator developed in the Matlab&Simulink environment is presented. The main processes characterising the machine operation mode have been modelled with the aim of assessing the overall OFPLG efficiency. A control strategy has been implemented to manage the dynamics mover's dynamic. Results achieved concerning a 2-stroke supercharged Opposed Free Piston Linear Generator are shown to demonstrate the implemented control strategy's reliability.

3D CFD Simulation of Room Air Temperature and Velocity with Ceiling Fan and Exhaust Fan and the Resulting Thermal Comfort

Trino Thomas, Sateesh Gedupudi

Indian Institute of Technology Madras

Email: trinothomas@gmail.com

The integration of energy-efficient solutions remains an essential hurdle in the pursuit of sustainable and comfortable indoor settings. As building energy consumption continues to climb, there is an urgent need to implement methods that reduce environmental effects and, at the same time, maintain adequate thermal comfort for occupant well-being and productivity. Ceiling fans, a standard fixture in many indoor settings, have gained popularity for their ability to provide better air movement and temperature adjustment while using substantially less energy than standard HVAC systems. This work presents a 3D Computational Fluid Dynamics (CFD) investigation of thermal comfort utilizing ceiling fans. A 3D fan zone model available in ANSYS Fluent is used to model the ceiling fan, considering the ceiling fan as a solid disc. Numerical methodology has been validated by verifying the transient simulation results against the experimental data available in the literature. The work analyzes thermal comfort parameters - temperature and velocity - in a room with a typical ceiling fan available in India. The room temperature and velocity distribution are presented for different temperatures of the roof and different speeds of the fan, both without and with windows, considering different ambient temperatures commonly encountered in a hot climatic zone of India. Based on the room's simulated air temperature and air velocity, the work identifies the optimum fan speed for thermal comfort in different conditions.

Magnetocaloric Refrigerator with Magnetic Pump and Liquid Metals

Keerthivasan Rajamani, Mina Shahi, Bob Stolwijk

Universiteit Twente

Email: k.rajamani@utwente.nl

One-seventh of the worldwide electricity is used for refrigeration related activities. When compared to the predominantly used vapor compression refrigeration system, magnetocaloric refrigeration provides an energy-efficient and eco-friendly alternative. It utilizes a magnetocaloric material which undergoes temperature changes when exposed to magnetic field changes. Nearly 60% of the total system cost is due to the permanent magnets used. For a given cooling power, the amount of permanent magnet needed is determined by the residence time required by the heat transfer fluid in each cycle. The start-of-the-art systems utilize water-alcohol mixture as the heat transfer fluid, and has a mechanical pump for its circulation. By utilizing non-toxic and non-hazardous gallium-indium-tin based liquid metals as heat transfer fluid, which has three order of magnitude higher thermal diffusivity, the residence time and the permanent magnetic material is reduced nearly by a factor of 10. Even accounting for the cost of the liquid metal, it results in considerable savings in the system cost. We present experimental results on the entropy change of using liquid metal with Mn-Fe-P-Si magnetocaloric material, and numerical results on the system level analysis. Further, to improve the reliability of the system, we propose to use magnetic pumping for heat transfer fluid circulation, which does not have any moving parts. The state-of-the art magnetic pump works at frequencies in the range of 100 to 1000 Hz. However, they cause detrimental heating effects due to eddy current losses. To overcome this, we have designed a pump that operates at 1 Hz and still achieves the comparable flow rates. We present numerical and experimental results on the pump design by studying the dynamics of fluid rise in a vertical pipe using a commercial ferrofluid.

Effect of Water Injector Location on Combustion and Performance of an HCCI Engine – a CFD Analysis

Bharat Naik, Jm Mallikarjuna

Indian Institute of Technology Madras

Email: me20d031@smail.iitm.ac.in

The homogeneous charge compression ignition (HCCI) engine is capable of producing high thermal efficiency with very low emissions. However, because of the limited upper load level constrained by knocking, the HCCI engine is not yet commercialized. Water injection can tackle this issue, which can enhance the upper load limit without excessive heat release rate. However, the effectiveness of water injection depends on in-cylinder water evaporation, which in turn depends on water injection parameters. Therefore, this study is conducted to evaluate the effect of water injector location on the combustion and performance characteristics of an HCCI engine by a computational fluid dynamic (CFD) analysis. The engine specifications and boundary conditions are taken from the literature. The primary reference fuel PRF90 (90% iso-octane and 10% n-heptane) is used as a surrogate fuel injected into the intake port during the early intake stroke, with an overall equivalence ratio of about 0.88. The engine is operated at a speed of 2000 rev/min., with a compression ratio 12.5. The CFD models used are validated from the experimental data available in the literature for the engine considered. Here, the water injector location is optimized based on the maximum pressure rise (MRPR) rate, the in-cylinder spatial distribution of water vapor, and nitric oxide (NOx) emissions. The results show that the water injector at 37 mm from the center of the cylinder gives better results than in other cases considered. With the optimum water injector location, the indicated mean effective pressure is increased by about 23.7% than without the water injection case. This study will be helpful in the development of modern HCCI engines.

Numerical Modelling of a Hybrid Vapor Compression Refrigeration Assisted Closed Loop Liquid Cooling System for High-Performance Computing Systems

Fazeel Mohammed Naduvilakath Mohammed, Gerard Byrne, Anthony Robinson

The University of Dublin Trinity College

Email: naduvilf@tcd.ie

As computing and electronics technology continue to advance and find themselves in new harsher environments (e.g. Edge computing), the need for effective cooling solutions to dissipate the heat generated by high-powered processors (CPUs, GPUs) and other peripheral components (DIMMs, SSDs, VRM, power supplies etc.) becomes more challenging. Traditional fan-fin air cooling methods are now not always feasible, and there is a growing interest in refrigeration technology as a means to improve thermal management. This study presents a numerical model of a miniature vapor compression refrigeration system that is specifically designed for electronics cooling applications. The model was created using a physical approach and calculation algorithm that relies on nested iterative loops to solve the system of coupled non-linear equations. To verify the model, a set of experimental tests were conducted using an in-house experimental test facility. The system components were modeled using fundamental laws of thermodynamics, established heat exchanger theory, along with empirical correlations obtained from experimental data when appopriate. This approach simplified the theoretical description of the refrigeration system and minimized potential uncertainties associated with complex physical laws or incomplete knowledge of certain components, such as the compressor. The primary objective of this research is to gain a better understanding of the system's behavior and predict its performance under different operating conditions. This knowledge will be used to improve the performance and effectiveness of the vapor compression refrigeration system design for advanced thermal management of computing devices and systems.

Numerical Investigation of Different Cooling Technologies During Heatwaves and Power Outages in Catalan Mediterranean Buildings

Núria Garrido, Roser Capdevila Paramio, Enrico Tontodonati, Luca Borghero

Universitat Politecnica de Catalunya

Email: nuria.garrido@upc.edu

The Mediterranean area is particularly affected by heatwaves, which are extremely critical for health and productivity of people inside the dwellings as well as for energy consumptions related to cooling demand. This study numerically investigated the resilience of different cooling technologies in the most common construction in Catalan building stock during present and future climatic conditions, heatwaves and power outages, evaluating also their impact on energy consumptions, environment and costs. The techniques implemented in the model include green roof, blinds, advanced windows, natural ventilation, air conditioning and their combinations. Typical years and heatwaves for the city of Barcelona have been created for three time periods, Present, Mid Future and Long Future. They have been modelled using dynamical downscaled Regional Climate Models, climate projections based on IPCC scenarios. Power outages have been simulated in some of these scenarios, following a new methodology that predicted their occurrence during the days and the hours with higher exposition to this risk. Resilience was assessed using indicators that take into account the linear correlation between the grade of overheating inside a dwelling and the severity of outdoor conditions. Thermal comfort and survivability have been also investigated, evaluating sensations of building's occupants related to indoor air temperature and relative humidity. The original building presented acceptable levels of resilience, while thermal comfort conditions resulted poor, especially during heatwaves. The application of the different cooling technologies significantly helped in the improvement of resilience and survivability inside the dwelling. However, their performances reduced in the future scenarios. Air conditioning resulted the best solution for avoiding overheating, especially in heatwaves. Natural ventilation achieved good results in the improvement of resilience and comfort, especially when combined with green roofs. Combinations of several passive measures resulted necessary for maintaining suitable conditions in case of power outages.

Enhancing Thermal Modelling of Electric Machines: Steady-state Verification of a Novel Methodology

Jasper Nonneman, Ilya T'Jollyn, Michel De Paepe

Universiteit Gent
Universiteit Antwerpen

Email: jasper.nonneman@ugent.be

The increasing power density of electric machines for electric vehicle applications necessitates accurate thermal analysis for both design and real-time condition monitoring through simulations while keeping computation times below one minute. A previously proposed hybrid modelling methodology utilizes a lumped parameter thermal network where thermal resistances are extracted from 2D FEM, with the aim of reducing computational demands while maintaining accuracy. Nonetheless, this modelling method lacks a clear methodology for network and parameter selection. To address these challenges, a spatial and temporal discretization methodology is introducedthat employs an extrapolation technique to select the necessary discretizations for achieving a specific accuracy level while minimizing computation time. This methodology leverages the potential of using a 2D FEM servingas an accurate reference to attain the desired accuracy level for a part of the thermal network. For other parts of the thermal network, the same extrapolation technique is applied to determine an accurate reference. Analysis of this discretization methodology demonstrates that model uncertainties remain within the desired accuracy level, and computation times stay within one minute for accuracy levels from 0.2°C and beyond. As a result, the newly proposed methodology effectively addresses the deficiency related to the selection of node quantity and placement within the thermal network and their impact on model accuracy.

On the Validation of Numerical Simulation with Experimental Results on Compressible Turbulent Flow in an Inertial Particle Separator Device

Linda Bahramian, Ahmad Amani, Joaquim Rigola, Carles Oliet, Carlos David Perez Segarra

Universitat Politecnica de Catalunya

Email: linda.bahramian@upc.edu

The Environmental Control System (ECS) is devoted to maintain the temperature and air pressure within the aircraft's cabin. Conventionally, the inlet air for feeding the compressors of the ECS is bled from the latter stages of the engine, inducing an additional fuel burn that can lead to the ingestion of harmful substances. The alternative approach is to use air coming from the ambient, which may contain Foreign Object Debris (FOD) and can reduce the service life of the compressor. To remove particles from entering the ECS, an Inertial Particle Separator (IPS), which consists of two types of outlets, core and scavenge, can be situated in the ECS's intake. In IPS, a rapid change in flow direction is induced, and particles separate from the core outlet based on their inertia. Small particles with a low Stokes number (St) follow the flow and often enter the core outlet, while the large particles with a high St are thrown into the scavenge by their high inertia. IPS performance has a direct influence on the ECS life; therefore, precise estimation of separation efficiency, defined as the mass of particles leaving the scavenge over the total mass of particles leaving the outlets, is crucial. The objective is to validate the numerical simulation of the compressible flow using Computational Fluid Dynamics (CFD) with the experimental data provided for an IPS device. After validation of the fluid phase, we can analyze the particle separation efficiency for different particle distributions through different scavenge-to-inlet mass flow rate ratios, known as flow splits. It is expected that the particles with St >> 1 show high separation efficiency regardless of flow split values, while the small particles with St << 1 present low separation efficiency sensitive to the flow split value. Finally, the conservation of the thermal energy will be studied.

Pumped Phase-Transitional CO2 Loop for Multi-Component Electronics Cooling

Wessel Wits, Myron Middelhuis, Miguel Muñoz Rojo

Universiteit Twente
Consejo Superior de Investigaciones Cientificas

Email: w.w.wits@utwente.nl

Continuous miniaturization and the demand for increased performance of electronic components require thermal management solutions capable of high power density cooling. Conventional methods such as air and single-phase liquid cooling no longer suffice and novel more effective solutions are in need. Phase-transitional concepts that make use of the latent heat of vaporization and condensation are excellent candidates. In fact, the use of passive two-phase systems, such as heat pipes and vapour chambers, is already common practice in electronics industry. These solutions however lack the ability to simultaneously cool multiple components within an electronics system. In this study, we seek to enhance the cooling performance at the system's level by utilizing a pump to circulate the working fluid in a saturated state. This pumped phase-transitional loop absorbs heat by evaporation thereby cooling hot electronic components and disposes this heat by condensation at a given location away from the component area thereby providing high heat transport capability. In this paper, the cooling performance of a such a loop is discussed and experimentally validated. We demonstrate that CO^2 is a suitable working fluid for the application of electronics cooling. The developed loop allows to connect multiple hot components simultaneously both in a serial or parallel manner. Heat transfer coefficients up to 10 kW/m^2K could be obtained at the component level. A numerical method is presented that is able to predict system's behaviour and simulate thermal performance for a given architecture.

Experimental and Numerical Pressure Drop Investigation of a Protruding Tube Microchannel Heat Exchanger

Carles Oliet, Mohamed Settati, A. Oliva

Universitat Politecnica de Catalunya

Email: carles.oliet@upc.edu

Experimental fluid flow study on micro Heat Exchanger (microHEX) is conducted in this work with the purpose to investigate hydrodynamic flow, pressure drop and friction factor of protruding microchannels model. This consists of the experimental characterization and validation of a new microHEX based on a large protrusion of circular microchannels into manifolds. For this workthe microHEX was manifactured joining a planar bundle of 34 microtubes of 300μm inner diameter and 600μm outer diameter of stainless steel material covered by copper material. The degree of protrusion of microchannels inside each of inlet and outlet manifolds is of 70%. The interface of the microHEX chip is approximately 4cm^2. The measurements were performed for distilled water, and laminar Reynolds number of 300-2200. Considering hydrodynamic developing flow the results found were contrasted using step-by-step fast numerical model for pressure drop and friction factor. The experimental pressure drop and apparent friction factor were corrected taking into account the losses of experimental circuit and the losses due to sudden contraction and expansion into microchannels. The relative mean error of the present results with respect to step-by-step calculation is 9.18%. The analysis of the results revealed that the flow exhibited a premature characteristics of transition regime in 700≤ Re ≤1600 as found by several authors. This work also discuses and reveals the validity of the available correlations of sudden contraction and expansion losses coefficients for protruded microchannels.

Low Enthalpy Geothermal Source for Sustainable Energy Production in Small Islands: A Real Case Study

Simona Di Fraia, Nicola Massarotti, Laura Vanoli

Universita degli Studi di Napoli Parthenope

Email: simona.difraia@uniparthenope.it

The global demand for clean and sustainable energy sources has significantly increased in recent years, mainly because of concerns about climate change and the finite nature of traditional fossil fuel reserves. In this context, geothermal energy has gained significant attention as a reliable and eco-friendly alternative. In particular, low enthalpy geothermal resources have emerged as an attractive option due to their accessibility, widespread distribution, and potential for decentralization. Moreover, reliability and flexibility of this source allow for year-round utilization to meet diverse energy demands, resulting in reduced dependency on traditional energy sources and associated greenhouse gas emissions. For this reason, geothermal energy appears to be a promising solution in small islands, where energy production is often a challenge due to the absence of infrastructures that make them energy-dependent on the mainland. Therefore, in this work, low enthalpy geothermal source is considered as effective and sustainable solution for energy production in small islands. As case study a low temperature source available in Ischia island, Southern Italy, is considered. In this work the possibility of producing electrical energy is assessed. Indeed, although the extracted fluid's low temperatures pose challenges for electricity generation, the main issues related to geothermal applications lie in the high investments and complex bureaucratic processes and authorizations compulsory for wells perforation, that in this case is not needed since the considered user already possess the wells. An Organic Rankine Cycle, due to its suitability to exploit low-medium source temperatures and the high effectiveness in small-scale applications, is considered as technology for electricity production. The proposed solution is analyzed from energy, economic and environmental point of view.

Multiscale System-Level Modelling of a High-Voltage Battery Pack Thermal Management System for Aerospace Applications

Jernej Pirnar, Ambrož Vrtovec, Tina Novak, Tibor Van Steenis

Pipistrel Vertical Solutions d.o.o.

Email: jernej.pirnar@pipistrel-aircraft.com

Electric propulsion systems represent one of foremost research areas in the effort to reduce the carbon footprint associated with air travel. These technologies offer several benefits, such as lower environmental impact of air travel, lower noise, and lower operating costs. Battery energy storage systems are essential enabler of an electric powered flight. Battery cell is a fundamental building block of a battery and its performance and safety depend on multiple factors, the temperature being one of the most important. Battery cells generate heat during charge and discharge modes, increasing their temperature. Therefore, a battery thermal management (sub)system (BTMS) is crucial for a reliable, optimal, and safe operation of an airborne battery pack. Temperature affects battery pack available capacity and lifespan, however, controlling the cell temperature is especially critical in preventing a thermal runaway, a dangerous phenomenon that may cause a fire on board the aircraft. The optimal temperature for lithium-ion batteries is recommended to range between 20 °C and 40 °C. Hence, the design of BTMS is challenging, especially because temperature envelope of aircraft may exceed 80 °C (difference between highest and lowest ambient temperature). For defining the BTMS architecture and verifying requirements, a system-level simulations supported by a high-fidelity CFD simulations are invaluable. Making a battery system decomposition, it becomes evident the multiscale modelling should be applied to acquire reliable simulation outcomes. One of the most relevant indicators about the design might be the battery temperature variations experienced across various stages of the aircraft's mission. Finally, it is crucial to contemplate the functionality of control logic and diverse failure scenarios already in the initial phases of the BTMS design process. The demonstrated approach has been verified to be suitable, essential, and computationally efficient for designing and optimizing a BTMS in an industrial setting, being proven within the NEWBORN project.

Session: Computing

Chair: Matjaž Hriberšek

Mitigating the Onset of Flow Separation Over Wind Turbines Blades at Low Reynolds Numbers: Numerical Conceptualization and Validation

Luca Pagliarini, Roberto Corsini, Enrico Stalio, Fabio Bozzoli

University of Parma
University of Modena
Universita degli Studi di Modena e Reggio Emilia

Email: luca.pagliarini@unipr.it

Reducing the role of fossil fuels on the global energy production is crucial for climate change mitigation. Renewable energy is considered as one of the primary solutions for moving towards carbon-neutrality. Systems based on wind energy harvesting can successfully meet part of the increasing green energy demand worldwide. However, wind turbines operation might be undermined by varying atmospheric conditions, which could result in an increase of angle of attack and consequent onset of stall phenomena. In particular, for similar working conditions, the occurrence of flow separation is mainly driven by the geometry of the blades. For this reason, it is crucial to define a tool capable of swiftly allowing numerical investigations on many different blade geometries. Hence, the present work aims at modelling laminar-turbulent transition and turbulent flow separation over a wind turbine blade section operating at angle of attack = 15° and Re = 66000 (chord-based) by means of a steady RANS approach. Turbulence is treated by means of the Transition SST k-ωmodel. The adopted model is successfully validated through data provided by a Direct Numerical Simulation, thus confirming its good accuracy and suitability for future investigations on a wide range of geometries.

Numerical Simulation of Drop Impingement onto Superheated Textured Walls

Henrik Sontheimer, Anh Tu Ho, Leon Elsässer, Peter Stephan, Tatiana Gambaryan Roisman

Technische Universitat Darmstadt
Institute for Technical Thermmodynamics

Email: sontheimer@ttd.tu-darmstadt.de

Spray cooling is a very efficient method for thermal management of high-performance electronics. Spray cooling performance is strongly influenced by the fluid dynamics and heat transport mechanisms during the impingement of a single drop onto superheated walls. The dominant mechanisms involved in single drop impingement onto superheated, smooth walls are largely understood. However, real-world surfaces are usually not ideally smooth. In addition, textured surfaces are known to improve heat transfer during spay and drop impingement. In this work, we numerically study the single drop impingement onto textured walls. Simulations are performed with a solver for incompressible fluids using the volume-of-fluid method in OpenFOAM. The refrigerant used is perfluorohexane (FC-72) in a pure vapor atmosphere under saturation conditions. The wall temperature is above the saturation temperature of the fluid but below the onset of nucleate boiling. In a first step, we study the fluid dynamics during an isothermal drop impact on four different surface structures: cubes, pyramids, and grooves with rectangular and triangular cross-sections. All of the surfaces studied increase both the wetted area and the length of the three-phase contact line compared to a single drop impinging onto a smooth surface. Small, flat structures generally lead to larger wetted areas and longer three-phase contact lines than large, high structures. In a second step, we consider heat transfer and study the drop impingement on cubes and rectangular grooves. It is shown that surface textures that lead to a larger wetted area and a longer contact line length result in higher heat transfer rates. The results contribute to a better understanding of the fluid dynamics and heat transport during drop impingement, which will allow in the future to optimize the heat transport during spray impingement by designing the surface texture with optimal geometric parameters.

Building a Reduced Order Model from CFD Data on a Full Plant To Evaluate Optimal Climate Conditions

Wito Plas, Michel De Paepe, Toon Demeester

Universiteit Gent

Email: wito.plas@ugent.be

Climate control in vertical farms or greenhouses is crucial in order to have the optimal climate for the plants. This climate can be simulated using Computational Fluid Dynamics or CFD. This way, different ventilation methods can for example be compared. Previously, plants were modeled as a porous zone, in which additional source terms are added to the energy or transpiration transport equation. In this study, a more realistic plant geometry is used to model the plants inside this enclosure. Multiple plants are being ventilated sideways using a uniform flow, but also by using different kinds of ventilation concepts, such as jet flow. This realistic plant geometry is comprised of multiple leaves and is based after a basil plant. The heat and mass balance for each leaf is solved and the boundary layer of the leaves is solved. Leaf temperature and humidity are a function of the incoming flow, but also of the incoming temperature, humidity and stomatal resistance. In order to be able to quickly find different values for leaf temperature and humidity, a Reduced Order Model (ROM) is made from the data. This method allows to quickly assess different inlet humidities, temperatures and stomatal resistances. The ROM was validated for other climate parameters and conditions inside the plant factory and proved to be an effective tool for analysing the local climate inside the plant factory.

Numerical Modelling of Heat and Mass Transfer During Cake Baking

Patrick El Helou, Pascal Le Bideau, Adrien Fuentes, Patrick Glouannec

Universite Bretagne Sud
Universite de Bretagne Occidentale

Email: patrick.el-helou@univ-ubs.fr

This article deals with the development of a numerical multiphysic model to study heat and mass transfer phenomena as well as the swelling during the baking of a cake contained in mould. The aim of this study is to provide an effective numerical tool, experimentally validated, for a better understanding of mechanisms leading to the desired end product. In this approach, the medium is assumed to be a deformable porous medium containing three phases: solid (dough), liquid (water) and gas. Gas phase includes two species, water and carbon dioxide (released by the leavening agent). Based on the governing equations for heat and mass transport and under few assumptions (homogenous medium, local thermodynamic equilibrium, gas phase assumed to be an ideal gas mixture), the problem consists in solving a system of five coupled partial derivative equations. The state variables are the temperature, the moisture content, the total gas pressure, the porosity and the displacement. The swelling of dough caused by the increase of total gas pressure is predicted by a viscoelastic model. This thermo-hydro-mechanical model is implemented in finite elements code. At the same time as numerical approach, experimental tests are carried out on a laboratory oven. In this context, an experimental laboratory set-up was developed in order to continually acquire temperatures, water loss and to correctly apprehend the boundaries conditions. The cake deformation is also tracked by camera. The numerical results are compared with experimental data and analysed. Various operating conditions are tested to check the robustness of predictions.

Numerical Investigation of Methane-Air Jet Flame with Hydrogen Addition in Industrial Kiln Burners

Jiannan Liu, Joaquim Rigola, Eugenio Schillaci, Eugenio Schillaci, Carlos David Perez Segarra

Universitat Politecnica de Catalunya

Email: jiannan.liu@upc.edu

Emission reduction and decarbonisation in the industry are crucial for low-carbon industry and energy transition at the global level. Replacing traditional fossil fuel with clean energy is an effective approach to reduce carbon emissions and optimize energy efficiency in manufacturing processes. Industrial kiln, which requires high natural gas fuel consumption, typically releases amounts of harmful combustion products. In this paper, the objective is to study the influence of hydrogen addition into methane-air jet flame in industrial kiln burners. The industrial burner analysed in this study is a cylinder vessel with axial orifices and swirl turbulent co-flow air jets in the fuel-air inlet structure. The fuel is radially injected from the nozzle hole into the vessel and mixed with air. Then, the combustible mixture is ignited and propagates towards the outlet of the burner into the outside domain. A more recent reduced chemical kinetic mechanism for methane-hydrogen combustion is utilized in the present flame simulation and validated in benchmark flames. The chemical mechanism involves 45 reactions and 18 species. Various methane-hydrogen blending fuels are studied in the jet flame, where flame structure and flame characteristics including chemical species, temperature, and velocity are critically tested. Furthermore, the influence of the proportion of air mass flow in the orifices and swirl inlet holes is also discussed.

Ensemble Averaging Parallel-in-Time Approach for Industrial LES

Josep Plana-Riu, Francesc Xavier Trias Miquel, Àdel Alsalti-Baldellou, Guillem Colomer, A. Oliva

Technical University of Catalonia

Email: josep.plana.riu@upc.edu

Real-world computational fluid dynamics (CFD) simulations demand not only robust and stable numerical methods, but also to be completed in a limited time. For this reason, industrial applications usually rely on RANS modeling, which have some remarkable limitations. Alternatively, using Large Eddy Simulations (LES) leads to higher fidelity simulations, while having much higher computational costs. CFD simulations are virtually always carried out with spatial parallelization, which, if exploited to a bigger and bigger number of cores will eventually lead to a saturated system and thus, using more cores will not provide a further speed-up. In order to deal with this, some concepts such as the Multigrid Reduction In Time (MGRIT) have been proposed as a way to extend the parallelization to both space and time domains. On the other hand, most CFD simulations are based on sparse matrix-vector products (SpMV) or SpMV-like stencil-based kernels, which have a low arithmetic intensity, i.e. the ratio of floating-point operators is low compared to the data traffic. This sets the bottleneck of CFD simulations in data transfer instead of in computational power. In order to deal with this, ensemble averaging parallel-in-time approaches consist on running a set of shorter simulations. Afterwards, the average result among this set of simulations is computed instead of running a single long simulation. In this work, this set of simulations is proposed to be run at once on the same device. This allows an increase of the arithmetic intensity, as the operation requires less data transfer, and it eventually generates speed-up. This will be tested in canonical flow and heat transfer cases targetting concentrated solar power (CSP) simulations, on both CPU and GPU-accelerated nodes, in order to prove that this method can lead to improvements in the efficiency of simulations and thus moving towards reliable industrial overnight LES simulations

On the Feasibility of Overnight Industrial High-Fidelity Simulations of CSP Technologies on Modern HPC Systems

Àdel Alsalti-Baldellou, Guillem Colomer, Johannes Arend Hopman, Xavier Álvarez Farré, Andrey Gorobets, Francesc Xavier Trias Miquel, Carlos David Perez Segarra, A. Oliva

Technical University of Catalonia
Cooperatie SURF UA
Russian Academy of Sciences

Email: adel.alsalti@upc.edu

In the last decades, computational fluid dynamics (CFD) has become a standard design tool in many fields, such as the automotive, aeronautical, and renewable energy industries. The driving force behind this is the development of numerical techniques in conjunction with the progress of high-performance computing (HPC) systems. Legacy numerical methods chosen to solve (quasi)steady problems using RANS models are inappropriate for more accurate (and expensive) techniques such as large-eddy simulations (LES) or direct numerical simulations (DNS). The main recognised limitations of LES in the industry are their computational cost and wall-clock simulation time. Thanks to the advent of new computational architectures, the former is becoming less and less critical, whereas the latter is still the most limiting factor precluding LES from being routinely used in the industry. For that to be possible, the consensus is that widespread adoption in the industry begins when a run can be carried out overnight. Namely, the industry is governed by shortening design cycles, faster time-to-market, and increased expectations of operability and reliability for established product lines. Therefore, it is willing to spend on hardware and software as long as analysts can obtain meaningful insights in a time commensurate with design cycles. In this context, this work assesses the feasibility of overnight LES simulations on GPU-accelerated supercomputers with TFA, our novel in-house code, which relies on a symmetry-preserving discretisation for unstructured collocated grids that, apart from being virtually free of artificial dissipation, is shown to be unconditionally stable. The cases of study will be taken from central receivers used in concentrated solar power (CSP) plants, and a comparison with open-source CFD codes will be made.

A Preliminary Model of Water and Salt Transport in a Laboratory Scale Pressure-Retarded Osmosis (PRO) Module

Piotr Łapka, Juliusz Wachnicki, Piotr Furmański

Warsaw University of Technology

Email: piotr.lapka@pw.edu.pl

Nowadays, more and more attention is being paid to producing green energy. Recently, energy from salinity gradients has also attracted interest. One such process that can generate useful work is the pressure-retarded osmosis (PRO), which uses a semi-permeable membrane to separate the feed and pressurized draw solutions. The semi-permeable membrane allows the transport of water from the feed (low pressure) side to the draw (high pressure) side, and thus, the excess of pressurized water on the draw side can be used to generate power, i.e., by expansion in the turbine. This work presents a preliminary numerical model developed to study the water and NaCl salt transport through the semi-preamble membrane and in the PRO module designed to perform experimental studies. The model was first verified for simple 2D module geometry. It was then used to study the flow through a real 3D module used in laboratory-scale experiments. The influence of the hydrodynamic pressure and the NaCl draw solution's mass flow rate on the energy generation efficiency was examined. The maximum power density obtained for half the osmotic pressure of the NaCl draw solution (i.e., for 28 bar) was found to be equal to approximately 4 W/m^2. An increase in the flow rate of the draw solution causes a decrease in the boundary layer at the membrane, eliminating the effect of the external concentration polarization. It causes an increase in the concentration difference on either side of the membrane, contributing to a non-linear increase in power density depending on this mass flow rate. Increasing the average velocity from 0.02 m/s to 0.1 m/s increases the power density by up to 80%, from approxymetly 2.6 to 4.7 W/m^2.

Session: Experiments

Chair: Michel Quintard

Effect of Acoustics on Droplet Grouping Behaviour in a Single Stream of Droplets

Manish Kumar, Visakh Vaikuntanathan, Bernhard Weigand

Shiv Nadar Foundation
Universitat Stuttgart

Email: manish.kumar@itlr.uni-stuttgart.de

Droplet and particle grouping can be influenced by applying an acoustic field and have practical applications such as particle scavenging and aerosol filters of engine exhaust and air purifiers. Previously, several studies have been done to understand the acoustic field effect on the coupling of droplet characteristics (i.e., droplet size, droplet velocity, and number density of droplet) and combustion instability. The present work experimentally investigates the influence of a standing acoustic wave on a single stream of droplets. The experimental setup consists of an acoustic transducer and a reflector plate through which the droplet stream passes in the presence or absence of an external pressure field generated by a standing acoustic wave. A droplet stream is generated with the help of a nozzle connected to a pressurized working fluid supply and piezoelectric transducer to control the spacing between droplets. The effect of the acoustic pressure field on the droplet stream generated by different sizes (i.e., diameter) of nozzles at different piezoelectric excitation frequencies and fluid pressure is investigated. Droplet stream characteristic at every nozzle excitation frequency is observed with a high-speed camera when the acoustic field is switched OFF and ON. The competing effect of nozzle excitation frequency and acoustic field is observed. At lower nozzle frequencies, an unstable stream of droplets is generated by the nozzle having different sizes and spacing between them. When the acoustic field is applied for these lower frequencies, the stream of droplets becomes organized, and in some cases, it becomes equispaced and of the same size. However, an opposite behavior has been observed at higher frequencies. In these cases, an equispaced stream of same-size droplets merge and show coalescence phenomena, and the droplet stream becomes unstable as the acoustic field is applied.

Experimental Study of Erosion in Heat Exchangers Immersed in a Fluidized Bed

Pedro Domínguez-Coy, Juan Ignacio Córcoles-Tendero, José A. Almendros-Ibáñez

Universidad de Castilla-La Mancha, Instituto de Investigación en Energías Renovables

Email: pedro.dominguez@uclm.es

Fluidized beds can be used as solar energy receivers in beam-down concentrated solar power plants. The fact that solid particles can withstand temperatures over 1000 °C is promising because it would mean higher thermal efficiencies than conventional concentrated solar power plants (CSP, that use molten salts, whose maximum operating temperature is limited to about 560 °C). Nonetheless, immersed heat exchangers are exposed to erosion because fluidized particles impinge on their surfaces. The multiple variables of a multiphase flow and the uncertainties of the material detachment process make erosion a complex problem that stills not fully solved. The aim of this study is to conceive an experimental procedure to measure erosion on cylindrical probes immersed in a lab-scale high temperature fluidized bed (able to operate up to 1000 °C and atmospheric pressure), simulating a tube of a heat exchanger in a fluidized bed working at expected temperatures in CSP applications. Preliminary results at ambient temperature are presented in this study. Silica sand particles were fluidized with air and erosion was measured on cylindrical probes made of mullite, a soft ceramic material. Silica sand was characterized with a microscope before and after each test to determine its size distribution and shape. Several parameters (length, diameter, mass, and circumferential profile) of each probe were also studied before and after each test. Different fluidization velocities were implemented to assess the influence of the flow rate on erosion.

X-Ray Computed Tomography Tracking of Calcium Chloride Hexahydrate Crystallisation Process

Dario Guarda, Jorge Martinez Garcia, Benjamin Fenk, Damian Gwerder, Anastasia Stamatiou, Jörg Worlitschek, Simone Mancin, Philipp Schütz

Universita degli Studi di Padova
Hochschule Luzern
Lucerne University of Applied Sciences and Arts

Email: dario.guarda@phd.unipd.it

The research conducted on phase change materials (PCMs) for latent thermal energy storages (LTESs) is continuously growing in terms of publications, pointing out the importance of this topic. In fact, PCMs present many advantages that could help the energy transition and reduce CO2emissions, by enhancing the performance of existing systems and better exploiting renewable energy. Therefore, it is of crucial interest to develop new and reliable methods to control LTES. Differently from sensible thermal energy storages, in LTESs the stored thermal energy is not proportional to the temperature. To really have an insight into the level of charge of these storages, it is important to know the liquid fraction, i.e., the amount of the liquid phase with respect to the whole amount of PCM. X-ray computed tomography (XCT) is a technology that allows to non-intrusively “look inside” the materials and, in the current study, it was used to analyse calcium chloride hexahydrate crystallization phenomenon. This transient process of calcium chloride hexahydrate was tracked with many XCT scans, one every 6 minutes, resulting in 3D image stacks that were processed to obtain the volumetric liquid fraction evolution over time. Repeatability tests were run to evaluate the reliability of the XCT technique and the volumetric liquid fraction data was used to validate a numerical model developed within ANSYS Fluent framework. XCT offers great opportunities to study the heat and mass transfer mechanisms underlying the main issues of phase change materials, like, for example, supercooling and salt hydrate segregation.

Experimental Investigations of Hydrogen Iodide (HI) Decomposition Process for Different Catalysts and Various Temperature Conditions

Patrycja Kowalik, Paweł Bocian, Piotr Jozwiak, Jarosław Hercog, Krzysztof Badyda

Rzeczpospolita Polska

Email: patrycja.kowalik@ien.com.pl

This article presents the results of experimental work carried out at the Institute of Power Engineering on the process of thermal decomposition of hydrogen iodide. This process is one of the three key steps taking place in sulfur-iodine (S-I) thermochemical hydrogen production technology. For this purpose, a laboratory test rig equipped with an electrically heated chemical reactor was constructed, enabling the study of hydrogen iodide decomposition under controlled conditions. The research was carried out using various catalytic substances, as well as different temperature conditions. The process of hydrogen iodide decomposition required keeping the temperature of the reactor above 300°C and continuous analysis of the concentration of hydrogen formed over time. The results of the study made it possible to determine the kinetics of the hydrogen iodide decomposition reaction under selected conditions of the process, allowing for future optimization of the process with the use of numerical methods.

Design of an Ultrahigh Temperature Liquid Metal Centrifugal Pump for Thermal Energy Storage

Kyle Buznitsky, Asegun S Henry, Mehdi Pishahang

Massachusetts Institute of Technology

Email: kylebuz@mit.edu

Liquid metals serve as efficient heat transfer fluids due to their exceptional thermal conductivity and broad temperature range, rendering them well-suited for demanding applications such as nuclear reactors and thermal energy storage. Nevertheless, designing liquid metal pumps poses significant challenges, including chemical compatibility with the containment vessel and various thermomechanical issues. This study focuses on elucidating these challenges and proposing potential solutions through the example of a centrifugal pump designed for handling liquid tin at an extreme operating temperature of 2400°C. To address chemical compatibility concerns, graphite is adopted as both the containment and pump material, presenting novel mechanical hurdles. Key obstacles include establishing pressure-tight liquid seals, accommodating thermal expansion, and navigating the intricacies of working with features approaching the grain size of graphite.

Experimental Investigations of the Potential of Organic Salt Solutions for Application in Pressure-Retarded Osmosis

Piotr Łapka, Fabian Dietrich, Łukasz Cieslikiewicz, Piotr Furmański

Warsaw University of Technology

Email: piotr.lapka@pw.edu.pl

Pressure retarded osmosis (PRO) enables useful work generation from the salinity difference of solutions. The literature most often considers using PRO with natural sodium chloride (NaCl) solutions such as seawater and river water or industrial solutions such as seawater desalination concentrate. Open cycle systems, which are dependent on the available source of streams feeding the system, are most often analyzed. In such systems, the full PRO potential as a power generation method cannot be explored due to the existing limitations of natural and industrial NaCl solutions. The second approach, found less frequently in scientific publications, is closed-cycle systems. They allow more extensive manipulation of the feed and draw stream parameters and the use of optimized synthetic solutions. This allows optimization of the process precisely for the intended applications. Furthermore, engineered solutions of various compositions can be used in closed-cycle systems, which opens new ways of increasing the system performance and application potential. The research described in this article is focused on the experimental evaluation of PRO potential for generating useful work by applying solutions of organic salts under varying conditions. The chemical compounds used were selected based on their theoretical performance, ease of use, and availability. The developed experimental stand allows full control over the temperature of the feed and draw solution streams. The studies were conducted without back-pressure, in a configuration typical of the forward osmosis process, with solution circulation on both sides of the membrane. As a result, the effect of solution temperature and concentration on water flow through the membrane was determined, and permeate flow is directly related to achievable work in the PRO process. The obtained results were compared to the system performance with NaCl solutions under the same conditions to evaluate potential gains. Work was supported by the National Science Centre (Poland), project no. 2021/43/B/ST8/02968.

Experimental Analysis of Flow-Focused Micro-Jets

Jernej Kušar, Krištof Kovačič, Bor Zupan, Gal Savšek, Saša Bajt, Božidar Šarler

University of Ljubljana
DESY
Institute of Metals and Technology, Ljubljana, Slovenia

Email: jk5259@student.uni-lj.si

This experimental study aims to investigate the characteristics of gas flow-focused jets. Using water as the focused liquid and air as the focusing gas, we varied the nozzle inner capillary diameter, outer orifice diameter, and gas and liquid flow rates to observe their effects on the generated jets. Nine different configurations of nozzle designs were tested under different operating conditions to produce the jetting mode, resulting in a total of 251 measurements. The gas and liquid Reynolds number ranges from 8 to 500 and from 76 to 558, respectively. The jet Weber and Capillary numbers range from 1.3 to 15 and 0.009 to 0.045, respectively. The average pressure drop within the nozzle varies in the 4 to 105 mbar range. The diameters of produced jets are in the range of 30.4-314.2 µm and jet breakup lengths 0.48-6.42 mm. This study analyses the flow-focused jets on a 200-1000 µm nozzle length scale, which fills the gap in the existing literature. The experimentally measured results agree with the non-dimensional scaling law from the previous studies on a smaller length scale. As proposed in the previous study, the theoretical jet diameter was calculated for analysed measurements. The discrepancy between the theoretical and experimental diameter was in the 0-16 % range, with a maximum value of 46 %. We attributed the deviation from the established theory to the influence of surface tension, whose effects should be additionally considered for flow-focused jets on the analysed length scale.

Visualization of Inclusions in a Water Model Steel Continuous Casting of Steel

Matjaž Perpar, Katarina Mramor, Blaž Hodej, Matic Cotič, Robert Vertnik, Vid Vengust, Božidar Šarler

Faculty of Mechanical Engineering, University of Ljubljana
Štore Steel d.o.o.
Institute of Metals and Technology, Ljubljana, Slovenia

Email: matjaz.perpar@fs.uni-lj.si

With the water model, we experimentally confirmed the possibility of using a suspension to imitate the molten casting powder (slag) at the upper edge of the melt during the continuous casting of steel. The project requires that the slag particles do not leave the suspension layer during operation or that they return to it from the molten steel. It turns out that this is possible if the selected suspension particles mimic sufficiently large inclusions in the real system. The imitation of slag particles in a real system with suspension particles in water model is ensured by the equal terminal flotation velocities of both. Among the discussed polyethylene and cork particles, only cork particles, with a size of 0.2 mm to 0.5 mm, and a density of around 400 kg/m^3, meet this condition. It was shown that the presented methodology could be used to estimate the proper position of the submerged entry nozzle.

Elevated Temperatures and Prestresses on Evolving Yield Surfaces for Modeling Experimental Data

Hong-Ki Hong, Kai-Min Hou

National Taiwan University

Email: hkhong@ntu.edu.tw

Phillips and his associates obtained a series of experimental data from thinwalled tubular specimens of commercially pure aluminium 1100-0 at room and at elevated temperatures. These data were neatly recoded in many aspects, executed faithfully in stresscontrolled experiments and well known already for five decades; however, modeling these experimental data encountered either tremendous difficulties or over complications regardless of various attempts. In this paper selecting prominent test evidences based upon experiences on aluminium alloy Al6061 gained in our lab [ASCE Journal of Engineering Mechanics 148(6):04022027, 2022] over the course of the years, we created a three-dimensional tensor model of flow elastoplasticity, grasping all axial-torsional experimental features reported in the literature; in particular, in Phillips et al. at room and at elevated temperatures. For each temperature the model needs a total of 8 material constants in addition to Young's modulus and shear modulus and presents an evolving cubic distortion yield hypersurface, which is articulated with two Mises hyperspheres, characteristic of internal symmetry of two elements of the projective proper orthochronous Poincare group in the plastic phase. Associated with each Mises hypersphere in stress space is a normality plastic flow rule of mixed-exp-AF, referring to a combined isotropic-kinematic rule of hardening-softening, which combines the isotropic exponential rule of degree 2 and the kinematic rule of Armstrong-Frederick. By using the model and employing Lie group theory, closed-form exact solutions are derived and used to identify a unique set of parameters for fitting successfully evolving shapes of yield surfaces with clear physical meaning.

Session: Fluid Flow

Chairs: Michel De Paepe, Karl Ponweiser

Numerical Contact Line Behavior Prediction for Drop-Wall Impact Using Basilisk

Maxim Piskunov, Ivan Vozhakov, Sergey Misyura

Nacional'nyj issledovatel'skij Tomskij politehniceskij universitet
Pravitel'stvo Rossijskoj Federacii
Kutateladze Institute of Thermophysics

Email: piskunovmv@tpu.ru

Drop-wall impact and spray cooling have a wide technical application. There are still fundamental problems associated with wettability and its effect on the drop spreading and fingering. The influence of dynamic and static contact angle on the evolution of a drop of homogeneous liquid falling on a smooth solid surface is examined. Experiments and simulations are performed in a wide range of Weber numbers (We = 1-360). The 3D simulation is implemented by solving the incompressible Navier-Stokes equations along with the VOF method in the Basilisk software. An adaptive mesh refinement near the interfacial surface provides a cell size of 5 μm. The effect of the static contact angle on the velocity and the nature of the drop stretching on the surface is explored. The contact angle affects the velocity and acceleration of the contact line, which in turn determine the wavelength and amplitude of the finger-like instability near the contact line. In addition, to determine the contact angle, the Hoffman function is used, which satisfies the Hoffman-Voynov-Tanner law on the dependence of the contact angle on the capillary number. Usually, for estimates of the maximum drop spreading diameter, the drop velocity before impact on a wall is taken. The research shows that to calculate the number of "fingers" and the dissipation rate, it is correct to take the average contact line velocity. Critical disturbances leading to the formation of fingers occur at small times (about 0.5 ms), significantly less than the time of maximum drop spreading. When approaching the maximum diameter, the contact line velocity has an oscillatory character. Considering the wetting hysteresis makes it possible to better describe the experimental data during the transition from the drop spreading to its receding. The work was funded by the Russian Science Foundation grant No. 23-71-10081.

Mixed Convection Flow Over a Horizontal Plate and the Horizontal Wake Far Downstream

Lukáš Bábor, Wilhelm Schneider, Endre Bozsó

Technische Universitat Wien

Email: lukas.babor@tuwien.ac.at

The mixed convection flow over a heated or cooled horizontal plate of finite length is a surprisingly intriguing problem. The hydrostatic pressure jump at the trailing edge and across the wake is to be compensated by induced circulation in the outer potential flow, similar to a plate at a non-zero angle of attack. The mixed-convection flows over semi-infinite and finite horizontal plates, respectively, are thus substantially different. As the hydrostatic pressure jump remains finite along the infinitely extended wake, the induced outer potential flow around the plate would grow beyond bounds in a domain extending to infinity. Thus, boundary conditions have to be imposed at the boundaries of a bounded domain. Several suitable boundary conditions will be tested, and their effect on the pressure distribution and heat transfer at the plate will be determined. However, the numerical solution of the Navier-Stokes equations remains a challenging problem due to the lack of appropriate outflow conditions. The traditional outflow condition cannot sustain the finite hydrostatic pressure jump across the wake. Thus, an asymptotic expansion for the wake far downstream from the plate is performed. Remarkably, the perturbation of the flow does not decay with increasing distance from the plate as in a classical wake. Furthermore, the viscous drag of the plate enters the asymptotic solution only as a higher-order correction. A self-similar solution that exists only within a certain range of the control parameters is obtained. In a certain part of this range, two different solutions exist. The second solution is characterized by a non-monotonic velocity profile, leading to flow reversal. The asymptotic solution is implemented as an outflow boundary condition for the numerical solution of the Navier-Stokes equations. The results are compared with boundary-layer solutions obtained in previous work for the limiting case of vanishing Prandtl number.

Computational Fluid Dynamics Modelling of Large-Scale Bubble Columns: From the Mono-Dispersed Homogeneous to the Pure-Heterogeneous Flow Regimes.

Nicolò Varallo, Giorgio Besagni, Mereu Riccardo, Inzoli Fabio

Politecnico di Milano

Email: 10608345@polimi.it

Bubble columns are multiphase reactors widely used in the chemical, biochemical, petrochemical, food, and pharmaceutical industries because of the several advantages offered. Despite the simple reactor configuration, bubble column fluid dynamics is extremely complex, and its understanding at the different scales is crucial for optimising the performance of these systems at given operating conditions. In this work, a CFD Eulerian multi-fluid approach is developed to describe the hydrodynamics of large-scale bubble columns. Transient three-dimensional simulations have been performed employing a commercial code (ANSYS Fluent 2021 R2) and the numerical results have been compered with available experimental data. The superficial gas velocity ranges between 0.0037 m/s and 0.2 m/s, covering both the mono-dispersed and pure-heterogenous flow regime, where bubbles coalescence and breakup has been modelled. In particular, this study contributes to the present discussion on the modelling closures for the momentum source term within the Eulerian multi-fluid approach, deeply analysing the role of the lift force. In this regard, different lift coefficient correlations have been considered and modified with a more suitable correlation for the bubble aspect ratio to predict the lift force. In addition, the influence of the turbulent dispersion force on the local void fraction profiles has been studied. The results have been critically analysed, and the discrepancies between the numerical and experimental results have been deeply commented on, setting the stage for future improvements.

Unstructured Conservative Level-Set (UCLS) Method for Reactive Mass Transfer in Bubble Swarms at High Density Ratio

Nestor Vinicio Balcazar Arciniega, Joaquim Rigola, A. Oliva

Universitat Politecnica de Catalunya

Email: nestorbalcazar@yahoo.es

This research presents a parallel Unstructured Conservative Level-Set (UCLS) method for reactive mass transfer in bubbles with a high-density ratio. This method uses the multiple-marker approach to circumvent the numerical coalescence of bubbles, which is a potentially unphysical artefact of interface capturing methods. This approach employs the finite-volume method to discretize transport equations on 3D collocated unstructured meshes. The fractional-step projection method solves the pressure-velocity coupling. The central difference scheme discretizes the diffusive term. The discretization of the convective term within the momentum transport equation, level-set advection equations, and mass transfer equation are performed by unstructured flux-limiters schemes. Indeed, numerical diffusion is minimized, whereas numerical oscillations around the interface are avoided. Such a combination of numerical techniques preserves the numerical stability in bubbly flows with a high Reynolds number and high-density ratio. Numerical and physical findings on the effect of physical properties ratios on the reactive mass transfer are reported, including their effect on the computation of interfacial area, Sherwood number and drag force coefficient.

Multiphase Flow Simulation in Hybrid Porous Structure

Majid Eshagh Nimvari, Michael Gibbons

The University of Dublin Trinity College

Email: majid.eshagh@gmail.com

Porous structures are ubiquitous in a range of applications, including electronics cooling, water harvesting, combustion, acoustics and flow control. They enable passive fluid transportation and ultra-high heat transfer. Traditional porous structures are monoporous in design. However, recent research has focused on the fabrication of hybrid porous structures consisting of multiple pore sizes and length scales to optimise capillary pressure and permeability. Despite several experimental investigations on hybrid porous structures, there is a lack of numerical simulations that explore the multiphase flow within these media. This research is focused on numerically simulating a hybrid porous structure. The hybrid structure is realised using a staggered regular arrangement of clusters, with each cluster comprising closely packed solid circles. The dimensions and characteristics of the simulated structure are derived from previous experimental hybrid porous media literature. Numerical simulations of capillary pumping flow are performed utilising the ANSYS Fluent software. To better compare, simulations are conducted for both mono-porous and hybrid porous structures and the effect of various flow conditions and geometry parameters, such as porosity, pore and particle diameters, porous layer thicknesses and gravity effect are explored. The obtained simulation results demonstrate that smaller pathways within each cluster of a hybrid porous media enhance the capillary effect compared to conventional mono-porous porous structures. Furthermore, the larger pores between the clusters and the overall higher porosity contribute to a greater permeability of the hybrid porous structure. Consequently, the combined effect of increased capillary action, higher permeability, and higher porosity results in the improved performance of the hybrid porous structure.

Study of Turbulent Wavy Annular Flow Inside a 3.4 mm Diameter Vertical Channel by Using the Volume of Fluid (VoF) Method in OpenFOAM

Emanuele Zanetti, Arianna Berto, Stefano Bortolin, Mirco Magnini, Davide Del Col

Technische Universiteit Delft
Universita degli Studi di Padova
University of Nottingham

Email: e.zanetti@tudelft.nl

In annular downward flow, an annular liquid film flows at the perimeter of the channel pushed down by the gravity force and by the shear stress that the vapor core exerts on it. Depending on the working conditions, the vapor-liquid interface can be flat or rippled by waves. The knowledge of the liquid film thickness is very important for the study of annular flow condensation because the thermal resistance of the liquid is often the most important parameter controlling the heat transfer. A new approach for the simulation of annular flow is here proposed using an in-house developed transient solver based on the Volume of Fluid (VOF) adiabatic solver interIsoFoam available in OpenFOAM. With the VoF method, in addition to the standard set of equations (continuity and momentum), a transport equation related to the advection of the volume fraction scalar field has to be solved. The numerical setup consists of 2D axisymmetric domain. An adaptive mesh refinement (AMR) method is added to the solver to better capture the interface position. The k-ω SST model is used for turbulence modelling in both the liquid and vapor phases and a source term (whose magnitude is controlled by a model parameter named B) is included in the ω equation to damp the turbulence at the interface. Simulations are run with refrigerants R245fa and R134a flowing inside a 3.4 mm circular channel at respectively 40 °C and 30 °C saturation temperature, mass velocity G= 100-150 kg m^{-2} s^{-1} and 0.3-0.85 vapour quality. CFD simulations are used to predict the instantaneous and averaged values of the liquid film thickness. Numerical results are compared against optical measurements of film thickness. A new method to determine the B parameter in the turbulence damping model based on the Reynolds number of the liquid is proposed.

A Numerical and Experimental Study of Breakdown of Falling Film on Horizontal Circular Tubes

Sateesh Gedupudi, Km Jyoti Singh

Indian Institute of Technology Madras

Email: sateeshg@iitm.ac.in

Falling film horizontal tube heat exchangers are used in several industrial applications such as sea water desalination, multi effect distillation, chemical industries, ocean thermal energy conversion systems and refrigeration. Different flow patterns that occur during falling film evaporation on an array of horizontal tubes are droplet, droplet-columnar, columnar, columnar-sheet and sheet modes. From the heat transfer point of view, a sheet mode (continuous falling film spanning the intertube space) is the most desirable falling-film mode. Falling film breakdown leads to the occurrence of dry patches on tubes which cause heat transfer deterioration. In the present study, falling film breakdown on horizontal tubes is investigated both numerically and experimentally. The study focuses on the influence of tube pitch to diameter ratio on the threshold film Reynolds number required to maintain continuous falling film. Results indicate that the threshold film Reynolds number increases with the increase in the pitch to diameter ratio for a given diameter of the tube. The study also shows that as the number of tubes in the column increases, the threshold film Reynolds number required to maintain continuous film over all the tubes increases. With the increase in surface wettability, the threshold film Reynolds number decreases. The study also compares the threshold film Reynolds number obtained from the present numerical study with the predictions made by different correlations for the column-sheet mode transition available in the literature. The predictions deviate by 0.2 –79 %, depending on the Galileo number and the number of tubes down the column. The reasons for the deviations are discussed.

Influence of Baffle Height on the In-Cylinder Mixing and Performance Characteristics of a Gasoline Direct Injection Engine – a CFD Investigation

Vishal V, Jm Mallikarjuna

Indian Institute of Technology Madras

Email: engr.vishalviswanath@gmail.com

The key focus of engine research is to create fuel-efficient engines that emit fewer emissions. To achieve this, researchers are working on two major approaches: redesigning conventional engines like gasoline direct injection (GDI) or using alternative fuels. In modern GDI engines, in-cylinder flows play a significant role in the performance and emissions of the engine. One of the ways to improve the in-cylinder flows is to use baffles on the cylinder head of an engine. The effectiveness of the baffle depends on its height. Therefore, this study aims to investigate the influence of baffle height on the in-cylinder flows and its impact on the performance and emission characteristics of a four-stroke, four-valve, spray-guided GDI engine using computational fluid dynamics (CFD) analysis. The maximum height of the baffle is decided based on the geometrical consideration of the engine. For the entire analysis, the engine is operated at a constant operating condition of 1000 rev/min under part load condition, with a fuel injection pressure of 80 bar and a compression ratio of 10. From the results, the baffle heights of 2 mm and 3 mm significantly improved the in-cylinder flows compared to that of the base engine. Moreover, it was also found that the indicated mean effective pressure (IMEP) of the engine with 2 mm and 3 mm baffle height improved by about 6% and 10%, respectively, than that of the base engine. In addition, with the increase in baffle height, there is a significant reduction in CO and HC emissions, with a marginal increase in NOX emissions.

Identifying Steady and Pulsatile Two-Phase Flow Regimes with Pressure Drop Signals and Acoustic Emissions

Matthew Hughes, Raj Agarwal, Srinivas Garimella

Massachusetts Institute of Technology
University System of Georgia
Georgia Institute of Technology

Email: mthughes@mit.edu

Identifying two-phase flow regimes is vital for understanding phase-change heat and mass transfer processes. Traditionally, flow regime identification has relied on subjective flow visualization studies, which are then converted to flow regime maps applicable to specific operating conditions. However, these conventional, largely subjective, maps fall short in predicting abrupt changes in flow patterns that often occur during regular operation of vapor compression and absorption heat pumps and other thermal systems. Conventional flow regime maps are also inadequate for addressing the transient, intermittent, or periodic flow regimes that occur in boiling and condensation enhanced using acoustic and other emerging techniques. Therefore, there is a pressing need for alternative flow regime identification techniques that can adapt and reliably track rapid and local changes in two-phase hydrodynamics. Promising candidates for dynamic flow pattern classification include high-resolution pressure drop signals and acoustic emission spectra, which can provide insights into the local hydrodynamics within a flow channel. To assess the feasibility of these measurement techniques, differential pressure and condenser microphone measurements are recorded alongside high-speed videos of a two-phase flow of saturated R134a. The total mass flux and vapor quality are varied to understand the effects of phase velocity and liquid inventory on wave propagation in the channel. Additionally, forced oscillations are introduced to a steady two-phase flow to analyze their impact on flow patterns and their corresponding pressure drop and acoustic signals. Statistical analyses, including Gram-Charlier series, are employed to reveal characteristic pressure drop probability associated with each flow pattern, which form the basis for developing a model capable of predicting flow regimes in both steady and oscillating flows. The resulting framework introduces a new flow regime identification technique that can adapt to dynamic operating conditions, benefiting a wide range of thermal systems and phase-change processes.

A Checkerboard-Free Symmetry-Preserving Conservative Method for Magnetohydrodynamic Flows

Johannes Arend Hopman, Francesc Xavier Trias Miquel, Joaquim Rigola

Universitat Politecnica de Catalunya

Email: jannes.hopman@upc.edu

A checkerboard-free symmetry-preserving conservative method for magnetohydrodynamic (MHD) flows is presented in this work. Simulation of MHD flows at high Hartmann numbers and low magnetic Reynolds numbers are of high interest for the design of a nuclear fusion breeding blanket. Conservative schemes are of special interest in this case, as the delicate balance between the generated Lorentz force and the high pressure drop needs to be preserved. Furthermore, accurate depiction of the motion of flow at any scale is required for the prediction of transitional and turbulent regimes present under these circumstances. Conservative schemes are essential in maintaining these physical qualities, while at the same time warranting unconditional stability. One of the aspects of this method is the use of predictor values for both the Poisson equation for pressure and electric potential. By using a predictor value, the dependence on time step of the order of accuracy of the pressure error shifts from dt to dt^2, greatly reducing the numerical dissipation seen during simulation. This method, however, can lead to increased occurrence of oscillatory solution fields, known as the checkerboard problem. This work introduces a method to quantify and deal with the checkerboarding in both fields during run-time, using a normalised, global, non-dimensional, time-step independent indicator. This value provides a negative feedback on the inclusion of the predictor values, effectively diminishing oscillations by dynamically introducing the necessary numerical dissipation. This method is implemented in the open-source software OpenFOAM and tested using several laminar, transitional and turbulent cases. The results are compared to widely-used methods available in literature, that form the accepted standard to perform such simulations.

On a Proper Tensorial Subgrid Heat Flux Model for LES

Francesc Xavier Trias Miquel, Andrey Gorobets, A. Oliva

Universitat Politecnica de Catalunya
Keldysh Institute of Applied Mathematics

Email: francesc.xavier.trias@upc.edu

In this work, we aim to shed light on the following research question: can we find a subgrid-scale (SGS) heat flux model with good physical and numerical properties, such that we can obtain satisfactory predictions for buoyancy-driven turbulence at high Rayleigh numbers (Ra)? This is motivated by our previous findings showing the reasons for the lack of accuracy of existing SGS heat flux models for LES. Namely, (i) linear eddy-diffusivity models are completely misaligned with the actual SGS heat flux, whereas (ii) non-linear models, such as the gradient model, may lead to numerical instabilities due to the presence of (eigen)directions with negative diffusivity. In this context, we first plan to study a priori the capability of exiting (non)linear models to provide accurate approximations of the actual SGS heat flux both in the bulk and in the near-wall regions. To do so, we will use the data of our previous DNS results of air-filled Rayleigh-Bénard convection at Ra up to 1e11. This analysis will include a new unconditionally stable non-linear model that can indeed be viewed as a stabilized version of the above-mentioned gradient model. In this way, we expect to combine the good a priori accuracy of the gradient model with the stability required in practical numerical simulations. Secondly, we plan to study a posteriori the performance of these models. In this case, LES simulations will be carried out with the same code and results compared with the DNS data. In the first stage, we plan to study the (a posteriori) modelization effects in the bulk region. This can be done by properly refining in the near-wall region. Finally, a posteriori tests including the modelization effects of the near-wall region will be performed.

Numerical Investigation of Air-Flow Distribution Within an Industrial Kiln for Sanitary Ware Manufacture

Eugenio Schillaci, Jesus Ruano, Jiannan Liu, Joaquim Rigola, Carlos David Perez Segarra

Universitat Politecnica de Catalunya

Email: eugenio.schillaci@upc.edu

A detailed knowledge of the distribution of hot air inside kilns for sanitary ware production will allow to optimize manufacturing processes. For this reason, the present work presents a set of CFD simulations to have a clear picture of how the fluid is moving inside the selected kiln. The geometry under study is a simplification of a real industrial kiln. The position of all the inlets (burners, injections of cool air) and outlets (chimneys) has been kept. On the other hand, the shape of the ceramic elements inside the kiln has been simplified in order to make the simulations more affordable. The effect of the solids in the flow has two main contributions: the first one is that a bluff body, such as the ceramic elements within the furnace, distorts the flow. The second one is that the solids heat up and store thermal energy at the hottest zones of the kiln, whereas these elements release it on the coldest parts. This second effect has been taken into account by imposing the temperature profile in the solid surfaces extracted from a previous 1D study. OpenFOAM, an open-source multiphysics software, will be used in the current work. OpenFOAM allows the simulation of the whole kiln considering the two major physics involved: turbulence and buoyancy effects due to temperature gradients. OpenFOAM has dedicated heat transfer solvers that can include turbulence modelling. As a further work, the conclusions extracted from this work will be used as inputs in a second set of more detailed simulations. In those future simulations, the conjugate heat transfer process will be analyzed in detail by studying at the same time the temperature and velocity profiles of the fluid and the temperature distribution inside the different solids.

Simulation of Non-Newtonian Gas-Focused Micro-Jets in Chocked Gas Flow Regime

Rizwan Zahoor, Saša Bajt, Božidar Šarler

University of Ljubljana
DESY
Institute of Metals and Technology, Ljubljana, Slovenia

Email: rizwan.zahoor@fs.uni-lj.si

Stable liquid micro-jets, which carry samples into X-ray beam, are crucial for successful serial femtosecond crystallography (SFX) experiments performed at X-ray free electron lasers. These micro-jets are usually produced by gas dynamic virtual nozzles. In the past, we investigated non-Newtonian liquid jets and made analysis in isothermal conditions with incompressible focusing gas. In the present study, we assess the performance of such nozzles in a chocked flow-focusing gas regime, typical for SFX experiments. An axisymmetric gas dynamic virtual nozzle is considered, where the liquid from an inner feeding capillary is focused by a co-flowing gas from a converging outer capillary. A fixed helium gas flow rate of 15 mg/min and liquid flow rate of 43 µl/min are used in a typical micro-nozzle configuration, resulting in a gas Reynolds number of 320. The Reynolds and Weber numbers for a reference water jet are 90 and 10, respectively. The jet lengths, diameters, velocities, and temperatures are analysed as a function of the power-law non-Newtonian modification of the reference liquid. A related laminar compressible multiphase problem is formulated within the mixture framework and solved with the finite volume method and volume of fluid interface treatment by using the modified OpenFOAM code. It is observed that jets from shear-thinning fluids ($0.9 \leq n < 1.0$) tend to be thicker, longer, and slower when compared with the shear-thickening fluids ($1.0 < n \leq 1.1$). The focusing gas experiences extensive cooling of around ~100-150 K after expansion into the vacuum chamber, whereas the liquid jet's core temperature drops by a few K. This new insight on the behaviour of non-Newtonian gas-focused micro-jets provides a possible new dimension in tailoring the serial crystallography sample delivery systems.

Acceleration of Flow-Focused Liquid Jets in the Presence of a Strong Electric Field

Bor Zupan, Saša Bajt, Henry N. Chapman, Alfonso M. Gañán-Calvo, Božidar Šarler

University of Ljubljana
DESY
Center for Free-Electron Laser Science
Universidad de Sevilla
Institute of Metals and Technology, Ljubljana, Slovenia

Email: bor.zupan@fs.uni-lj.si

Flow-focused micro-jets produced with a gas dynamic virtual nozzle (GDVN) are now commonly used in serial femtosecond crystallography (SFX) experiments at X-ray free electron lasers (XFELs). In these experiments, micron-sized protein crystals are carried inside a liquid jet to the interaction region, where they are exposed to high-intensity X-ray pulses. The jets must be very fast (due to the high repetition rate of X-ray pulsed source), thin (to reduce background in diffraction patterns due to liquid) and long enough so that the X-rays do not “see” the nozzle. To achieve this kind of performance, the nozzle orifice is very small and thus prone to clog, which results in loss of valuable beamtime. We are exploring ways to accelerate carrier fluid using strong electric fields using a novel solution which can be achieved with larger nozzle geometry, thus mitigating the problem of clogging. In this contribution, we present experimental measurements of jet acceleration in the electric field, where nitrogen and 50 % vol mixture of water and ethanol are used as the focusing gas and liquid sample, respectively. The experiments were carried out with the electric potential in the 0 – 7 kV range between a submerged electrode in the sample and an exposed electrode downstream of the nozzle. Measurements were performed via direct jet imaging. The postprocessing was done using purposely developed computer vision software. This investigation established that micro-jets under strong electric fields accelerate with an acceleration of four orders of magnitude larger than the conventional micro-jets.

Experimental Study of the Multiple Impeller Characteristics in the Fermenter Tank.

Andrej Bombač

University of Ljubljana

Email: andrej.bombac@fs.uni-lj.si

Mixing of liquids in a forced mixing tanks is the most commonly used procedure to ensure appropriate levels of homogeneity in mixtures in the process-chemical and pharmaceutical industries. For the purpose of dispersing larger quantities of gas into liquid, tall mixing vessels are used, reactors that are typically equipped with a multi-stage mixer. Such a multi-stage mixer can consist of identical mixers, such as a multi-stage Rushton mixer, or a combination of axial and radial mixers, which have recently become dominant. The choice of mixers significantly influences the basic characteristics of the process, such as mixer power both in liquid mixing and simultaneous gas dispersion in the liquid, the increase in the gas phase fraction, mixing time in liquid mixing, and gas dispersion in the liquid, the occurrence of flood conditions, etc. The selection of a suitable mixer is crucial for the optimal execution of the technological fermentation process for a given geometric configuration of the mixing vessel: the fermentor must ensure an appropriate flow field that supplies organisms with air throughout the entire liquid volume. Any stagnant (dead) zone can lead to the death of cultures or incorrect fermentation products. A multi-stage mixer of the appropriate configuration thus ensures proper substance circulation in the fermentor and, consequently, the most even distribution of the gas phase throughout the volume of the liquid, representing an appropriate hydrodynamic regime. The article presents the characterization of a multi-stage mixer in a pilot fermentor with three different mixer assembly configurations. Measurements of characteristic variables were carried out at various operating regimes using both Newtonian and non-Newtonian fluids (xanthan).

Cavitation Bubble Interaction with Compliant Structures on a Microscale

Jure Zevnik, Žiga Pandur, David Stopar, Matevž Dular

University of Ljubljana

Email: jure.zevnik@fs.uni-lj.si

Numerous studies have already shown that the process of cavitation can be successfully used for water treatment and eradication of bacteria. However, most of the relevant studies are being conducted on a macro scale, so the understanding of the processes at a fundamental level remains poor. In attempt to further elucidate the process of cavitation-assisted water treatment on a scale of a single bubble, the present work numerically addresses interaction between a collapsing microbubble and a nearby structure, that mechanically and structurally resembles a bacterial cell. A fluid-structure interaction methodology is employed, where compressible multiphase flow is considered, and the bacterial cell wall is modeled as a multi-layered shell structure. The contribution of two independent dimensionless geometric parameters is investigated, namely the bubble-cell distance δ and their size ratio ς. The results show that local stresses arising from bubble-induced loads can exceed poration thresholds of cell membranes and liposomes, and that bacterial cell damage could be explained solely by mechanical effects in absence of thermal and chemical ones. Based on this, the damage potential of a single microbubble for bacteria eradication is estimated, showing that larger bubbles carry a higher cleaning capacity. Microstreaming is identified as the primary mechanical mechanism of bacterial cell damage, which in certain cases may be enhanced by the occurrence of shock waves during bubble collapse. The latter is also confirmed through experimental work, which in conjunction with numerical simulations offers further insights into the required threshold hydrodynamic loads for a reliable destruction of E. coli bacteria on the nano- to microsecond time scale. Understanding of the cavitation phenomenon at a fundamental level of a single bubble will enable further optimization of novel water treatment and surface cleaning technologies to provide more efficient and chemical-free processes.

The Kinetic Energy Transfer Analysis Between the Gas and the Liquid in Gas Dynamic Virtual Nozzle Micro Flow-Focusing

Krištof Kovačič, Božidar Šarler

University of Ljubljana
Institute of Metals and Technology, Ljubljana, Slovenia

Email: kristof.kovacic@fs.uni-lj.si

The present study aims to analyse the kinetic energy transfer from the gaseous to the liquid phase in the jetting regime, which appears in micro flow-focusing, produced by a gas dynamic virtual nozzle (GDVN). The liquid phase is focused and accelerated by the gaseous phase, transferring its kinetic energy to the liquid phase. The kinetic energy transfer is investigated by analysing the computational fluid dynamics numerical simulation results based on the Finite Volume Method. The experimentally validated half-space three-dimensional model mesh addresses the unsteady, incompressible, isothermal, Newtonian, turbulent two-phase flow. Along the continuity and momentum conservation equation, the k-ω SST turbulence model is employed to resolve the fluid flow, meanwhile, the Volume of Fluid (VOF) with a geometric reconstruction scheme tracks the gas-liquid interface. The results analyse the efficiency of kinetic energy transfer from the gas to the liquid. It is found that around 50 % is successfully transferred from the focusing gas to the liquid jet before its breakup. The dynamic pressure of the gas and the liquid, which represents the kinetic energy of the fluid, are analysed spatially and temporarily. We present the first detailed study of the kinetic energy transfer from the focusing gas to the liquid jet. Its findings and understanding of the kinetic energy transfer are essential to increase the liquid jet velocity in micro flow-focusing.

Keynote Lecture: Thermodynamic Effects in Cavitating Flow

Martin Petkovšek

University of Ljubljana

Email: martin.petkovsek@fs.uni-lj.si

Cavitation is a phenomenon characterized by vapor generation and condensation in liquid flows, at approximately constant temperature. Cavitation is usually triggered by a local pressure drop in the vicinity of a cavitation nucleus. As the bubble grows, latent heat is supplied from the surrounding liquid to the interface, creating a thermal boundary layer. This creates a thermal boundary layer. The result is a local drop in the liquid temperature, which leads to a drop in vapor pressure. This delays the further development of the bubble, as a greater pressure drop is required to maintain the process. This phenomenon is known as thermal delay or thermodynamic effect and plays a moderating role in the development of cavitation. Thermodynamic effects can usually be neglected for liquids whose critical temperature is much higher than the operating temperature. However, this is not the case for liquids whose operating temperature is close to the critical temperature, such as in cryogenics. The understanding of the thermodynamic effects of cavitating flow is crucial for applications like turbopumps for liquid hydrogen LH_2 and oxygen LO_x in space launcher engines. Experimental studies of this phenomenon are rare as most of them were performed in the 1960s and 1970s. Due to the extreme difficulties of experimental investigation, predicting of thermodynamic effects in cavitation often bases on data in liquids other than cryogenics. Most often used surrogate liquids are hot water or certain refrigerants, which are selected by a single fluid property, most commonly by the thermodynamic parameter $\sum$.

Investigation of Unsteady Skin Friction Effects in Transient Two-Phase Pipe Flow

Anton Bergant

Litostroj Power d.o.o.

Email: anton.bergant@litostrojpower.eu

Liquid-filled pipelines should operate safely over a broad range of operating regimes. Water hammer and transient cavitating flow may induce large pressure fluctuations in pipelines. Transient events in pipeline may cause a drop in pressure large enough to break liquid homogeneity and continuity (liquid column separation, distributed vaporous cavitation). Trapped air pockets in liquid may be a major problem in piping systems. Undesirable transient effects disturb overall operation of hydraulic systems and may even damage the system components. Transient loads can be kept within the prescribed design limits by an adequate control of operational regimes, installation of surge control devices (air valves, control valves, air chambers, surge tanks) or redesign of the original pipeline layout. The first part of the paper deals with mathematical tools for modeling unsteady skin friction, transient vaporous cavitation (liquid column separation) and transient gaseous cavitation (trapped air pockets). The method of characteristics transformation of the unsteady liquid pipe flow equations gives the water hammer solution procedure. A convolution-based unsteady skin friction model using the state-of-the-art numerical tools is explicitly incorporated into the staggered grid of the method of characteristics. Incorporating discrete cavities into the water hammer model leads to the discrete cavity model. An advanced discrete gas cavity model (DGCM) with consideration of unsteady pipe flow friction effects is presented in the paper. The DGCM is capable to simulate vaporous and gaseous cavitation at the same time. Pipeline apparatus for measurement of transient liquid flow (water hammer) and transient two-phase flow (vaporous and gaseous cavitation) is briefly described. The paper concludes with a number of case studies showing how inclusion of unsteady friction into the DGCM significantly affects pressure histories during transient two-phase flow events. Results of numerical simulations and laboratory investigations show profound effects of unsteady skin friction on pressure histories.

Session: Heat Transfer

Chairs: Andrzej J. Nowak, Reijo Karvinen, Sasa Kenjeres, Gian Luca Morini, Anton Bergant, Cao Zhen

Optimising Liquid Impingement Jet Arrays for Concentrated Heat Sources

Jonathan Elliott, Gerard Byrne, Anthony Robinson

University of Dublin, Trinity College

Email: joelliot@tcd.ie

As technology advances, the overall power density requirements of integrated circuit packages are rapidly increasing. However, thermal management technology has not developed at anywhere near the same rate. This has not only created a technological disparity between advanced electronics and associated cooling technologies, but thermal bottlenecks now limit performance and advancement of electronic devices and systems. This work seeks to investigate advanced convective cooling solutions and heat spreading technologies by gaining a deeper understanding of the conjugate heat transfer problem, utilising integrated heat spreading and microjet impingement for maximum thermal hotspot mitigation and source temperature minimisation. The multiphysics conjugate problem was addressed by developing new design methodologies that leverage multi-objective optimisation to inform the thermal-fluids design of the cold plate heat exchanger. This presents the opportunity to tailor the cooling system design to specific applications, which vary greatly within the microelectronics industry. By prioritising the minimisation of multiple parameters such as the hydraulic penalty and device junction temperature, the optimum microjet orifice design could be given for a specific CPU architecture, based on a set range of variable input parameters that influence the geometry of the orifice plate which generates an array of impinging microjets. For this study a single, centralised heat source was utilised in various sizes and investigated practically to verify a computational fluid dynamics tool. This tool was used to run a series of multi-objective optimisation parametric studies to provide the optimum arrangement of microjets for different heat source configurations. The successful implementation of this tool would enable the development of an automated optimisation process with rapid design cycle times that would be capable of keeping pace with the rapidly evolving demands of CPU cooling within the electronics industry.

Two-Phase Cooling System for Electric Vehicles' Battery

Luca Cattani, Matteo Malavasi, Alessandro Benelli, Fabio Bozzoli

University of Parma

Email: luca.cattani1@unipr.it

Our era faces a paramount challenge: transitioning to sustainable transportation. Electric propulsion in vehicles plays a pivotal role in this shift. Currently, Lithium-Ion cells dominate this technology, boasting significant electrochemical optimization. However, a persistent issue is effective thermal management of these batteries. The power supply unit often demands rapid, high-power delivery, causing the batteries to generate substantial heat. Elevated temperatures can compromise battery performance, accelerating degradation or causing malfunctions. In response, a Battery Thermal Management System (BTMS) is crucial to ensure optimal battery efficiency and extend their lifespan. Our study introduces an innovative BTMS that leverages a combination of two-phase direct liquid cooling and Pulsating Heat Pipes (PHPs). This system offers an attractive solution, blending high thermal efficiency, passive operation, and cost-effectiveness. Within this setup, batteries are immersed in a low-boiling dielectric fluid enclosed within a Plexiglas container to facilitate efficient heat exchange. The dielectric fluid operates in equilibrium between its liquid and vapor phases. The evaporator section of the PHP is placed within the vapor phase region, while the condenser is positioned outside the Plexiglas container and cooled by convection from an airflow. An essential challenge arises when the fluid begins to evaporate preventing a hazardous dry-out condition due to the closed enclosure. This situation could result in a sharp rise in temperature and pressure, leading to equipment malfunction and safety hazards. To address this challenge, the PHP was sized and engineered to effectively mitigate heat spikes, maintaining safe operational temperatures by promoting vapor recondensation. The proposed BTMS has demonstrated remarkable efficiency, ensuring that battery temperatures remain within the recommended range, even under high load conditions. An advantage of this cooling system is its complete passivity, eliminating the need for energy-consuming coolant circulation. This enhances the system compactness and elevates the reliability of the power unit cooling system.

A Phase-Splitting Approach to Describe Macroscopically Non-Equilibrium Transport in Porous Media

Michel Quintard, Yohan Davit, Brian D. Wood

Toulouse INP
Institut polytechnique de Grenoble
Oregon State University

Email: michel.quintard@toulouse-inp.fr

Classical macro-scale dispersion equation cannot correctly represent non-local, non-equilibrium effects, and alternatives had to be proposed to overcome these discrepancies. Two-equation models have been widely used and proved useful to incorporate, to some extent, local non-equilibrium effects. More accurate descriptions may be obtained through the introduction of additional complex terms in the one-equation dispersion equation, possibly terms involving spatial and time convolutions. An alternative more practical route rely on the introduction of a N-equations description. For instance, instead of a two-temperature model, transfers in the solid and/or fluid phase maybe represented by several equations, each one defined by some splitting framework. Splitting of the solid phase has been proposed in the past (multi-rate models). Splitting of the flowing phase has a rustic illustration in the case of the Coats and Smith model, i.e., "flowing" and "stagnant" fluid equations. In this paper, we explore further the possibilities offered by N-equations splitting of the flowing phase. The N-equations macro-scale model is developed applying an averaging technique to N-pseudo phases obtained by splitting the flowing pĥase based on different criteria, e.g. the velocity histogram. Most important properties in the resulting macro-scale model are the matrix of dispersion tensors, and the matrix of exchange coefficients, which are provided explicitly by a system of specific "closure" problems. To illustrate the potential of this approach, the methodology is applied to Taylor's dispersion problem, in which non-local effects are produced by: (i) pore-scale spatially distributed inlet conditions, (ii) pore-scale spatially distributed initial conditions. The proposed N-equations description was compared to estimates of the N-temperatures obtained from direct numerical simulations, with initial and/or boundary conditions of type (i) or (ii). The N-equations model was able to reproduce accurately the dynamic of pore-scale computations, for such reputedly non-homogenizable problems, with a relatively small number of equations.

Numerical Simulation of Droplet Impact onto Heated Surfaces Below the Boiling Point

Rishav Saha, Bernhard Weigand

Universitat Stuttgart

Email: rishav.saha@itlr.uni-stuttgart.de

Droplet impact on heated surfaces is a widespread phenomenon in industrial applications, particularly in the context of spray cooling processes. Therefore, it is very important to study the complex phenomenon of droplet splashing, surface wetting, and flow features on heated surfaces. The primary focus of this research centers on the wetting dynamics at surface temperatures below the boiling point of the liquid. The effects of varying temperatures on droplet spreading and heat removal rates are discussed in detail. After careful consideration of the numerous benefits associated with simulations, this study employs a computational fluid dynamics framework to simulate the impact of a liquid droplet on a heated surface, using the Direct Numerical Simulation approach. The simulation tool used is the DNS code Free Surface 3D, an in-house code at the Institute of Aerospace Thermodynamics, which has been used for more than 20 years and is constantly being developed further. The Finite-Volume method is utilized, and the interface is defined by the Volume-Of-Fluid (VOF) method. As industries continue to push the boundaries of thermal performance, the findings from this research contribute to a broader knowledge of cooling strategies based on droplet-surface interactions at temperatures below boiling.

Design and Numerical Simulation of a 45 kWel Multi-Source High-Flux Solar Simulator as a Heating System of Dielectric Materials Placed in a High-Frequency Characterization Cavity

Jesse Allens Touoyem Talla, Baptiste Henriot, Thierry Duvaut, Olivier Tantot, Nicolas Delhote, Michaël Charles, Jaona Harifidy Randrianalisoa

Universite de Reims Champagne-Ardenne
Universite de Limoges
Commissariat à l'Energie Atomique (CEA Le Ripault)

Email: jesse-allens.touoyem-talla@univ-reims.fr

Dielectric properties such as permittivity and tangent loss of ceramic materials play a crucial role in numerous fields including optics, electronics, aerospace, and telecommunications. While these properties are frequently investigated at ambient and moderate temperatures, scarce studies have been reported at higher temperatures and especially above 1000°C. This communication reports the development of a heating system based on high-flux solar simulator (HFSS) and designed for the characterization of high frequency dielectric properties of ceramic materials with a resonant cavity device. This HFSS system comprises seven xenon arc lamps, each coupled to an ellipsoidal reflector. Each lamp delivers a maximal electrical power of 6.5 kWel and the ellipsoidal reflectors have a reflectivity of 96%. Lamps positions and orientations were chosen on a virtual sphere of about 1600 mm radius to provide concentrated irradiation distributed over the entire front face of the sample, positioned in the center of the virtual sphere, without exceeding maximum rim angle. The sample was placed inside a resonant cavity, which works as the high-frequency dielectric characterization system. The cavity was equipped with a circular opening to allow the entry of the irradiation flux. The ray propagation from the HFSS to the sample located in the cavity is modelled with a Monte Carlo ray-tracing simulation. The thermal behaviour of the sample and its surrounding (air and cavity) was studied through numerical simulations of coupled flow and multi-mode heat transfer, implemented in ANSYS Fluent. It was found that a maximum flux of 1.84 MW m^{-2} on the irradiated face of the sample was achieved. The maximum temperature of the sample was around 1470°C with a non-uniformity of 10%. The simulation results show that the designed heating system is suitable for the intended purpose and allows to characterize thermophysical and infrared properties of dielectric materials at high temperatures.

A Numerical Investigation on a Liquid-Cooled Battery Thermal Management System: Effect of Inlet Temperature and Flow Rate

Soner Birinci, Melisa Albayrak, M. Yusuf Yazıcı, Buğra Sarper, Orhan Aydın

Karadeniz Teknik Universitesi

Email: sonerbirinci@ktu.edu.tr

An effective battery thermal management system (BTMS) is needed for the safe and efficient operation of electrical vehicles. A BTMS with an innovative cold plate design is proposed in this paper for cooling of a cylindrical Li-ion battery pack arranged in 2×18 layout, and the effects of inlet temperature of the cooling fluid is numerically investigated for 2C discharge rate for various flow rates from 0.1 to 1.0 l/min. The highest temperature difference inside of the battery pack, the peak temperature, pressure drop throughout the cold plate is analyzed via ANSYS Fluent software.

The Influence of Cold Plate Height on Thermal Management of a Liquid-Cooled Battery Pack

Soner Birinci, Melisa Albayrak, Mehmet Saglam, Buğra Sarper, Orhan Aydın

Karadeniz Teknik Universitesi

Email: sonerbirinci@ktu.edu.tr

Li-ion batteries require an effective battery thermal management system for electric vehicles to operate more effectively and safely. The cooling performance of a Li-ion battery pack configured in a 2×18 arrangement and cooled by liquid-cooled active BTMS is numerically examined in this work. A new cold plate design is presented for this purpose, and the thermal performance of the system is evaluated under various operating scenarios. Under varying operating conditions, such as five-volume flow rates (0.1-1.0 l/min) and different discharge rates (0.5C, 1C, and 2C), four distinct cold plate heights (30-40-50-60 mm) are investigated. In numerical simulations, ANSYS Fluent software is utilized, and the greatest temperature difference, energy density of the battery pack, peak temperature, pressure drop, and cooling efficiency are all determined for various configurations.

An Advanced Numerical Radial Sub-Resolution Technique to Correct Heat Fluxes in the Vicinity of Bubbles at Saturation

Mathis Grosso, Guillaume Bois, Adrien Toutant

Commissariat a l'energie atomique et aux energies alternatives
Centre National de la Recherche Scientifique

Email: mathis.grosso@gmail.com

Simulating two-phase flows presents a challenge in accurately capturing temperature variations near sharp interfaces. Many interface tracking methods tend to spread the interface across grid cells, making it exceedingly difficult to impose interfacial temperature conditions with precision. To address this issue, the authors have developed a physically motivated methodology called the Laminar Radial Sub-resolution (LRS). It benefits from a clear scale separation and noticeable uni-directional normal temperature variations in the interface vicinity. The Laminar Radial Sub-resolution takes advantage of the Front-Tracking method. In a frame of reference attached to each interface portion, a radial probe is used to solve for a steady spherical advection-diffusion temperature equation. The boundary condition at the probe's tip is deduced from the Eulerian temperature field. Velocity components are evaluated radially while tangential effects may be incorporated as source terms. Initially tested a priori on fully resolved simulations, this method is now being evaluated in simulations by locally substituting the Eulerian fluxes computed using conventional numerical schemes to continuously correct the surrounding field evolution. A first test case in pure diffusion is performed. Then a single bubble rising in quiescent sub-cooled liquid is simulated at moderate Peclet and liquid Prandtl numbers ($Pe=Re_bPr<400$, $Pr_l<5$). Spatial resolution is constrained deliberately to observe the effect of the sub-resolved region on the Eulerian solution and establish the best compromise between accuracy and computational cost. Results are compared with fully resolved simulations employing a temperature extrapolation technique known as the Ghost Fluid Method to treat the interfacial temperature gradient discontinuity. The LRS coupling to the resolved field has proved efficient in correctly capturing the interfacial heat transfer and at a very reduced computational cost. The method opens up exciting new opportunities for the precise investigation of complex multi-bubble configurations.

Numerical Simulations of Bubbly Turbulent Convection in Cubical Geometries

Joauma Marichal, Pierre Ruyer, Yann Bartosiewicz

Universite catholique de Louvain
Gouvernment Francais
Université Catholique de Louvain

Email: joauma.marichal@uclouvain.be

In this work we present numerical results of pool boiling flow in a turbulent Rayleigh-Bénard convection configuration, using our in-house code in a cubical geometry. The problem in hand is encountered in various natural phenomena as well as in industrial applications. A Eulerian-Lagrangian approach is developed for the mixture of liquid water and vapor bubbles. The liquid mean temperature is close to the saturation temperature and is governed by the low-Mach number Navier-Stokes equations that are solved using DNS standards. The motion and growth/shrinkage of each individual vapor bubble is modeled and the backward effect of the bubbles on the fluid is accounted via momentum and energy exchanges between the two phases (two-way coupling), as well as variations in fluid phases volumetric fractions (volumetric coupling). In such pool configuration the non-dimensional parameters governing the flow are both those relative to Rayleigh-Bénard convection, namely Rayleigh number, Prandtl number and aspect ratio, and those to boiling, namely the overall vapor volumetric fraction, the bubble size based Reynolds number, the degree of superheat of the liquid and the Jakob number (sensible heat to latent heat ratio). At first, we describe the model used and its corresponding validation, involving natural convection, isolated bubble dynamics and coupling between bubbly and bulk flows. In the second part, we consider the study of the relationship between heat transfer through the pool (Nusselt number) and the flow topology for different settings of the bubbly configurations.

A Coupled Point Particle Two-Phase Heat and Mass Transfer Model for Dispersed Flows Based on Boundary Element Methods

Matjaž Hriberšek, Timi Gomboc, Matej Zadravec, Jure Ravnik

University of Maribor

Email: matjaz.hribersek@um.si

In drying processes we consider complex physical processes, which involve coupled multiphase heat, mass and momentum transfer. In porous particle drying, the drying process has to be separated into several stages, because, besides surface moisture, particles include moisture in their interior. In this contribution a two-way coupling model is presented based on the use of the elliptic fundamental solution and the Dirac delta function properties to accurately evaluate the heat and mass point particle source impact on the continuous (air) phase, solved by the Boundary Domain Integral Method (BDIM). In addition to the BDIM model of the particle-fluid heat and mass transfer interaction, the two-phase flow case under consideration is extended to the case of porous spherical particle drying with internal moving drying front, which is solved by the Boundary Element Method. The applicability of the developed method is highlighted on the case of a simplified spray dryer, where a comparison between the obtained drying times for the cases of the one-way and the two-way heat and mass transfer coupling results shows, that the developed numerical model accurately captures the effect of moisture accumulation and temperature decrease in the fluid phase, leading to realistic computations of drying rates of porous particles in the flow.

Estimation of Thermophysical Properties of a Pouch-Type Li-Ion Battery Using an Inverse Methodology

J Jithu, Kasavajhula Naga Vasista, Suraj Kumar, Srinivasan, Balaji, C Balaji

Indian Institute of Technology Madras
Indian Institute of Technology, Delhi

Email: me22d001@smail.iitm.ac.in

Electric vehicles are experiencing a surge in popularity, primarily driven by the imperative to combat pollution. Batteries play a pivotal role in the operation of electric vehicles. Nevertheless, a significant challenge faced by electric vehicles, particularly in tropical regions, is the tendency for battery temperatures to exceed permissible limits. Consequently, effective battery thermal management assumes paramount importance in enhancing the overall performance, reliability, and safety of electric vehicles. To achieve this, first a comprehensive understanding of the thermophysical properties of batteries becomes imperative to proceed with any meaningful modelling. The present study concerns the estimation of the temperature-dependent orthotropic thermo-physical properties (kxx,kyy,kzz,cp) of the active material in a pouch type Li-ion battery using an inverse methodology. An experimental study is conducted on a commercial AMP20M1HD-A0 Li ion battery to measure the surface temperature at various locations using thermocouples. The forward model simulates the three-dimensional unsteady state conduction problem of the set up with the same experimental boundary conditions using a finite element based commercial software COMSOL Multiphysics. The surface temperatures of the battery are simulated using the forward model for various values of kxx,kyy,kzz, and cp for the given heat flux. An Artificial Neural Network (ANN) is developed based on the input-output data obtained from the forward model, and acts as a replacement for the forward model. In the inverse model, Bayesian statistics, along with Metropolis Hasting-Markov Chain Monte Carlo (MH-MCMC) algorithm is used for analyzing the posterior distribution to estimate the mean, the maximum a posteriori and the standard deviation of the thermal properties (kxx,kyy,kzz, and cp). The inverse methodology is validated by solving the forward model to determine the surface temperature using these estimated properties and comparing it with the measured surface temperatures.

Approximate Analytical Solution for Solidification of PCM in Cylindrical Geometry with Temperature Dependent Thermal Conductivity-Perturbation Method

Milad Tajik Jamalabad, Cristobal Cortes

University of Zaragoza

Email: milad.t.jamalabad@gmail.com

The Solidification process of Phase change materials inside cylindrical enclosures is analyzed and analytical solutions are derived to determine the positions of the interfaces at different time intervals. This enables the prediction of transient interface locations and the overall duration required for complete phase change during solidification. Initially, analytical solutions are compared with those already present in the literature, specifically in cases where the phase change material (PCM) maintains a constant thermal conductivity. The anticipated results exhibit a strong agreement with the findings previously documented in the literature. In this current investigation, we address the scenario involving temperature-dependent thermal conductivity. For the first time, we employ perturbation techniques to analytically derive the dimensional temperature for both phases. Our study specifically incorporates a linear model to account for the variation in thermal conductivity with temperature. The results show that the duration of complete solidification relies on the Stefan number. As these parameter is augmented, the total solidification time of the cylinder diminishes and escalates, respectively. Furthermore, it has been observed that an augmentation in the thermal conductivity of the phase change material (PCM) leads to a swifter solidification process.

Sustainable Heating and Cooling for Residential Buildings: Coaxial Ground Heat Exchangers

Fawad Ahmed, Nicola Massarotti, Božidar Šarler

Universita degli Studi di Napoli Parthenope
University of Ljubljana
Institute of Metals and Technology, Ljubljana, Slovenia

Email: fawad.ahmed001@studenti.uniparthenope.it

As the world fights climate change challenges and the soaring demand for energy-efficient solutions, the need for sustainable heating and cooling systems in residential buildings has become paramount. Traditional systems rely on fossil fuels and contribute significantly to greenhouse gas (GHG) emissions. Ground source heat pumps (GSHPs) have emerged as a promising technology, offering a sustainable alternative to conventional space conditioning systems. Central to the success of GSHPs is the efficient and reliable operation of their ground heat exchangers, with lower costs needed for installation, as prospective solution for future residential applications. This paper investigates the thermal performance of coaxial ground heat exchangers (CGHE) that incorporate phase change material (PCM) in the ground for GSHPs applications. The analysis of the CGHE with PCM grout is presented, highlighting the advantages in terms of heat exchange performance over normal grouts for GHE. Additionally, considerations for system sizing, maintenance, and compatibility with different heating and cooling strategies are discussed. In conclusion, this paper underscores the significance of CGHE as a sustainable heating and cooling solution for residential buildings. It offers valuable insights for homeowners, builders, and policymakers seeking to reduce energy consumption, lower carbon footprints, and promote a greener future for residential heating and cooling systems.

Experimental Investigation of the Supercritical Heat Transfer of the Low GWP Refrigerant R1234ze(E): Calibration and Validation of Test Rig

Jera Van Nieuwenhuyse, Steven Lecompte, Michel De Paepe

Universiteit Gent

Email: jera.vannieuwenhuyse@ugent.be

Heat transfer of supercritical refrigerants is applied in practical systems such as the transcritical organic Rankine cycle or supercritical heat pump, which are thermodynamic cycles suited for the conversion of low-grade heat into electricity or high-grade heat. The concept of supercritical heat transfer is not new and has already been studied and applied for decades in several applications, including supercritical steam generators. However, results on one fluid cannot be directly translated to another one. This is because the thermohydraulic behavior of fluids is heavily influenced in the near-supercritical region due to sudden variations in the thermophysical properties when the fluid is heated or cooled close to its critical point. Secondary flow phenomena, including buoyancy and flow acceleration, can be induced, and the heat transfer can no longer be accurately predicted by constant property correlations such as the Dittus-Boelter or Gnielinski correlation. For refrigerants under horizontal flow, the research is limited. Most studies in literature are on small diameter tubes and on R134a, which has a high GWP. Therefore, in this work, an extensive measurement campaign on several low GWP refrigerants over a large temperature and pressure range in the supercritical region is performed. First, the experimental test rig used for the experiments is discussed. The test section consists of a 4 m long horizontal tube with an inner diameter of 22.9 mm, equipped with numerous thermocouples at both the tube wall and in the bulk of the flow, and is heated through Joule heating of the stainless steel tube. Next, the validation of the setup and data reduction is highlighted. Finally, the influence of several parameters (including mass flux, heat flux and pressure) on the supercritical heat transfer to several low GWP refrigerants is discussed.

Modelling of the Conduction-Radiation Multilayer Flash Method by a Fully Stochastic Approach

Loïc Seyer, Franck Enguehard, Denis Rochais

Commissariat a l'energie atomique et aux energies alternatives
Universite de Poitiers

Email: seyer.loic@gmail.com

In the last decades, heterogeneous materials presenting 3D complex morphologies such as porous, cellular or fibrous materials have drawn much attention due to their excellent thermal properties, especially at high temperatures. On the other hand, the characterization of their thermal behavior through numerical methods remains a big challenge due to the complex material morphologies involved. Especially, the classical deterministic methods exhibit limits regarding their high memory need when studying transient coupled conduction-radiation through heterogeneous 3D complex structures. Thus, new so-called stochastic methods are being developed, with the advantage of a very low memory need, allowing the study of the transient thermal behavior of more representative numerical structures, at the cost of computing times equivalent to those obtained by deterministic methods when the asymptotic steady state is wanted. In this study, a fully stochastic method is presented for the resolution of the transient coupled conduction-radiation heat transfer within heterogeneous media. Here, this method is used for the simulation of a classical flash experiment applied to 1D multilayer samples made of opaque and semi-transparent materials. The semi-transparent media are grey, emitting, absorbing and scattering, and the model fully accounts for the non-linear nature of the physical problem treated. A ray-tracing method is used for the modelling of radiation, and Brownian walkers are used for the simulation of transient conduction. First thermograms are presented on multilayer samples made of classical Beerian materials but also of fictitious non-Beerian media.

Improving Virtual Heat Transfer Description of Developed and Undeveloped Annular Gap Flows at High Taylor Numbers

Anton Žnidarčič, Tomaž Katrašnik

Faculty of Mechanical Engineering, University of Ljubljana

Email: anton.znidarcic@fs.uni-lj.si

Increasing power densities of electric machines are leading to increased rotor velocities and with them to higher flow complexity in their annular gap. At the same time, axial flow through the annular gap is utilized to improve heat transfer abilities, which are crucial for ensuring safe and reliable operation of high power density electric machines. These flow conditions of high rotating and axial velocities, characterized with high Taylor and Reynolds numbers, are influenced by the operating conditions, annular gap geometry and the conditions at its inlet, which can importantly shape the entrance region appearace. The impacts on convective heat are in such conditions due to the complex interactions between mentioned impacts still not resolved. And while the majority of research work is experimental, the numerical studies focus mainly on developed flow conditions due to the high computational costs required to study presence of undeveloped flow at high Taylor and Reynolds numbers. To make a step forward and provide feasible tools for valid definition of convective heat transfer in these conditions, useful for a wide range of end-users, the work considers performing 3D CFD simulations of flows at high Reynolds and Taylor numbers. This relies on a holistic approach to description of the flow conditions, which considers also the conditions outside of the annular gap through the study of the impact of various flow introduction into the electric machine. The question of suitable turbulence modelling to adequately capture various flow phenomena that affects heat transfer is also addressed through comparison of results, obtained with RANS and LES models. The effects of developed and undeveloped flow on heat transfer are specifically considered. The modelling requirements neccessary to obtain valid results are discussed. Finally, the most suitable, currently available modelling approach for the considered flow conditions is stated.

Numerical Modeling of a Confined Falling Liquid Film Sheared by a Gas Flow in a Plate Heat Exchanger of an NH3/H2O Absorption Chiller

Jana Sleiman, Benoit Stutz, Hai Trieu Phan

Commissariat a l'energie atomique et aux energies alternatives (CEA)
Université de Savoie

Email: jana.sleiman@cea.fr

In recent years, according to the IEA, the demand for cooling has seen a substantial increase, primarily due to rising summertime temperatures. Conventional mechanical compression air conditioners have played a predominant role in fulfilling this demand, albeit at the cost of significant electricity consumption. In response, absorption chillers have emerged as an eco-friendly alternative, powered by sustainable heat sources like solar energy or industrial waste heat. The present work is conducted within the framework of developing compact absorption chillers incorporating plate heat exchangers. It builds upon prior research by improving an existing numerical model for falling-film plate heat exchangers within an NH_3/H_2O absorption chiller. This former numerical model evaluates the coupled heat and mass transfers between the liquid film and the vapor flow while excluding hydrodynamic interactions at the liquid-vapor interface. The current investigation aims to address compactness and cost-efficiency limitations in absorption chillers. The consequences of operation within confined spaces were numerically evaluated. An innovative analytical model was developed to quantify the influence of liquid-vapor flow interactions. Shear stress and interfacial perturbations were considered compared to the classical Nusselt model, enabling the evaluation of the changes in average film thickness and flooding phenomenon. This analytical model has then been added to the previously developed heat and mass transfer model. The numerical model was applied to both the desorber and absorber units. The results show that increased confinement has minimal effects on the transfer coefficients. Additionally, an approach is suggested for estimating flooding in confined spaces and the results are compared with various correlations available in the literature.

Simplified Computational Model of the Primary and Secondary Freeze-Drying Process of Agriculture and Marine Foods

Andrzej J. Nowak, Edyta Piechnik, Michal Stebel, Michal Haida, Bartlomiej Melka, Agnieszka Ciesielska

Silesian University of Technology, Poland

Email: andrzej.j.nowak@polsl.pl

The proposed paper considers an analysis of the primary and secondary freeze-drying process taking place on one shelf of the commercial freeze-dryer. The freeze-drying itself is a low-temperature dehydration process consisting of the following basic steps: (1) freezing of the product in the chamber, (2) vacuuming of the freezing chamber, (3) ice sublimation from a frozen product (primary drying), (4) removing the water bound in the product (secondary drying), and (5) defrosting. Proper selection of the thermal parameters in the freeze-drying process and minimising energy consumption requires a computational model of the process under consideration. The worked-out computational model of freeze-drying is a time-marching model which consists of equations which govern separately primary and secondary drying. For each stage of the drying process, the model contains coupled heat and mass transfer equations supplemented with equations describing the equilibrium either on the ice front in the primary drying or the equilibrium on the top surface of the dried layer in the case of the secondary drying. In the model, it is also assumed that the heat transfer process is quasi-steady within the one time step while the mass transfer process remains unsteady within this time step and results either in the movement of the ice front or in the water content change within the product. These all equations describing what is happening within the product have to be coupled with CFD equations defining the fluid flow and heat transfer within the lyophilisation chamber. This model was also partially validated using the measured temperature at the core of the food samples located in various places inside the storage chamber. The experimental campaign has been performed for various food products including meat, fruit and vegetables.

Thermal Analysis of an Integrated Motor Drive with a Switching Cell Array Power Converter

Mattia Grespan, Elisabet Mas De Les Valls, Sergio Busquets Monge, Xavier Jordà, Diego Angeli

University of Modena and Reggio Emilia
Polytechnic University of Catalonia
Institute of Microelectronics of Barcelona

Email: 215689@studenti.unimore.it

In modern electric propulsion systems the electric motor is tightly integrated with the power converter and control units. These so called Integrated Motor Drives (IMDs) can allow for vast improvements in efficiency and volume reduction, due to the elimination of redundant electric components and the shared case and cooling system between the motor and the power converter. The cooling systems used in these applications must be carefully designed as the motor and power electronic components feature greatly different temperature limits. This work discusses the development of a thermal model for an IMD composed by a six phase permanent magnet motor with hairpin windings and a switching cell array based power converter. The overall system is modelled by means of a lumped parameter thermal network which includes an accurate model of the motor stator and rotor, as well as a detailed arrangement of the power switches on the converter boards. The conductive resistances are simply obtained from geometric data and material properties. Instead, the convective resistances are initially evaluated by means of empirical models available in the literature, then the results are refined by developing dedicated CFD models. The developed model is employed to asses the feasibility of multiple arrangements of the electric motor and power electronics components. In addition, a sensitivity analysis is carried out to identify the most relevant parameters in cooling systems for IMDs.

DNS of Marangoni Effects on a Suspension of Droplets in Microgravity Using the Unstructured Conservative Level-Set Method

Nestor Vinicio Balcazar Arciniega, Joaquim Rigola, A. Oliva

Universitat Politecnica de Catalunya

Email: nestorbalcazar@yahoo.es

Direct Numerical Simulation of marangoni effectes (thermocapillarity) of a bi-dispersed suspension of droplets is performed using multiple markers unstructured conservative level-set method for two-phase flow with variable surface tension. Surface tension is a function of temperature on the interface. Consequently, the called Marangoni stresses induced by temperature gradients on the interface lead to a coupling of the momentum transport equation with the thermal energy transport equation. The finite-volume method on 3D collocated unstructured meshes is used to discretize the transport equations. The unstructured conservative level-set method is employed for interface capturing, whereas the multiple marker approach avoids the numerical coalescence of droplets. The pressure-velocity coupling is solved by the fractional-step projection method. Unstructured flux limiters approximate the convective term of transport equations. Adaptive mesh refinement is introduced for the optimization of computational resources. Verifications, validations and numerical findings are reported.

A 3D Symmetry-Preserving Simulation of a Concentrated PhotoVoltaic Thermal (CPVT) Solar Collector

Daniel Santos Serrano, Joaquim Rigola, Jesus Castro, Francesc Xavier Trias Miquel

Universitat Politecnica de Catalunya

Email: daniel.santos.serrano@upc.edu

In this work, a general collocated and unconditionally stable framework on unstructured meshes for solving Conjugate Heat Transfer (CHT) problems is presented by means of preserving the underlying symmetries of the continuous differential operators, thus not introducing uncontrolled artificial numerical dissipation to our system. Then, this framework is applied to solve a Concentrated PhotoVoltaic Thermal (CPVT) Solar Collector, including the Radiative Transfer Equations (RTE).

Thermodynamic Analysis of Two-Fluid, Two-Phase Immersion Cooling System for Cooling of Electronic Components

Balaji C, V.B. Krishnadasan, Pratheek Suresh

Indian Institute of Technology Madras

Email: balaji@iitm.ac.in

Two-phase immersion cooling system is a specialized and increasingly popular method for cooling high-power density electronic components, such as CPUs, GPUs, and other server equipment. In a typical two-phase immersion cooling system, a dielectric fluid is employed for immersing electronic components. The challenge lies in the fact that the fluid chosen often has limited thermal conductivity, hindering effective heat transfer. In a multi-fluid immersion cooling system, electronic components are submerged in a dielectric fluid known as the boiling fluid. Within the same tank, a separate condensing fluid is introduced, which is immiscible with the boiling fluid. The condensing fluid, characterized by its lower specific gravity and higher thermal conductivity compared to the boiling fluid, enhances heat transfer within the system. In the current study, NOVEC 7100 serves as the boiling fluid, while deionized water is employed as the condensing fluid. Exergy analysis is a valuable tool for identifying areas of inefficiency within a process, offering opportunities for improvement. This is possible because exergy analysis takes into account the temperature of heat transfer during various stages, including those occurring in the tank and related processes. In this work, exergy analysis is conducted to assess and compare the efficiency of a multi-fluid, two-phase immersion cooling system in contrast to a single-fluid, two-phase immersion cooling system. Additionally, a comparison is carried out to evaluate how the system's performance varies under diverse operating conditions. The study was conducted for a maximum heat flux of 12.3 W/cm^2 and performance of the system was evaluated at condenser inlet temperatures spanning from 5 to 250 C. The study findings indicate that the exergy efficiency of the multi-fluid system surpasses that of the single-fluid system.

Thermal Management of a Complex System on a Chip Using Multiple PCM Based Heat Sink

Balaji C, Srinivas Ramineni, Kasavajhula Naga Vasista

Indian Institute of Technology Madras

Email: balaji@iitm.ac.in

In pursuit of high computational performance and an efficient data transfer on a miniature electronic equipment, the microelectronics industry has pioneered the development of an advanced 2.5D packaging technology known as the System-on-a-Chip (SoC). This includes several heat generating chiplets like CPU, GPU, DRAM and others on a single SoC. In view of different thermal dissipation rates pertaining to each chiplet, thermal stresses and hotspots are inevitable. In the present study we propose an air-cooled heat sink with multiple phase change material (PCM) for thermal management of SoC. This study investigates the effect of organic PCM combinations (n-Eicosane, Docosane and Tetracosane) in the heat sink chambers, fin dimensions, number of fins, heat transfer coefficient, surge duration and surge interval on thermal performance of heat sink during power surges. The nominal and power surge heat loads for a SoC are considered as 30 and 60 W, respectively. The governing equations along with their respective boundary conditions of the present experimentally validated numerical model are solved using ANSYS Fluent 2022R2. The preliminary results show over a 300% increase in the time taken for SoC to reach steady state temperature for a heatsink with PCM compared to that without PCM for a constant nominal heat load. Further, a 4.5 0C reduction in the maximum temperature of SoC is obtained by using PCM during power surge compared to that without PCM. The present study provides physical insights on utilizing multiple PCMs in reducing thermal non-uniformity of a SoC to improve its longevity and reliability.

Thermal Performance Against Gravity of an AlSi10 AM Heat Pipe with a Diamond Lattice Structure Wick

Luigi Vitali, Simone Menini, Manfredo Guilizzoni, Alfonso Niro

Politecnico di Milano

Email: luigi.vitali@polimi.it

Metal Additive Manufacturing has gained momentum as a viable production technique for specialized heat transfer devices, in particular for industrial sectors characterized by small production numbers associated with high-performance requirements. Within the space industry, ESA is leading and funding several applied research programs to explore the possibility to employ AM for building electronic boxes with embedded heat pipes, in order to reduce manufacturing post-processing steps and contact thermal resistances. In the context of tender 1-10238, the project “Heat Pipe Solutions for High Power Systems” (HPS2) has developed and tested lattice-based heat pipes, that are intended to be integrated in electronic modules. In this paper, the heat transfer performances as function of input power and tilt angle against gravity of a 150 mm long heat pipe with a 20 mm evaporator section and a 40 mm condenser section are presented and compared with the results of the models of performance limits, based on the measured properties of the lattice.

Numerical Analysis of Transport Phenomena in a Steam Reforming Reactor with Optimal Multi-Segments Catalyst Distribution

Janusz Szmyd, Marcin Pajak, Grzegorz Brus, Shinji Kimijima

AGH University of Science and Technology
AGH University of Krakow, Poland
Akademia Gorniczo-Hutnicza im Stanislawa Staszica w Krakowie
Gakko Hojin Shibaura Kogyo Daigaku

Email: janusz.szmyd@agh.edu.pl

The most crucial issue with current technology is the emission of greenhouse gases and their negative impact on climate. One of the possible approaches to limit the issue of emissions is the steam reforming of natural gas, leading to the production of hydrogen. Fuel cells are a robust technology, able to conduct a catalytic conversion of hydrogen and oxygen, for the direct production of electrical energy. Fuel cells are one of the most environment-friendly technologies to this day, as their exhaust gases mostly consist of steam. Currently, almost 50% of the hydrogen produced is acquired via hydrocarbons reforming. The process described in the presented analysis occurs between methane and steam. The presented numerical analysis regards small-scale reactors, which are more suitable when it comes to the processing of distributed or stranded resources for hydrogen production. To optimize the small-scale unit’s performance, the macro-patterning strategy is introduced. Steam reforming has a strong endothermic character and tends to produce unfavourable thermal conditions. The process enhancement is acquired by introducing non-catalytic regions to the catalytic insert geometry. The non-catalytic segments are introduced to suppress the reaction locally, decreasing the magnitude of temperature gradients. Unification of the temperature distribution is proven to increase the reforming’s effectiveness. The presented analysis introduces a new approach to the catalytic insert division, to investigate if a complete temperature field unification is possible. The catalytic insert is simultaneously divided along the reactor’s radius and length, resulting in a set of concentric rings, placed along the reactor’s axis. The calculations are conducted using in-house numerical procedure, coupled with a genetic algorithm. The algorithm optimizes the process effectiveness by modification of the segment's alignment and porosity.

Temperature Forecasting for Single-Phase Immersion Cooling System Based on Machine Learning

Pratheek Suresh, Sai Ashwin Ramaseshan, Balaji C

Indian Institute of Technology Madras

Email: me19d404@smail.iitm.ac.in

High-heat-flux-density data centres face significant thermal management challenges, particularly those housing 2.5D/3D packages for 5G and AI applications. Two-phase immersion cooling technology, which submerges heat-generating chips in a thermally conductive dielectric liquid like 3MTM NovecTM 7100 Engineered Fluid, has emerged as an efficient solution. This technology leverages the phase change from liquid to vapour to draw heat away from components, enhancing heat transfer efficiency. Server overheating can reduce efficiency and cause hardware damage, highlighting the need for effective thermal management strategies. Deep learning-based temperature prediction algorithms have shown promise in proactively managing the thermal condition of the system. We used a Raspberry Pi cluster to simulate the workloads of an actual data centre, including cryptographic workloads, Fast Fourier Transform (FFT) workloads, and training workloads based on data analytics. The dataset comprises the temperature, frequency and utilisation of each Raspberry Pi's processor. Experimental results show that attention-based encoder-decoder long short-term memory (LSTM) networks can predict the processor temperature with a mean absolute percentage error (MAPE) of less than 4% within a 60-second forecast window across all the workloads considered in the study.

Sensitivity of an Electrical Impedance-Based Sensor for the Liquid Fraction Estimation During Melting and Solidification Inside a Vertical Rectangular Enclosure

Carolina Mira Hernandez, Simone Mancin

Universita degli Studi di Padova

Email: carolina.mirahernandez@unipd.it

Low-cost latent thermal energy storage (LTES) with phase change materials (PCMs) can be hugely advantageous in decarbonizing heating and cooling loads by harnessing renewable energy sources. In addition, monitoring the liquid fraction in an LTES system can enable the implementation of smart control strategies for improved performance and increased utilization of highly variable renewable energy sources. The liquid fraction of the PCM determines the total stored energy, and its temporal evolution is representative of the heat transfer dynamics. However, measuring the liquid fraction is challenging because solid-liquid phase change processes occur at nearly constant temperature. The present study explores an electrical impedance-based sensing technique that could become a non-intrusive and accurate alternative to determine the liquid fraction. The study examines the melting and solidification of a PCM inside a vertical rectangular enclosure with an isothermal wall. A basic sensor configuration is proposed by locating a small set of electrodes at the boundary of the PCM encapsulation region. Initially, numerical simulations of the phase change processes for different wall temperatures are performed to generate physically meaningful solid and liquid phase distributions. Then, these phase distributions are used as input for electrical simulations to estimate the response of the electrode configuration to changes in the liquid fraction. The electrical simulations consider different values for the electrical properties of the liquid and solid phases. In this manner, a suitable range for the contrast in electrical properties between liquid and solid is determined that makes the proposed sensing approach feasible. These results are discussed in connection with reported electrical properties of PCMs and attempts to enhance these properties via the inclusion of nano-additives.

Verification and Calibration of Two-Fluid Models for the Analysis of Pressurization Scenarios in LH2 Tanks

Eugenio Schillaci, Ahmad Amani, Carles Oliet, Joaquim Rigola, Jesus Castro

Universitat Politecnica de Catalunya

Email: eugenio.schillaci@upc.edu

According to recent research trends, electrification in different transportation fields will pass through hydrogen-powered propulsion. To guarantee the necessary quantity of fuel within limited storage spaces, the hydrogen must be stored in liquefied form and isolated from the external environment. The H2ELIOS project will develop a lightweight, innovative and efficient LH2 aircraft storage prototype, ready to be integrated into the aircraft architecture for flight demonstrations at later stages. Hydrogen tanks would be subjected to highly variable conditions in terms of pressure and temperature, as well as by sloshing effects, triggering drastic evaporation and condensation processes. High-fidelity simulations constitute a fundamental tool for the study of basic concepts in this field. The current work focuses on the verification of two-fluid closure-models for momentum and energy equations. Two-fluid models for vapor-liquid two-phase flow systems have reached popularity thanks to studies related to nuclear reactor safety, to accurately predict the appearance of Critical Heat Fluxes in fuel rod bundles. However, their application does not have a high degree of maturity in fields of more recent development such as cryogenic hydrogen applications. In this work, the model is verified and calibrated by comparison to experimental references. The proposed numerical set-up models heat transfer through different media as the external shell, insulation and internal tank. Particular attention is given to the role of pressure and temperature dependent physical properties as well as in correlations for two-fluid closure models. The authors employ open source codes like OpenFOAM. The project H2ELIOS is supported by the Clean Aviation Joint Undertaking and its members. Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European union or Clean Aviation Joint Undertaking. Neither the European Union nor the granting authority can be held responsible for them.

Comparison Between DNS and RANS Approaches for Liquid Metal Flows Around a Square Rod Bundle

Danila Trane, Mattia Grespan, Diego Angeli

University of Modena and Reggio Emilia

Email: danila.trane@unimore.it

The thermal-hydraulic characteristics of liquid metal flows around rod bundles are of great interest for the research and design of fourth generation nuclear reactors. Currently, a large research effort is aimed at the development of accurate numerical models for low Prandtl number fluid flows, since the data available in the literature are quite scarce. Direct Numerical Simulation (DNS) is undoubtedly the most accurate approach, but its large requirements of computational resources and time make it less practical than other simplified methods such as the Reynolds-Average Navier Stokes (RANS) approach. The present paper provides a comparison between numerical results of a flow of liquid Lead-Bismuth Eutectic (LBE) at Pr=0.031 around four vertical cylindrical rods arranged in a square lattice, obtained by DNS and RANS. Several turbulence models are considered, including the standard k-epsilon, k-omega SST, and two Reynolds stress models, namely the ones by Launder, Reece and Rodi (LRR), and Speziale, Sarkar and Gatski (SSG). The accuracy of these models is assessed by comparing the mean Nusselt number, the pressure drop, and local field distributions with those obtained by DNS.

Understanding the Impact of Oblique Fin Angle on the Thermo-Hydrodynamic Performance of Oblique Microchannels Under Laminar Forced Convection Regime

Prasad Kangude, Parth S Kumavat, Evgeny Shatskiy, Anthony Robinson

The University of Dublin Trinity College

Email: kangudep@tcd.ie

Microchannel-based heat sinks have proven to be highly effective in transferring the high heat fluxes typically encountered in various applications, such as microelectronics, data centres, spacecraft, ultrafast data processing units, and high-power laser cooling. With the emergence of nanotechnology and the growing demand for high-performance computing, the need for efficient heat dissipation in these applications has increased more than ever. To ensure reliable operation under extreme thermal limits, it is crucial to explore various feasible methods towards the effective removal of excess heat. One of the widely explored approaches involves enhancing conventional microchannels by providing geometrical and topological modifications. Oblique microchannels classify as one of the potential approaches of advanced microchannels and has attracted significant attention. This is mainly due to their ability to induce the interruption and redevelopment of the thermal boundary layer and secondary flows, which promote heat transfer enhancement. It is well-established that the oblique angle significantly influences secondary flows and, consequently, the overall thermal-hydraulic performance. However, the literature reports conflicting results regarding the relationship between the oblique fin angle and the performance. Interestingly, despite the potential of oblique microchannels, there is a gap in knowledge to elucidate the underlying factors influencing this relationship. In view of this, the present study numerically investigates the influence of the oblique fin angle on the thermal-hydraulic performance of microchannels. A conjugate heat transfer model is developed with a uniformly heated wall using Ansys fluent. Three oblique angles (20, 40, and 60o) are tested across various Reynolds numbers ($400 \leq Re \leq 1000$), using water as the working fluid. The numerical results have been sufficiently validated against experimental results. Through a comprehensive analysis of flow and thermal parameters, this study identifies the key factors that determine the relationship between the oblique fin angle and the overall performance of the microchannels.

Enhancing Thermal Management in Cylindrical Li-Ion Battery Through PCM Integration with Variable Contact Area

Ekta Singh Shrinet, Lalit Kumar

Indian Institute of Technology Bombay

Email: ekta.shrinet@iitb.ac.in

Cylindrical Li-ion batteries are widely embraced in various sectors, notably electric vehicles and renewable energy storage systems. An effective thermal management system is vital for their safe and dependable operation, enhancing both the performance and reliability of the system. In this study, a distinctive hybrid cooling strategy is employed, featuring the integration of Phase Change Material (PCM) inside and liquid cooling outside of the cylindrical batteries with a variable contact area between the liquid coolant and the batteries. This approach is instrumental in maintaining thermal uniformity and regulating the maximum temperature. In the numerical analysis, we employed ANSYS Fluent 2020 R2 to create the computational model encompassing the Li-ion battery, PCM, and liquid cooling system. In this arrangement, eight 18650 Li-ion batteries were employed, with each battery housing a 2mm radius PCM rod inside. A heat-conducting element (HCE) was introduced to facilitate contact between the battery and the coolant channel. Water was selected as the coolant, and the coolant channel exhibited a cross-sectional area measuring 65×2 mm^2. The relationship governing the variable contact area is determined by the designed velocity and the initial battery contact area. The variable contact geometry maintains uniformity throughout the battery module while employing variable contact areas ensures a 74% enhancement in thermal uniformity. Nevertheless, the integration of PCM inside the battery effectively prevents individual batteries from surpassing specified temperature limits. The presence of PCM inside the battery plays a pivotal role in averting critical temperature thresholds. It yields a remarkable 20.16% enhancement compared to situations in which PCM is not present within the battery, albeit with a 3.75% capacity loss. Furthermore, the study expands its scope to thoroughly investigate the effects of coolant flow rates and perform an extensive parametric analysis of design variables at various C-rates.

Multi-Objective Optimization in Heat Exchanger Design and Operation

Reijo Karvinen

Tampereen korkeakouluyhteisossa

Email: karvinenreijo@gmail.com

Heat transfer using different types of heat exchangers governs many daily living and industrial applications. The size scale of heat exchangers is very large ranging from very small chips in electronics to huge equipment in process and energy industry. In the paper optimization procedures of following examples are given. 1. A wiring board of a mobile phone, where tens of components with known heat release must be located on the board so that the length of connecting wires and simultaneously component temperatures are small. 2. Sizes and locations of air jets impinging the moving hot glass in a glass tempering furnace in order to obtain uniform and high convective heat transfer in order to create compressive stress in the surface and tensile in the center and simultaneously to minimize energy consumption in the production of pressured air. 3. In electronicsheat generating components are located at the base plate of a heat sink, which is cooled by natural or forced convection. The flow can be laminar or turbulent. The main problem is to prevent overheating of components with minimum mass and volume in sinks. Thus, we have a multi objective optimization problem with different restrictions in heat transfer and weight. 4. We have different types of heat exchangers in pulp and paper industry. In a huge recovery boiler heat exchanger called economizer water is heated by flue gas of black liquor. The length can be even 30 meters and total weight one million kilograms. This is actually a counter-flow heat exchanger composed of tubes and fins. The main problem is the overall heat transfer coefficient between gas and water, which cannot be solved with CFD. In the paper procedures, where analytical, numerical and optimization methods are combined during the design or operation are presented.

Study of the Thermal Dynamic Behavior of a Rectangular Enclosure with Fins Embedded in PCM

Abderrahmane El Hanafi, Nicolas Blet, Abdelhamid Kheiri, Benjamin Remy

Universite de Lorraine

Email: abderrahmane.el-hanafi@univ-lorraine.fr

Phase change materials (PCMs) are used in thermal energy storage (TES) system to store/release heat in a wide range of applications with intermittent behavior, such as building and solar systems. Many parameters influence the thermal performances of PCMs. They must be used according to their phase transition temperature that shall matches the considered application. Moreover, the PCMs have a low thermal conductivity which affects the effectiveness of the dynamic thermal storage. Researchers considered different ideas to improve the heat transfer in a TES with PCMs. Few studies in the literature studied a rectangular enclosure with fins embedded in PCM. These studies have been mainly conducted under fixed boundary conditions and the key role of conduction and convection in the overall heat transfer in the PCM, from numerical considerations. In the present work, experimental and numerical devices are proposed to study the dynamic behavior of a rectangular enclosure with fins embedded in PCM (paraffin RT42®). The experimental device consists of a Plexiglas enclosure filled with PCM. An imposed cyclic temperature boundary condition controlled by Peltier cells is done on a copper plate on one side of the enclosure. An infrared camera is used to measure the 2D temperature fields through a Teflon® plate, infrared-transparent and thermally insulating. Copper fins are positioned perpendicular to the copper plate to improve heat transfer through the PCM. The effect of length of these fins is studied to optimize the design of the TES regarding its performances. Furthermore, using a CFD commercial software based on volume finite method, 2D numerical simulations are carried out and the findings are compared to the experimental results. The TES enhancements are evaluated by analyzing the liquid fraction of the PCM with different length of fins.

An Experimental Investigation of Forced Convective Liquid Immersion Cooling of a 12-Cell Li-Ion Battery Module

David W. Salter, Guillaume Bachelier, Parth S Kumavat, Seamus O'Shaughnessy

The University of Dublin Trinity College

Email: salterd@tcd.ie

This study details an experimental investigation of forced convective liquid immersion cooling of lithium-ion batteries. Twelve Samsung INR 18650 20S cylindrical cells, each with a nominal voltage of 4.2 V and nominal capacity of 2 Ah, are placed in a 4-in-series, 3-in-parallel arrangement inside a polycarbonate chamber. The dielectric fluid SF-33, which has a boiling point of 33.4℃ at 1 atm, is pumped through the chamber at volumetric flow rates selected to ensure laminar flow. The dielectric fluid is cooled after exiting the chamber by a plate heat exchanger connected to a recirculating water chiller, before returning to the chamber inlet at 20.5 ± 0.5℃. Flowmeters and thermocouples monitor the water and SF33 flow loops. Using a programmable power supply and electronic load along with a smart charger, the cells were charged and discharged at C rates in the range of 1C to 4C where the 1C rate corresponds to a 1-hour (dis)charge, 2C rate to a ½-hour (dis)charge, etc. Temperature sensors placed along the body of the cells near each electrode monitor their thermal behaviour during experiments. Results show that liquid immersion cooling is an effective method to maintain all cell temperatures within the desired temperature range of 15℃ to 35℃ for all cases tested, maintaining temperature differences within individual cells to ≤1.5℃, and those between cells to ≤2.3℃. Through optimisation these values can likely be reduced. Nevertheless, this research contributes to the understanding of thermal management strategies for lithium-ion batteries, particularly in scenarios involving high discharge rate applications.

Topology Optimization of Heat Exchangers

Ahmad Fawaz, Yuchao Hua, Steven Le Corre, Yilin Fan , Lingai Luo

Nantes Universite

Email: ahmad.fawaz@univ-nantes.fr

Reducing the energy consumption has been widely considered an essential topic in energy systems. In most of heat recovery systems, heat exchangers (HXs) play a crucial role to benefit from the energy carried by the emitted gases leading to significant cost savings. In recent decades, numerous active and passive methods were involved to improve the efficiency of HXs. Although the passive techniques such of using extended surfaces as fins has been well-demonstrated one of the most effective techniques in improving heat transfer rates. In the early stages, researchers designed fins intuitively based on their background and understanding of physics. Thereafter, due to the advances in computer technology and numerical simulations, the possibility of designing optimal fins increased with the help of optimization algorithms. Fin size/shape optimization has been developed for years; however, it cannot considerably changes the configuration or arrangement provided by designers. Different from the aforementioned techniques, the topology optimization (TO) operates on the design domain's topology by optimizing the material (solid or fluid) distribution spatially. In the current work, the density-based TO is conducted on a 2D counter flow HX unit for the purpose of enhancing its thermo-hydraulic performance. The optimization objective is the maximization of the exchanged heat between cold and hot fluids while the pressure loss is set as a constraint. To validate the design methodology, high fidelity CFD simulations are conducted on the optimized HX to evaluate its thermo-hydraulic performance and compare it with a benchmark HX. The performance evaluation criteria (PEC) that fractionally combines the Nusselt number and friction coefficient raised to 1/3 is used to evaluate the simultaneous thermo-hydraulic performance of the benchmark and optimized HXs. At Reynolds number = 514, the thermo-hydraulic performance of the TO-optimized HX is higher than the benchmark case with a 43 % improvement in the PEC number.

Modelling Turbulent Gas-Solid Heat Transfer in a Packed Bed Heat Exchanger: Experiment and Validation

Jérémie Lagarde, Harry Simpson

SynchroStor Ltd

Email: j.lagarde@synchrostor.co.uk

SynchroStor Ltd. is developing a system for Pumped Thermal Energy Storage based on the Closed Brayton Cycle principle. To increase power density, the high temperature part of the cycle is also at high pressure (around 200 bar); a pressure that would be unreasonable to use with coupled packed-bed thermoclines. A key element of SynchroStor's system is a direct contact moving packed-bed heat exchanger in which a low-cost, non-toxic solid material "flows through" to allow for efficient heat transfer. The heat exchanger is coupled to an air lock system that allows to pressurize/depressurize the solid material from/to ambient pressure, before transporting to/from the stores. Evaluation of process conditions and sizing heat exchangers requires accurate modelling of the convective heat transfer that occurs between gaseous working fluids and solids. Existing correlations have been developed to predict heat transfer, though the range of experimental data limits applicable flow conditions to flows with Re <= 10,300. A series of tests were conducted across a range of flow conditions, whereby high temperature, high pressure gas was pumped through a packed bed of uniform spherical glass beads. Tests were conducted with Re in the range of 5,000 to 15,000. Practical challenges of distinguishing measurement of solid and gas temperatures in the heat exchanger vessel mean analysis of heat transfer performance is greatly assisted with a model. A gas-solid direct contact heat transfer model has been developed in the MATLAB/Simulink Simscape domain, which will be validated against the available experimental data. The study aims to use the model to improve data analysis and develop a new Nusselt number correlation for convective heat transfer or evaluate if existing correlations are suitable for flows Re <= 15,000.

Modeling Thermal Dynamics in Intermittent Operation of a PEMEL for Green Hydrogen Production

Elisabet Mas De Les Valls, Roser Capdevila Paramio, Juliana Jaramillo, Wilhelm Buchholz

Universitat Politecnica de Catalunya

Email: elisabet.masdelesvalls@upc.edu

The intermittency of renewable power generation like photovoltaic has generated significant interest in the development of carbon-free energy carriers like green hydrogen. Water electrolyzers efficiently convert surplus energy into hydrogen and oxygen, responding rapidly to intermittent voltage sources, and their effectiveness continually improves through advances in materials and designs. Proton exchange membrane water electrolyzers (PEMWE) are particularly appealing due to their high current densities, technical maturity, and quick electrochemical response. Nevertheless, thermal processes are considerably slower compared to electrochemical ones, introducing significant thermal inertia that can impact system efficiency. Therefore, optimizing the thermal management of the electrolyzer is critical, necessitating the development of experimentally validated thermal models. This work discusses the development of a transient thermal model that accounts for the influence of large intermittencies on the electrolyzer's temperature. The model utilizes an interconnected framework that couples electrochemical, thermal, and mass transport submodels. Ordinary differential equations are employed to enable the code's application during large transients while keeping computational costs to a minimum. Special attention is given to modeling heat losses to the atmosphere, incorporating both natural convection and radiation. The resulting model has been experimentally calibrated under steady-state conditions using a single-cell PEMWE test bench. This setup collects thermal, electrical, and hydrogen production data and provides control over the cell potential and inlet water temperature. In a second experimental phase, the electrolyzer was operated according to a typical intermittency pattern, mirroring the surplus energy resulting from the misalignment between photovoltaic production and a typical electricity consumption curve. The experimental data collected during this phase was used to validate the developed transient thermal model under the same intermittency pattern. The feasibility of a simplified numerical thermal model for capturing transient thermal effects within a PEMWE is discussed, with an emphasis on identifying future developments needed to study large PEMWE stacks.

Numerical Investigation on Liquid Cooling of Batteries in Phase Change Materials

Aanandsundar Arumugam, Bernardo Buonomo, Pasquale Romano, Oronzio Manca

Universita degli Studi della Campania Luigi Vanvitelli

Email: aanandsundar.arumugam@unicampania.it

Electric cars, in addition to representing an ecological solution for the serious problems of climate pollution due to the use of fossil fuels, can also represent a turning point in terms of renewal for the world economy with a product that in a short future will become a standard in all the advanced economies of the globe. One of the main problems of electric cars is given by the thermal control of their batteries, since, below and above a certain temperature range and also with the use of air conditioning, they abruptly decrease the vehicle autonomy. In this study, an attempt is made to control both thermal aspects by trying to thermally isolate the battery from the vehicle's external climate and by trying to control the temperature peaks due to the operation of the battery itself. A three-dimensional model is considered to investigate numerically the thermal control of a cylindrical lithium polymer based battery as a single module. The battery has a height of 65 mm and a diameter equal to 18.4 mm and its capacity is equal to 3000 mAh. The thermal control is realized by means of a phase change material, PCM (paraffin), with a single module where a cylindrical battery is placed inside in the center of a parallelepipedal and cooled by a convective flow by means tubes. The tubes are assumed at assigned temperature and the surface of the module is considered adiabatic. The governing equations are solved by finite volume method using the commercial code Ansys-Fluent. Different cases are simulated for two different PCMs at C-rate equal to 0.25, 0.5 and 1.0. The results are given in terms of temperature and liquid fraction fields, surface temperature profiles as a function of time and temperature distributions along the external surface of battery.

Design of Thermal Management System for a Battery Pack with Malfunctioning Battery: A Numerical Parametric Study

Sayan Majumder, Rajesh Akula, Balaji C

Indian Institute of Technology Madras
Indian Institute of Technology Bombay

Email: sayaniitmadras@gmail.com

Lithium-ion batteries are well-known for their high energy density, rechargeability, and versatility in powering various devices. However, effective thermal management is essential to ensure their safe and efficient operation. In this study, a hybrid cooling technology comprising phase change material (PCM) and water-based active cooling was employed for a battery pack arrangement with a circular configuration. The numerical model comprised simulated 18650 lithium-ion batteries made of aluminium, which generate heat, housed within a cylindrical steel casing filled with a PCM. The cylindrical aluminium batteries were symmetrically positioned at a 30 mm distance from the center. The PCM utilized in this investigation consisted of expanded graphite (EG) combined with paraffin (PA). Numerical simulations were carried out using the ANSYS-FLUENT 2020 R2 edition, and the same is validated against the experimental results. A comparative analysis was conducted on various battery configurations, including 4-battery and 6-battery setups, subjected to constant heat loads of 5W, 10W, and 15W, cyclic heat loads where batteries generate a constant 5W, 10W, or 15W for 500 seconds within each cycle, followed by a 500-second interval with no heat load, and charge-discharge cycle analyses. In the latter analysis, three batteries operated at 5W while one malfunctioning battery operated at 15W, and another analysis involved a 60W malfunctioning battery alongside five 5W batteries. The cooling system employed aluminium pipes of 1 mm thickness, utilizing water as the working fluid. Two configurations were tested, including a single pipe and four pipes. Preliminary results revealed that the hybrid cooling system with a single pipe reached a maximum temperature of 340K for the malfunctioning battery configuration within mere 500s, whereas the four-pipe configuration reached a maximum of 321.8K before achieving a steady-state condition. These findings underscore the effectiveness of the proposed cooling mechanism, even when handling heat loads exceeding the batteries' rated capacities.

Experimental Study of Convective Heat Transfer in Additive Manufactured Mini-Channels: The Impact of the Prandtl Number

Mohammadreza Kadivar, Luca Cozzarolo, Mats Kinell, Karl-Johan Nogenmyr, David Tormey, Gerard Mcgranaghan

Atlantic Technological University

Siemens Energy AG

Siemens Industrial Turbomachinery AB

Email: mohammadreza.kadivar@atu.ie

Additive Manufacturing (AM) is an advanced manufacturing technology that has received significant attention in the fabrication of heat transfer devices, particularly functional cooling channels. Due to natural phenomena in AM processes, the internal surface of the fabricated channels is rough, characterized by random irregularities. While a substantial amount of literature focuses on the impact of roughness on flow friction, heat transfer over rough surfaces has received less attention. Previous studies have predominantly investigated airflow conditions with Prandtl numbers in the region $Pr \approx 0.71$. The present study seeks to investigate differing ranges of Prandtl number. The study experimentally investigates the forced convective heat transfer of water in AM mini-channels which are fabricated using Laser-based Powder Bed Fusion (L-PBF) technology. The experimental method utilizes novel Joule heating to generate heat within the solid body of the channel, directly heating the water inside the channel and eliminating the effects of contact resistance. The flow Prandtl number was varied ($5 < Pr < 8$) by changing the water inlet temperature, regulated by a chiller. The results demonstrate that the roughness effect is more pronounced at higher Prandtl numbers; however, there is a saturation in the roughness effect on heat transfer at higher Reynolds numbers. Changing the Prandtl number led to variations in heat transfer enhancement by roughness at a given Reynolds number, indicating that the $Nu \propto Pr^{0.4}$ scaling is not valid for rough channels.

Review of “Cold Shock” Cases in Operation of Loop Heat Pipes and Related Thermal Instabilities

Luka Ivanovskis, Donatas Mishkinis

Allatherm SIA

Email: luka.iv.sky@gmail.com

Loop Heat Pipes (LHP) are passive two-phase heat transfer devices, driven by capillary pumps that source energy for circulation of working fluid solely from the heat source. They have a proven track record in spacecraft thermal control as well as terrestrial applications. Ever new technology developments are reported, as for the heat flux density, compactness of evaporator, width of temperature range and so on. However, LHP may experience partial or complete wick dryout and thermal instabilities at rapid power-up steps due to a rapid inflow of subcooled liquid from the condenser into the evaporator. Such cases do not have very wide coverage in literature and are usually presented as a little nuisance in testing of a particular LHP, for which some design workaround was found. Oftentimes suggested solutions trade performance or other design merits for robustness of operation, while leaving the theoretical coverage of the observed phenomena out of the scope of report. This article aims to review and systematize such case studies where phenomena similar to the described "cold shock" affected LHP performance. The aforementioned instabilities become even more pronounced in grid-like multievaporator LHP due to multiple parallel connections between evaporators and compensation chambers, giving paths and possibility for occurrence of parasitic heat and mass transfer, detrimental for overall LHP performance because of a decent increase in heat leak. Vulnerability of LHP with distributed network of evaporators to seemingly "safe" transient regimes fosters deeper analysis of the "cold shock" phenomenon. With comprehensive definition to the problem, technical measures for getting rid of it in perspective LHP designs can be foreseen.

Free-Surface Jet Heat Transfer Revisited: From an Indifferent Regime to the Re-Emergence of the Off-Center Peak

Herman Haustein, Ron Harnik

Tel Aviv University

Email: hermanh@tauex.tau.ac.il

Laminar free-surface jet impingement is a common configuration for heat transfer processes, though involves diverse convection phenomena. Following the understanding that the stagnation-point heat transfer depends directly on the near-axis radial acceleration, its dependence on arriving jet width and profile shape is studied in detail. Thus, heat transfer is expected to vary with: Fluid properties, Flow rate, normalized nozzle length, normalized nozzle-to-plate spacing, surface tension and gravity (Pr, Re, L/d, H/d, We & Fr, accordingly). As existing theory is very limited in terms of relevant geometries and forces, a new general description is developed. In this study experiments and two-phase flow simulations were employed to validate the predicted flow and heat transfer for all relevant conditions. Correspondingly, the new description addresses the key dynamics of the jet's centerline velocity magnitude and curvature decay and evolving jet width during flight. The impingement part, adapts a previous streamline-bending analysis to tie these arrival characteristics to wall pressure distribution and resulting radial acceleration. The new description is shown to capture the flow and heat transfer up to the point of boundary layer emergence from the liquid film, beyond which Watson's classical solution applies, when appropriately scaled as shown here. Finally, it is seen that the present analysis is physically sound and is able to continuously capture non-monotonous and diverse phenomena: both the occurrence and re-emergence of an off-center peak in the heat transfer, corresponding to under-developed or over-relaxed arrival profiles – a phenomena previously dealt with only in a case-specific manner. The study has also led to the clarification, identification and quantification (physical criterion) of the "indifferent jet" regime – wherein gravity's acceleration is countered by the profile relaxation to generate a constant-centerline velocity jet. Such a jet's stagnation heat transfer is indifferent to the nozzle-to-plate spacing, and has multiple useful applications.

Modulation of Flow, Heat, and Mass Transfer in Turbulent Double-Diffusive Convection

Sasa Kenjeres, Rona Roovers

Delft University of Technology

Email: s.kenjeres@tudelft.nl

Combined temperature and concentration field interactions under the influence of the gravitational field are defining so-called double-diffusive convection phenomena. These phenomena are present in various situations in nature and technological applications, which include mixing in upper layers of oceans, seas, and lakes, solar pond energy storage systems, etc. In the present work, we focus on the double-diffusive convection within finite-size enclosures (1:1:4 and 1:1:8) filled with a working fluid with high values of both thermal (PrT=9) and concentration Prandtl numbers (PrC=2000), which are closely matching recent experimental studies of Rosenthal et al. (2022). We focus on highly turbulent regimes at a relatively very high concentration Rayleigh numbers (1011<RaC<1013), and a wide range of thermal Rayleigh numbers (0<RaT<1013). To be able to cover this range of working parameters, we adopt the wall-resolving LES approach with Vreman's sub-grid closure. We focus on the instantaneous local heat and mass transfer along the thermally and concentration active boundaries, and how these are affected by 3D flow, thermal, and concentration structures (e.g. thermal and concentration plumes). The power spectral density (PSD) and histograms of the time series of instantaneous velocity, temperature, and concentration signals at characteristic monitoring points placed within the boundary layers in the proximity of horizontal walls and at the center of the enclosure are compared. Furthermore, the dependency of the integral Nusselt and Sherwood numbers for variously imposed thermal stratification is analyzed. Finally, changes in the vertical profiles of the long-term time-averaged first- (mean temperature and concentration) and second-order (rms of velocity, concentration, and temperature variances, as well as of the cross-correlations (turbulent heat and concentration fluxes)) statistics are discussed in details.

Engineered Advanced Materials for Latent Thermal Energy Storage

Amit Mishra, Alessandro Ribezzo, Matteo Morciano, Luca Bergamasco, Matteo Fasano, Eliodoro Chiavazzo

Polytechnic University of Turin

Email: amit.mishra@polito.it

Phase change materials (PCMs) are capable of reversibly storing and releasing large amounts of thermal energy across the isothermal phase transition and offer great potential for the design of advanced renewable energy infrastructure. We present a thorough review of the use of additives in PCMs as well as their advanced functionalization, emphasizing both key experimental and theoretical aspects. The study delves into the enhancement of PCMs' thermophysical and optical properties, addressing – for instance the challenge of low thermal conductivity which can lead to protracted charge and discharge periods and suboptimal heat storage. Particularly, focusing on classical pristine solid-to-liquid PCMs, we scrutinize the influence of additives and the protocols for their incorporation on the resulting thermo-physical and stability properties of PCMs. Furthermore, we discuss details on new approaches aiming at generating shape-stabilized composite PCMs to focus on improving heat transfer as well as their practical application suitability along with energy conversion efficiency. Our critical analysis of the current literature reveals gaps in the comprehensive understanding of the optimal use of specific additives and supporting structure materials for engineering PCMs to achieve desired thermal/optical properties. As an example, the thermal resistance at the filler-matrix interface poses a substantial barrier to heat transport, and current methods of managing this aspect are not yet fully effective. Furthermore, the key issue of both achieving high energy storage capacity and long-term cyclability stability of advanced PCM composites is thoroughly analyzed and discussed. The paper calls for advancements in experimental techniques to develop more effective composite materials for thermal and solar energy storage. Our work emphasizes the necessity for more comprehensive theoretical work to better understand and model heat transport in advanced engineered PCMs. Finally, critical remarks on the emerging solid-to-solid transition materials are also made, as they hold significant promise for long-term heat storage solutions.

Experimental Analysis of an Innovative Geo-Exchange System Installed on the Island of Ischia, in Southern Italy

Vincenzo Guida, Nicola Massarotti, Alessandro Mauro

Parthenope University of Naples

Email: ing.guida.vincenzo@gmail.com

The need to use efficient and eco-friendly air conditioning systems is a fundamental requirement for today's society. In this context, low and medium enthalpy geothermal energy plays a crucial role. The research and development activities carried out in the present work have made it possible to successfully design and analyze an innovative technology capable of supplying thermal energy to environments without pumping fluid from the subsoil. The proposed system is based on the use of a downhole heat exchanger (DHE). The authors have developed an innovative system for the sustainable use of low and medium enthalpy geothermal energy, installed on the island of Ischia, near Napoli, in southern Italy. The proposed system is based on the use of an ad hoc designed DHE, capable of optimizing the heat transfer with the subsoil, without the need to withdraw fluid from the aquifer. A numerical finite element model was developed to study the interaction between the DHE, the well and the surrounding aquifer. The experimental set-up consists of the heat exchanger and an above ground system, necessary to test the efficiency of the exchanger. The DHE is inserted inside a geothermal well made with a steel casing, equipped with a filtering section in correspondence of the DHE, in order to increase the heat transfer performance due to increased convection with the surrounding aquifer. The experimental data show that the DHE allows to exchange more than 40 kW with the ground, obtaining overall heat transfer coefficient values larger than 450 W/m^2 K.

Structured Illumination with Infrared Imaging for Measuring Thermal Conductivity

Ashwath Bhat, Chris Dames

University of Californoia, Berkeley

Email: ashwathbhat@berkeley.edu

With developments in advanced manufacturing and materials by design comes the need for high-throughput thermal characterization and inspection. Towards this end, Structured Illumination with Thermal Imaging (SITI) is an all-optical pump-probe thermal characterization technique recently developed by our group. In the first generation [Zheng et al., Appl. Phys. Rev. 9, 021411 (2022)] SITI uses an LED with a digital micromirror device (DMD) to "structurally illuminate" and heat the sample with dynamic patterns, a visible light camera for thermoreflectance based "thermal imaging" [leveraging a Microsanj MTIR120], and the resultant temperature response was fit with a thermal model to characterize the sample's thermal properties. This represents a novel approach to dynamic and flexible spatial mapping of thermal properties by virtue of being a non-contact technique and having a simpler scanning means (computer control only) than conventional pump-probe laser methods. SITI also can tolerate rough samples with diffuse reflections. This talk presents the second generation of SITI. The pumping is now based on a lower cost off-the-shelf digital projector. The thermometry is now performed using an infrared (IR) camera, which we find is a more flexible and accessible hardware approach compared to the thermoreflectance microscopy used previously. With these updates the setup can deliver higher heating power and a broader range of frequencies, allowing an extended range of samples that can be studied. We have demonstrated SITI's ability to measure the thermal conductivity of a range of materials from polymers to metals.

LES Investigations of Turbulent Heat Transfer Structures with Exponential Power Escalation

Elie Roumet, Raksmy Nop, Nicolas Dorville, Marie-Christine Duluc

Commissariat a l'energie atomique et aux energies alternatives
Université Paris-Saclay, CEA
CNAM

Email: elie.roumet@cea.fr

The BORAX-type accident is a key scenario in the safety design of pool-type experimental nuclear reactors. It considers a reactivity insertion large enough to initiate an exponential power escalation with a period as small as a few milliseconds, necessitating prompt and effective heat removal by the coolant system. This study presents a comprehensive computational analysis of single-phase transient heat transfer under thermal-hydraulic conditions relevant to such reactors, focusing on turbulent channels flow formed by planar fuel plates with highly subcooled water. Our investigation uses Quasicompressible (Low Mach approximation) Large Eddy Simulation (LES) performed with TrioCFD, the open-source CFD code developed by the French Atomic Energy Commission (CEA). These simulations incorporate high subcooling levels (up to 160 K), varying mass fluxes (from 300 kg/m^{2}s up to 15,000 kg/m^{2}s), diverse power escalation periods (5, 20, and 100 ms), and two different channel widths. We ensure detailed turbulence resolution at the Batchelor scale near the heated wall and confine subgrid viscosity modeling to isothermal flow regions. The simulation framework was first validated against Direct Numerical Simulation (DNS) results for steady-state heating scenarios. After this initial step, our analysis extended to transient heating, incorporating comparisons with experimental infrared thermometry measurements of the wall heat transfer. Subsequent analyses focused on the impact of temperature-dependent fluid properties on turbulent structures and heat transport during a power transient. Early in the transient phase, peak temperatures are confined within turbulent streaks, as reported in the existing literature. However, as the exponential power transient unfolds, the viscous layer and streak structures destabilize, leading to a more dispersed distribution of thermal hot spots. Our preliminary findings suggest that conventional heat transfer correlations tend to underestimate the heat transfer rates, especially when the influence of temperature-dependent fluid properties is not considered.

Particle Dispersion Produced by a Turbulent Free Convection Flow in a Room-Size Cubical Cavity

Jordi Pallares, Akim Lavrinenko, Cristian Marchioli, Salvatore Cito, Alex Fabregat

Rovira i Virgili University
University of Udine

Email: jordi.pallares@urv.cat

This paper introduces the framework for the ongoing "2024 International CFD Challenge on the Long-Range Indoor Dispersion of Pathogen-Laden Aerosols." The Challenge is designed as a blind test to assess the accuracy of computationally efficient turbulence modeling techniques, including URANS and LES, in replicating both the hydrodynamics and aerosol dispersion in an idealized indoor environment. To evaluate the simulations, DNS data of turbulent natural flow at a high Rayleigh number within a room-sized enclosure will serve as a reference benchmark. Participants have the flexibility to conduct simulations of the same flow configuration using their preferred CFD software, employing URANS, LES, and/or hybrid methods. The Challenge was officially launched on October 16, 2023, and has garnered participation from 31 teams representing 18 different countries, with the expected submission of results in May 2024. The outcomes of the comparison between the different modelling approaches and the reference DNS will be presented and discussed during the conference.

Experimental Investigation of Effects of Convective Heat Transfer and Thermal Performance of Impinging Jet on a Metal Plate with Porous Media

Abdul Qadeer Khoso, Atiq Ur Rehman Fareedi, Hurmat Khan, Bernardo Buonomo, Oronzio Manca, Sergio Nardini

University of Campania "Luigi Vanvitelli"

Email: aqkhoso47@gmail.com

In the present study an experimental investigation is accomplished to evaluate the heat transfer phenomena in an aluminum metal flat disc/plate using a single round jet of air impinging on a metal foam in a confined region. To understand the heat transfer phenomena in a metal plate (of radius 50mm) with metal foams (of diameter 40mm and height 20mm) facing round air jet of diameter 14mm via impingement; four metal foams of similar geometries but of different porosities and permeabilities are taken into consideration to study, named as s1, s2, s3, s4. Each experiment is performed with seven different flowrates including 10000, 5000, 3500, 2000, 1000, and 500 (liters/hr.) measured by rotameter connected with the compressor and at two different heights from air jet exit with respect to the plate: 20mm and 40mm while the metal foam is placed on the aluminum disc/plate heated by an electric resistor at two different voltages 10V and 20V. There is a thin film of thermal paste between aluminum metal disc/plate and the metal foam. A total of 10 different thermocouples are used in this experiment of which five of them are placed on the plate at different radius ratio positions of 0.9, 0.5, 0.2, 0.1 and 0. The recorded measurements are temperatures, pressure differences, flow rates, resistance voltages and currents, and time. The outcomes of the experimental data are analyzed, and graphs are made to compare results of all four different kinds of metal foams with different jet to plate height and at different heat fluxes. The conclusions are very useful in engineering applications and Jet impingement techniques can be applied in various applications, mainly heating and cooling such as cooling of turbine blades, cooling of electronic devices, drying etc.

Effect of Using an Ionic Liquid as Additive in Water/LiBr Solution for a Horizontal Falling Film Absorber

Hussain Ahmed Tariq, Mahmoud Bourouis, Alberto Coronas

Universitat Rovira i Virgili

Email: hussainahmed.tariq@urv.cat

Absorption Heat Pumps (AHPs) excel at producing cooling using low grade energy resources. This technology becomes particularly compelling when employing water/LiBr mixture as the working fluid. Water, being a natural refrigerant, cost-effective, non-toxic, and environmentally friendly, holds particular significance in this context and mitigates contributions to global warming. Nonetheless, water/LiBr AHPs encounter certain inherent issue that hinder the operation of this technology to be air cooled and operate in reversible mode for heating, as LiBr crystallizes at higher concentrations. The solubility of LiBr in water/LiBr solution can be improved using Ionic Liquids (ILs) as additives in the solution. In this work, 1,3-dimethylimidazolium chloride [DIMIM][Cl] Ionic Liquid will be utilized as additive in water/LiBr solution to increase the solubility of water/LiBr solution. Horizontal tube falling film absorber will be used to study the heat and mass transfer of new solution ([DIMIM][Cl]+LiBr)/water solution. The absorber is made of six horizontal tubes with solution distributor at the top. The cooling water flows inside the tubes in a crossflow direction to the solution.

Numerical Insights into Turbulent Penetrative Convection Over Localized Heat Sources

Sasa Kenjeres, Amir Zilic, Kemo Hanjalic

Delft University of Technology

Email: s.kenjeres@tudelft.nl

The penetrative convection describes the vertical heat flux transport in an initially thermally stratified environment. This situation is typical for the onset of the early morning heating of the earth’s surface caused by incoming solar radiation. In the present study, we focus on performing a series of numerical simulations over a range of various localized heat sources (with different strengths of heating and shapes of heat source) for different scenarios of the initially stable thermal stratification. To model turbulence effects, we apply two approaches: (i) the wall-resolving large eddy simulation(LES) with a non-isotropic filtering approach to represent the sub-grid turbulence, and (ii) the time-dependent Reynolds-Averaged Navier-Stokes (TRANS) with the algebraic form of the non-isotropic turbulent heat flux model (AFM) embedded into the three-equation (k-ε-θ) scheme that accounts for the dynamics of the fluctuating temperature and its effect on heat transfer. We cover an intermediate range of Rayleigh numbers, i.e. $107<Ra<109$. In contrast to the LES approach, where a very fine numerical mesh is employed with local refinements in the proximity of the ground and stratification interface, the TRANS employs the buoyancy-extended wall functions for turbulence parameters and much coarser uniform mesh in the vertical direction. In addition to the comparison of classical parameters between the LES and TRANS (vertical profiles of the temperature, temperature variance, vertical turbulent heat flux, turbulent kinetic energy), we focus on the analysis of the flow and turbulence coherent structures by using various identification and extraction techniques (e.g. vorticity, helicity, second-invariant of the velocity gradient tensor, etc.). The role of these coherent structures in the transport of momentum, heat, and turbulence in the proximity of the heat sources will be analyzed in detail.

Numerical Simulations of Temperature Loads of Multilayer Laue Lenses

Zlatko Rek, Henry N. Chapman, Sasa Bajt, Božidar Šarler

University of Ljubljana
Universität Hamburg
DESY
Institute of Metals and Technology, Ljubljana, Slovenia

Email: zlatko.rek@fs.uni-lj.si

X-ray free electron lasers (XFEL) provide intense, coherent and pulsed X-ray beams. Amongst other things, they are used in single-shot imaging of biological nanocrystals. The short wavelengths and short durations of XFEL pulses allow doing these studies with Ångstrom spatial and femtosecond temporal resolution. XFEL beams have to be focused. A novel type of X-ray optic, multilayer Laue lenses (MLLs), was used for the first time to focus X-rays at the European XFEL a few years ago. An MLL is a diffractive optic where the diffracting layers are deposited onto a substrate. Numerical simulations, presented here, indicated the cause of rapid heating by the pulse and helped to improve the lens designs. In these numerical simulations we investigated heating of MLLs consisting of different material pairs, geometries and ways of mounting, taking into account the special pulse structure of facilities such as the European XFEL. There, X-rays are delivered in bursts or pulse trains. Different materials have different heat capacities and thermal conductivities. We investigated materials for MLLs as well as their mounts geometry, considering both the X-ray optical performance and the thermal management. After defining the geometric model, computational grid, material properties, and boundary conditions we performed a grid sensitivity study. In the presented numerical simulations, we solved the transient heat energy transport equation in solids for mixed boundary conditions. We calculated temperature distribution inside MLLs and studied how the temperature changes within the short time between pulses and after being exposed to 20 XFEL pulses. Finally, a promising MLL design based on a new material pair and geometry that reduces the X-ray dose in the structure (energy deposited per unit mass) and effectively dissipates the heat will be presented. This new design is planned to be tested in an upcoming experiment at the European XFEL.

Numerical Modelling of a Solar Thermochemical Heat Storage System for a Concentrated Solar Power Plant

Abhishek Singh, Zhen Cao, Ramin Roushenas, Bas J. Leeuw

University of Twente, Enschede, the Netherlands

Email: a.k.singh@utwente.nl

A redox oxide-based high-temperature thermochemical heat storage (TCHS) system could address the intermittency challenges of solar energy in a concentrated solar power plant. In the present research, the copper oxide pair (CuO/Cu_2O) which interacts directly with air to facilitate the uptake and release of oxygen, is being considered as a storage material. A 2-D, axisymmetric numerical model to simulate the heat and mass transfer coupled with the chemical kinetics has been developed using COMSOL. The model is validated using experimental results from the literature. The transient air inlet temperature and mass flow rate obtained from the experiments are utilized as boundary conditions to validate the model. The temperature profile generated using the numerical model at various locations within the storage system is compared against the experimental values. The error between the numerical model output and the experimental results is found to be below 10% for both the charging and discharging processes. The validated model is used to perform parametric studies to gauge the influence of various operational and geometrical parameters on the performance of the storage system. This study enhances the understanding of the copper oxide-based TCHS process and helps to identify the effect of variation of boundary conditions on the system.

Experimental Investigation of Phase Change Material (PCM) Thermal Energy Storage with and Without Metal Foam at Different Temperatures

Abdul Qadeer Khoso, Atiq Ur Rehman Fareedi, Bernardo Buonomo, Abdul Qadeer Khoso, Hurmat Khan, Oronzio Manca, Sergio Nardini

University of Campania "Luigi Vanvitelli"

Email: aqkhoso47@gmail.com

This paper presents an experimental investigation of indirect phase change materials (PCM) thermal energy storage. The heat exchanger setup consists of a square channel of 20cm X 20 cm dimensions with single flow tube with 5 swirls having distilled water as a circulating fluid. PCM type RT-42 (melting Temp. 37-42 ^{o}C) is used a storage material. In this experimental setup we have designed the channel in two configurations with and without Aluminum metal foam having a porosity of 95% is used. For each configuration, the thermal energy storage for PCM is analyzed and compared for two temperature variations at 55^{o}C and 65^{o}C, respectively. A thermal bath with a flow pump is used to achieve the desired temperatures while, numerous in house calibrated J-type thermocouples are employed to measure the temperature variations at different locations within the experimental setup. A flow meter at the exit of channels is installed and constant flow rate of 2.70 L/min is maintained throughout the experimentation. To minimize thermal losses through the channels, an insulation in the form of double glazing is used at the top while a polyethylene foam is used as an insulation at the bottom. The conclusions suggest that heat storage rate in Metal foam (MF) integrated PCM thermal energy storage is better as compared to the setup without metals foam for both temperatures (55^{o}C and 65^{o}C). It is also observed that, the heat transfer rate for 65^{o}C is comparatively greater than that of 55^{o}C for both systems. Overall, this study concludes that, the heat transfer rate is higher with integrated metal foam PCM setup while, the higher input temperatures results in high heat transfer rate.

Topology Optimization of Pcm-Based Finned Heat Sink for Cooling a System-on-Chip

Sayan Majumder, Azharuddin Mohammad, Balaji C

Indian Institute of Technology Madras

Email: sayaniitmadras@gmail.com

PCM (Phase Change Material) based finned heat sinks represent a cutting-edge solution for addressing the intricate challenges of electronic cooling. These heat sinks leverage the unique properties of phase change materials, substances that undergo a phase transition between solid and liquid states while maintaining a constant temperature. By exploiting the latent heat absorbed or released during the phase transition, PCM-based heat sinks effectively buffer temperature fluctuations, preventing electronic components from reaching critical thermal thresholds. The incorporation of fins in the heat sink design enhances heat dissipation by maximizing the surface area available for convective cooling. This synergistic combination of PCM and finned architecture ensures a more sophisticated and reliable means of dissipating heat generated by electronic devices, thus improving overall system performance. In this study, we have considered a system on Chip (SoC) where a single CPU, two GPUs and one RAM is present. The average heat flux generated from the SoC is about $20 W/cm^2$. Optimizing the material distribution of PCM-based heat sink space is critical for enhancing the thermal performance of the system. Utilizing a Topology Optimization methodology facilitates the generation of innovative designs devoid of predetermined assumptions regarding fin structures. This numerical study employs the Solid Isotope Material with Penalization (SIMP) technique to formulate the optimization problem, wherein a fictitious material is introduced whose thermophysical properties are interpolated by a continuous density parameter ranging between 0 and 1. The optimization problem incorporates solely heat load and boundary conditions, and the Method of Moving Asymptotes (MMA) solver, implemented in the COMSOL Multiphysics commercial software, is utilized to solve the optimization problem, leading to the creation of innovative fin designs. Initial simulation results indicate roughly a 12% increase in the time required to achieve the set point temperature on the chip compared to the unfinned System-on-Chip (SoC).

CFD Design and Optimization of a Multi-Probe Device for Temperature Measurements in Waste-to-Energy Plants

Giorgio Grossi, Fausto Arpino, Christian Canale, Gino Cortellessa, Giorgio Ficco, Tonino Lombardi

University of Cassino and Southern Lazio
HERAmbiente S.p.a

Email: giorgio.grossi@unicas.it

Monitoring the temperature of flue gases is a critical concern for Waste-to-Energy (WtE) plant operators, as it is essential for preserving the materials in the post-combustion chamber, optimizing energy efficiency, and managing pollutant emissions. Current methods for monitoring flue gases temperature in such plants (i.e., thermocouples, infrared pyrometers, and aspirated thermocouples) have inherent limitations in terms of accuracy and reliability within post-combustion chambers. In addition, they do not allow to obtain a temperature profile (required to validate the numerical codes employed to verify the plant operating conditions) with a single immersion measuring the temperature at different depths simultaneously; the same instrument must be moved back and forth resulting in an uncertainty of the position and measurements in correspondence of potentially different operating conditions of the WtE plant. In previous research, by means of Computational Fluid Dynamics (CFD) simulations, the authors realised the thermo-fluid dynamics design of a novel device, with the aim of mitigating the issues associated with the existing techniques. It is air-cooled and relies on the use of one thermocouple to measure the inner wall temperature and of three thermocouples to measure the temperature of flue gases flowing in the post-combustion chamber. Thanks to the designed shape, the framework itself shields the sensing elements from the radiative heat exchange between the sensors and the chamber walls. In the present paper, the newly designed measuring device is further characterized by means of multi-region CFD simulations, replicating the real operating conditions of a WtE plant located in southern Italy. The numerical tool, once validated by on-field tests, is employed to: (i) determine the optimal temperature and flow rate of the cooling air; (ii) verify the maximum temperature reached by the materials; (iii) determine the time constant of the device; (iv) estimate the measuring errors.

Experimental and Numerical Convective Heat Transfer Performance Analysis of a Confined Round Jet with Triangular Tabs Under Crossflow Influence

Mehmet Saglam, Buğra Sarper, Soner Birinci, Orhan Aydın

Karadeniz Technical University
Tarsus Universitesi

Email: mehmetsaglam@ktu.edu.tr

In this study convective heat transfer performance of a confined round jet with/without nozzle tabs under channel crossflow influence is investigated experimentally and numerically for a wall mounted heated module. Numerical investigations are carried out for different jet to crossflow velocity ratios (Vr) and compared to experimental data. Four different Vr values (5,6,8, and 10) are tested for single target plate distance to jet diameter ratio (H/D=8). ANSYS Fluent software is used for numerical simulations while infrared thermography is utilized for experimental investigations. Surface temperature values of the wall mounted module are obtained and presented along with complex flow structures obtained with numerical simulations. Results show that crossflow has decisive effect on temperature distribution on the component. The triangular tabs inserted jet nozzle outlet has minimal effect on module surface temperature and jet deflection caused by crossflow.

Impact of One and Two Partial Cooling Ducts on the Temperature Rise in Transformer Windings

Goran Bulatovic, Peter Bokes

Slovenska technicka univerzita v Bratislave

Email: goran.bulatovic@stuba.sk

We apply a quasi-1D model of conductive heat transfer with convective boundary conditions [1] to obtain temperature distribution in transformer windings with two symmetrical partial cooling ducts. The results are compared to windings' 2D FEM simulation and industrial testing [2]. We also compare the FEM simulations between the windings with one partial cooling duct to the windings with two partial cooling ducts.

[1] P. Bokes, Average temperature of oil-filled transformer windings with partial cooling ducts, Journal of Electrical Engineering 72 (2021) 35–39. [2] provided by BEZ Transformatory, a.s. https://bez-transformers.com/

Numerical Analysis on the Energy Efficiency of Drying of Masonry Wall After Flooding Using Variable Drying Air Temperature Profile

Michał Wasik, Piotr Łapka

Warsaw University of Technology

Email: michal.wasik@pw.edu.pl

A numerical simulation using the in-house non-equilibrium heat and moisture model was performed to investigate the impact of variable temperature profiles on the thermo-injection method drying efficiency. A masonry wall initially highly saturated with moisture and without internal or external water sources (a model of a wall after flooding) was considered. A drying process lasting two weeks was simulated. Based on previous studies, three drying strategies with varying drying air temperatures, i.e., jump, stepwise, and periodic, and reference strategy with a constant temperature, were compared. The drying air temperature profile was changed from 20 to 60°C using three different heating intervals (i.e., 12h, 24h, and 48h) and with different strategies. In the jump strategy, the drying air temperature changed rapidly to 60°C in a one-time step after the initial period of low temperature of 20°C. In the stepwise ones, the temperature changed by 10°C in each heating interval until reaching 60°C, and in the periodic strategy, the temperature changed cyclicly from 20°C to 60°C or from 60°C to 20°C after each heating interval. Furthermore, the relative humidity of drying air corresponded to the three seasons in Poland (winter, summer, and spring), ranging between 70 and 90% at ambient conditions. It was found that the drying strategy with a variable temperature profile may decrease energy consumption compared to the reference case (i.e., constant drying air temperature).

Evaluation of the Effects of Various Fan Parameters on the Cooking Process in Commercial Cooking Ovens

Remzi Timur, Zafer Kahraman, Murat Haci, Hakan Soyhan

Yildiz Teknik Universitesi
Oztiryakiler Madeni Esya San. ve Tic. A.S
Sakarya Universitesi

Email: rtimur@oztiryakiler.com.tr

Commercial kitchen ovens are utilized in various capacities and models within commercial enterprises, operating on either electric or gas power. Numerous parameters affect the cooking process. Particularly, for effective cooking in an oven, it's crucial to evenly distribute airflow over the food. Correct airflow significantly impacts the cooking process. Considering the differences in processing requirements based on the characteristics and quantity of the food (such as temperature, humidity, fan speed, duration, etc.), the method of heat transfer should be primarily assessed. Non-uniform distribution of hot air during the cooking process leads to varied cooking qualities among trays within the oven. Therefore, in achieving effective cooking, factors like oven cavity, number of trays, fan geometry, fan rotation speed, direction, movement intervals, and the turbulent airflow generated by the fan in the cooking chamber need thorough examination. This study analyzed simulation data evaluating the effects of different fan geometries, fan rotation directions, and timing on heat transfer in a prototype electric commercial cooking oven using gastronomic GN-1/1 sized with 10 trays. Additionally, comparison of test and design validation data in the specially developed prototype cooking oven was conducted. Assessments were made with different fan movements (rotation direction, speed, and timing) and a fan protection plate in front of the oven to achieve more efficient heat distribution on the food. Depending on various fan parameters, this study contributed to reducing energy consumption in the prototype commercial cooking oven.

Thermal Performance of a Chaotic Heat Exchanger: An Experimental Study

Nouhaila El Hani, Tom Lacassagne, Souria Hamidouche, Loïc Le Bihan, André Bontemps, S. Amir Bahrani

Institut Mines-Telecom

Robert Bosch GmbH

Université Paris Diderot, Sorbonne Paris Cité, LIED, UMR 8236, CNRS, 75013 Paris, France

Email: nouhaila.elhani@imt-nord-europe.fr

Improving the efficiency of heat exchangers requires innovative approaches that reconcile performance and sustainability. Several intensification techniques are available to improve heat exchanger performance. The aim of these intensification techniques is to reduce the exchange surface area by increasing the heat exchange coefficient while maintaining a low pressure drop and, ideally, to reduce the cost of components and energy systems by making them more efficient. In order to increase the convective transfer coefficient, chaotic advection is known to be an interesting mechanism for enhancing heat transfer. The generation of chaotic advection depends on the geometric perturbation added to the main flow.This study examines the potential of a curved geometry employing chaotic advection as a passive technique for improving internal heat transfer. The Chaotic Heat Exchanger (CHE) studied was constructed using 18 bends at 90-degree angles, and by simply turning each bend at a 90-degree angle to the previous one. These successive changes in curvature plane orientation induce chaotic trajectories at low Reynolds numbers. With each change of orientation, the centrifugal force is reoriented and so the positions of the Dean cells, formed in the secondary flow and acting as internal agitators, are modified so that the cells are destroyed and rebuilt in a perpendicular plane.The goal of this work is to propose a chaotic heat exchanger geometry by focusing on heat transfer efficiency. A comparative analysis is carried out between the chaotic heat exchanger and a conventional Helical Heat Exchanger (HHE), using an experimental approach with water as the internal hot fluid. The Reynolds number examined ranges from 250 to 3500. It has been observed that CHE outperforms HHE in terms of heating capacity and internal Nusselt number, particularly in the intermediate Reynolds number range, with a spectacular increase between Re=600 and 1000, and a plateau in improvement at Re>1000.

Simple Model of Liquid Piston Compressor

Nejc Cerkovnik, Luka Čurović, Jurij Prezelj

University of Ljubljana

Email: nejc.cerkovnik@fs.uni-lj.si

The study proposes a new 0D thermodynamic model for the fast and simple evaluation of Liquid Piston compressors whose geometry and operation are determined by large number of parameters. With the proposed model, the evaluation of the parameters becomes easier, more dependencies are found and the understanding of the influences of parameter changes is deepened. Therefore, optimal designs for the mentioned machines can be created and tested experimentally. During the study, the model was validated with experimental and CFD data, and the results showed good agreement. It was found that evaluation criteria need to be defined for each set of geometric and operational parameters in order to compare the designs.

Session: Miscellaneous

Chair: Igor Vušanović

Making Temperature Fields in Fluid Flows Visible from Discrete Sensor Data

Michel Speetjens, Ruud Lensvelt, Henk Nijmeijer

Eindhoven University of Technology

Email: m.f.m.speetjens@tue.nl

Active control of scalar transport (e.g. temperature or chemical species) in engineering flows by the strategy proposed in Lensvelt et al. (Int. J. Thermal Sciences 180, 2022) can substantially improve the performance of such processes. This strategy relies on feedback of intermediate scalar fields to the controller to determine the control action. Key for its practical application is the fast and accurate estimation of such fields from discrete sensor data (e.g. thermocouples in case of heat transfer). This study develops a so-called “state observer” that enables such sensor-based field estimation for a representative problem: temperature field evolutions during heat transfer in the 2D Rotated Arc Mixer (RAM). The study adopts a so-called “Luenberger observer”, which estimates the temperature field from the deviation between predicted and actual temperature in the sensor positions via a model for the internal heat transfer. Crucial to this end are, first, a sufficiently accurate and fast model and, second, “strategic” sensor placement for full state observability and data quality. The observer model emanates from discretization of the governing heat equation and subsequent model reduction by truncation of the spectral decomposition of the system matrices. Sensors are placed to optimally monitor the dominant eigenmodes in this spectral decomposition. The performance of the observer is investigated by its ability to reconstruct known temperature fields from the corresponding values at the sensor locations using fields obtained from (i) numerical simulations and (ii) infrared thermography in a RAM laboratory set-up. Application to the numerical data demonstrates a fast and accurate field reconstruction for an appropriate observer tuning. Application to the experimental data demonstrates a good qualitative reconstruction of the spatial features of the temperature fields yet with notable quantitative deviations. Error analysis attributes this to both modelling limitations and experimental imperfections and this offers first leads to remedy this.

Periodic Laser Heating Technique for Thin Film Characterization

Erika Hahn, Sebastian Feulner, Vincent Linseis

Linseis Messgerate GmbH

Email: e.hahn@linseis.de

In recent years, the trend towards miniaturization of devices has become more and more important. As a result, there is an increasing interest in the studying thin films, particularly in relation to heat transport systems. The analysis of the thin films, mostly in µm range, is crucial as the behavior might differ from that of bulk material. Typical measurement techniques may not be suitable for precise measurement of thin films. For instance, the well-known Laser Flash method, LFA, needs to reduce the laser pulse to a few microseconds for an accurate measurement which is technically challenging. Therefore, we have introduced a commercial instrument using the measurement method called Periodic Laser Heating (PLH). The setup follows the classical laser flash, where one side of the sample is exposed to a laser and the rear surface is monitored by an infrared detector. The key difference is that the PLH setup operates in the frequency-domain rather than the time-domain, thus, a periodic modulated beam is used to induce a thermal disturbance. By measuring the phase shift and the change in amplitude of the emitted infrared signal by the sample and using the sample thickness, the thermal diffusivity is calculated. Advantageous is also, that there is no need to choose a specific model for evaluation. The non-contact technique is especially suited to determine the properties of thin polymeric films with thicknesses of a few tens to hundreds of micrometers. However, it can also be used for higher conductive and thicker samples up to several hundreds of micrometers. We verified the validity of this technique by comparing its findings with the values documented in literature across a broad range of material categories, thereby demonstrating its broad applicability. The unique combination of the PLH and LFA technique allows a further extend of the measurable samples.

Impact of Variation of the Cooling System Operating Strategy on Energy Efficiency and Waste Heat Quality: A Preliminary Investigation on a Hybrid-Cooled Data Centre

Lorenzo Testa, Philip Stuart, Cathal O'Donnell, Tim Persoons

The University of Dublin Trinity College

Email: testal@tcd.ie

The high reliance of modern society on digital services and the corresponding need for data centers require the implementation of effective measures to provide such services in a more sustainable way. This issue has grown in importance in recent years as more attention is paid to sustainable operating practices and the need for optimized energy systems to reduce our environmental footprint. In the context of data centers, this could be achieved by maintaining reliable working conditions of the cooling system while enabling the recovery and reuse of waste heat. In a data center, the cooling system can account for approximately 30% of the total energy consumption. Such cooling systems often work at more intense regimes than required to readily face any unexpected variation of the cooling demand which, if not promptly addressed, could lead to overheating of the servers and interruption of the service. This tendency for overcooling puts a higher strain on key cooling system components such as pumps and fans. It would also lead to the generation of low-quality waste heat, limiting its exploitability. This paper investigates the possible benefits on both energy efficiency and waste heat quality that would follow a reconfiguration of the cooling system operating strategy of a small-scale modern air-cooled data center located in Dublin, Ireland. The data center is installed within a university and supports the daily activities providing a variety of services, including HPC and online lecture streaming. It has 250 kW of installed power capacity, and the server room occupies an area of approximately 70 m^2. This assessment is divided in two main parts: the first is an analysis of the working conditions set by the manufacturer of the data center. In the second, a new operating strategy is proposed, and the resulting potential benefits are assessed.

Session: Thermodynamics

Chair: Igor Vušanović

Modelling of the Refrigerant Distribution in a Critically Charged Propane Heat Pump Cycle for Performance Evaluation

Jana Rogiers, Tanya Deblaere, Xander Van Heule, Bernd Ameel, Steven Lecompte, Michel De Paepe

Universiteit Gent

Daikin Europe NV

Email: jana.rogiers@ugent.be

The investigation of heat pumps employing natural refrigerants as working fluids has gained substantial momentum, driven by increasingly stringent regulations addressing the global warming potential (GWP) and ozone depletion potential (ODP) of refrigerants. Propane (R290) stands out as an attractive option due to its low GWP and zero ODP. However, a major drawback of propane lies in its flammability. To mitigate the safety concerns when using propane as working fluid, it will be necessary to limit the total refrigerant charge within the system, without compromising on the heat pump performance. This study endeavors to address this challenge by developing a model of the refrigerant charge distribution in a critically charged heat pump system. The modelled heat pump cycle thus merely consists of an evaporator, a compressor, a condenser, and an expansion valve. In the model, the charge distribution in the two heat exchangers is modelled in detail using a suitable void fraction correlation. For the compressor and expansion valve, a simplified model is employed, in which no refrigerant charge is calculated. Using an iterative procedure, the model can then be used to determine the coefficient of performance (COP), evaporator - and condenser pressure for a heat pump with an established total charge and heat capacity. The influence of different design and operation parameters, such as the subcooling, the superheat, the volume ratio of the evaporator and the condenser, etc., can be assessed.

Characterisation of Different Darken Sand Particles and Behaviour Under Fluidized and Irradiated Conditions

Leonel Mario Cerutti Cristaldo, Minerva Díaz Heras, José A. Almendros-Ibáñez, Jesús Canales Vázquez, Juan Carlos Pérez Flores

Universidad de Castilla-La Mancha
Instituto de Investigación de Energías Renovables, Univ. Castilla La Mancha

Email: leonelmario.cerutti@uclm.es

The use of solid particles in a fluidized bed with concentrated solar irradiation from the top is a promising technology for the next generation of concentrated solar power (CSP) plants. Sand is an inexpensive and abundant material easy to fluidize, but it has a low absorptivity, which is around 0.5 according to different previous works. This optical property is a key parameter for CSP applications with solid particles. This work presents a novel methodology to induce darkening of the sand surface by inducing solid state diffusion of Mn in SiO2, rendering a stable material resistant to abrasion upon the fluidization process. For this study, two different samples considering different SiO2:MnCO3 ratios (1:30 and 1:50) were analysed. The objective is to compare the two samples and optimise the concentration of $MnCO_3$ to get the desired darkening of the sand and high absorptivity. First, the main properties of the particles were analysed (particle size, morphology, colour and absorptivity). Second, the samples were tested in a lab-scale fluidized bed directly irradiated from the top with a beam-down 4kW Xe lamp. Both samples were tested under three different fluidized velocities: 1.5, 2.0, and 2.5 times the minimum fluidization velocity (Umf). In both cases, there is a significant increase in the maximum temperature reached during the process, with temperatures exceeding 260°C. This is clearly higher than the case of raw sand, which reaches 230°C under the same conditions. Furthermore, these values exceed the highest temperature reached by SiC in the same facility, which is 250°C, and it is considered as one of best tested materials for CSP applications.

Optical Characterization of a Solar Concentrating Dish System up to 2000 Suns

Clemens Suter, Saurabh Tembhurne, Isaac Holmes-Gentle, S H

ETH-Rat
Sohhytec SA, EPFL Innovation Park, 1015 Lausanne, Switzerland
EPFL - EPF Lausanne

Email: clemens.suter@epfl.ch

Solar dishes concentrate the quasi-collimated sun light towards a focal point. Typically, the optical characterization consists of experimental flux map measurements and Monte-Carlo ray-tracing (MCRT) simulations, which are fitted based on experimental data and serve for the prediction of the radiative flux distribution in receivers or reactors. However, the MCRT simulations usually rely on idealized mirror geometries (i.e. parabolic mirrors) and not the actual dish geometry given by its construction method, such as petals, facets or segments. Thus, the parameter fitting might yield unrealistic parameters and local radiative flux peaks ("hot spots"), which are often experimentally detected behind the focal plane, cannot be predicted. Here, we characterized the 7m diameter solar dish at EPFL comprising 27 petals with a nominal focal length of 3.8 m and a rim angle of 50.3°. The measured peak concentration was 1781 suns and the received integrated solar power was 20.0 kW over an 18 cm diameter spot. We proposed an advanced MCRT model considering the curvature of the dish, i.e. geometries mathematically described by exponents not equal, but close to 2. The advanced MCRT fitting approach was then applied for experimentally measured focal and off-focus planes. The simulated reflectivities were 55% and 86% for the standard and advanced MCRT model, respectively. Thus, the standard MCRT model only found an unrealistic low value whereas the advanced MCRT model's value was within 1% of the manufacturer's specification. Furthermore, the advanced MCRT model predicted an exponent of 1.94, which emphasizes that the best geometry to describe the solar dish deviates from a perfect paraboloid. We conclude that MCRT models should consider non-perfect parabolic geometries for solar dishes in order to obtain realistic values for the fitted parameters. Furthermore, we expect with this new approach a higher accuracy for the prediction of radiative flux distributions for off-focus planes.

Effects of Isochoric Freezing on Quality Characteristics of Raw Bovine Milk

Alan Maida

State of California

Email: alan_maida1@berkeley.edu

This study investigates the effects of isochoric freezing (IF) on the shelf-life and quality of raw bovine milk over a 5-week period. Isochoric freezing (IF) leverages confined aqueous thermodynamics to passively generate elevated pressures at mild sub-freezing temperatures, producing a technologically simple means of reducing metabolic processes while avoiding formation of ice in the stored food product. The process involves filling a rigid container with an aqueous solution and a food product, subjecting it to sub-freezing temperatures, and allowing the expansion of ice to passively pressurize the interior of the system. Per Le Chatelier's principle, the rising pressure hinders further freezing, inducing a pressurized two-phase liquid-ice equilibrium with a stable ice fraction at any given sub-freezing temperature. The IF treatment process entailed storing liquid raw milk in isochoric chambers in thermodynamic equilibrium at -5°C /77MPa and -10°C /96MPa. Several parameters were analyzed, including microbiology count, physicochemical properties, indigenous enzyme activity, protein content, volatile organic compounds profile, and lipid degradation. Microbiology count decreased significantly during storage for both IF treatment conditions but was more pronounced for the higher pressure (96MPa) treatment, leading to undetectable levels of microorganism after 5 weeks. IF treatment maintained stable pH, titratable acidity, viscosity, lipid oxidation, volatile profiles, total protein content, and lactoperoxidase activity throughout the storage period. Overall, the study demonstrates that isochoric freezing could significantly increase the shelf-life of milk by reducing microbiology activity, whilst maintaining its nutritional content. These results underscore the potential role of isochoric freezing as a valuable tool in eliminating pathogens while maintaining quality characteristics similar to raw milk over long storage periods.

Take Away Talk

Reflections on Some Achievements in Thermofluid Sciences Related to Slovenians and Slovenia

Božidar Šarler

University of Ljubljana, Faculty of Mechanical Engineering
Institute of Metals and Technology, Ljubljana, Slovenia

Email: bozidar.sarler@fs.uni-lj.si

Slovenians' population grew from about 1 million at the beginning of the 18th century to above 2 million today. The ethnic map of where the Slovenians lived in the 18th century has shrunk thrice and amounts today 20 271 square kilometres. The first preserved written text in the Slovenian language dates back to the tenth century. So, it is not surprising that the Slovenians did not contribute much to the overall treasure trove of science. The talk aims to illuminate the work of three exceptional Slovenians and a non-Slovenian living on Slovenian soil associated with thermofluid sciences. These are Janez Vajkard Valvasor (1641-1691), Jožef Stefan (1835-1893), Ernst Mach (1838-1916) and Zoran Rant (1904-1972). I have picked up Valvasor because he described the hydraulics of Cerknica intermittent lake, Stefan, because of his contributions to heat transfer, Mach because of his experiments with hypersonic flow, and Rant because of his contribution to thermodynamics. A large number of concepts and terms have roots in their work. The Stefan number, the Stefan problem, radiation law, Mach number, exergy and energy terms are examples of standard use nowadays. I have selected some interesting scientific and private elements from their lives with the hope of being interesting to the Eurotherm community and considering them as a take-away message from the 9th Plenary Eurotherm Conference, organised in Slovenia in June 2024.