15th International Conference on Computational Heat & Mass Transfer

(*ICCHMT'25*)

BOOK OF ABSTRACTS

Welcome Message from Local Organizing Committee

Dear esteemed researchers, industry specialists, and students from all over the world, we would like to welcome you to the *15th International Conference on Computational Heat and Mass Transfer* (ICCHMT'25) in the enchanting city of Antalya, Türkiye, where you can enjoy the breathtaking coast of the Mediterranean Sea to its fullest.

In this event where leading researchers and outstanding scientists working on top-notch areas of science and engineering, distinguished industry experts from diverse areas, and enthusiastic students around the world meet, participants will be able to share information and expertise on computational methods as well as build professional and academic networks for collaboration while discussing possible future directions to benefit the scientific world at large. A wide range of research subjects on computational methods vary from energy, thermal, and process engineering to energy efficiency, nanotechnology, biomedical applications, environmental and biological processes.

Stimulating and inspiring discussions with top experts in the field and building life-long connections with like-minded professionals will be simultaneously conducted while enjoying the luxurious facilities of the hotel and breathtaking crystal-clear Mediterranean waters.

The conference city of Antalya, known as the Turkish Riviera, is a world-class tourist destination with its silvery turquoise waters, golden sandy beaches, and historical sites that exhibit the roots of ancient human civilizations. Modern amenities you will enjoy will not distract you from indulging in this natural beauty and historical charm.

Join us in Antalya for a conference to remember that joins academic and professional excellence with luxury, history, and the Mediterranean. As the conference committee, we are looking forward to welcoming you to this phenomenal event where ideas join and benefit each other, collaborations develop, and innovations grow.

Prof. Dr. Barbaros Çetin (Bilkent University, Local Organizing Chair)

Prof. Abdulmajeed Mohamad (University of Calgary, Founding Chair)

Prof. Ali Cemal Benim (Dusseldorf Uni. Appl. Sci., European Chair)

Prof. Dr. Zafer Dursunkaya (Middle East Technical University, Local Organizing Co-Chair)

Asst. Prof. A. Alperen Günay (Bilkent University, Local Organizing Co-Chair)

Asst. Prof. Gökberk Kabacaoğlu (Durham University, Local Organizing Co-Chair)

Dr. Cem Gözükara (ASELSAN A.Ş., Local Organizing Co-Chair)

Objectives

The ICCHMT conference series, held periodically since 1999, focuses on heat and mass transfer processes. Given the importance of heat and mass transfer processes for a wide range of engineering applications, especially in energy/thermal/process engineering, the main objective of ICCHMT is to bring together specialists from all around the world to share their most novel findings and discuss possible future directions. With today's urgent need for the efficient use of energy resources, coupled with environmental concerns, the importance of the topic becomes even more apparent. Nonetheless, the conference also reaches the recently growing fields pertaining to heat and mass transfer, including nanotechnology, biomedical applications, and environmental as well as biological processes.

The conference addresses specifically the research, development, and application of computational methods, without, however, excluding experimental and theoretical approaches, especially as a means of validation and inspiration. The highly multifaceted field that can include multi-phase phenomena and chemical reactions is correspondingly demanding for computational methods. The latter can cover a wide range of scales, extending from the macroscopic level to the nano-level, using continuum or discrete mechanics, depending on the phenomenon and purpose considered. As various procedures such as Finite Volume Method (FVM), Finite Element Method (FEM), and Finite Difference Method (FDM) can be utilized to discretize the field equations of a continuum, particle-based methods utilizing different philosophies such as Smoothed Particle Hydrodynamics (SPH) or Lattice Boltzmann Method (LBM) or can also be employed, which can model continuum as inspired by discrete mechanics or molecular dynamics, and, thus, build a bridge to meso-scales, whereas truly Molecular Dynamics (MD) simulations can be performed at non-continuum scales.

The conference series provides a platform for scientists and engineers to meet regularly in a relaxed environment to discuss new ideas and developments on computational methods and their applications, as well as a good opportunity for young scientists and engineers to explore the art of computational methods and future perspectives while further broadening their horizons.

Conference Secretariat

Ms. Özge Ezici Çetin (Bilkent University)

Mr. Çağın Aydın (Bilkent University)

Ms. Duru Çınar (Bilkent University)

Mr. Kerem Dülger (Bilkent University)

Mr. Ö. Tarık Mumcu (Bilkent University)

Ms. Ayda Önsoy (Bilkent University)

Mr. Artun A. Öztaş (Bilkent University)

Ms. Selena Şahin (Bilkent University)

Mr. M. Yiğit Zaif (Bilkent University)

Partners

Türkiye '25
ICCHMT

Our Sponsors

iogengineering

computation
an Open Access Journal by MDPI

UK National Heat Transfer Committee

SPRINGER NATURE

TECHNICAL PROGRAM

DAY-1 (May 20, TUESDAY)

Time	S1-A	S1-B	S1-C	S1-D	S1-E
09:00-09:45	**PLENARY SESSION: Prof. Ali Beskok (MAIN HALL)** **From Atoms to Flow: Exploring Evaporation and Condensation in Nano-Channels** **Chair: Prof. Barbaros Çetin**				
09:45-10:00	**COFFEE BREAK**				
10:00-12:00	**S1-A: Thermal Management/Electronics Cooling** **Chair:** Prof. Damena Agnofer **MAIN HALL**	**S1-B: Numerical Heat Transfer - I** **Chair:** Prof. Andrey Kuznetsov **Room-B**	**S1-C: Energy Systems** **Chair:** Prof. Prof. Tassos G. Karayiannis **Room-C**	**S1-D: Multi-phase Modeling - I** **Chair:** Prof. Ali Cemal Benim **Room-D**	**S1-E: Multiphysics Modeling** **Chair:** Prof. Kerem Pekkan **Room-E**
	[074] Performance Analysis of Wavy Microchannels: A Comparative Study of Traditional and Manifold Microchannels for Electronic Cooling	[213] Numerical Simuation on Liquid Metals Flowing through Rough Parallel Plates with Uniform Heat Flux Heating	[154] Experimental Analysis and Dynamic Energy Model of a Gas Liquid Energy Storage (GLES) for Electric and Thermal Storage	[059] Pore-scale Study on Melting Characteristics of Phase Change Materials in Rectangular Porous Media Based on Thermal Resistance Analysis Methods	[032] Multiphysics Synergy Analysis of Light, CO2 Mass Transfer and Fluid Dynamics in Microalgae Photobioreactors
	[075] Enhancement of Phase-Change Efficiency by the Synchronized Reciprocating Rotation and Heaving Motion of the Thermal Storage Unit	[060] A Numerical Framework for Conjugate Heat Transfer Using the Immersed Boundary Method with a Compressible Solver	[023] Integrating Waste Heat Recovery System in Biomass Boiler: Experimental Insights and 1D Modelling	[049] Numerical Analysis of the Frost Formation Over Protruded Flat Surfaces	[024] Modelling of Electromagnetic Field Distribution in the Digital Twin of a Laboratory-scale Microwave Drier for Clay Roof Tiles
	[221] Microsystems Direct Cooling Using a Bi-phase Microfluidics Droplet	[073] Exploring the Synergistic Impact of V-Ribs with Cylindrical Vortex Generators on Solar Air Heater Performance: A CFD Approach	[069] Fluid-structure Interaction Analysis of a Small-scale Piezoelectric Wind Energy Harvester	[159] CFD Coupled With a Lognormal Model for Modeling an Evaporation-Condensation Type Aerosol Generator	[012] Multi-physics Analysis and Design of Lightweight Composite Enclosures for Enhanced Thermal Management in Electric Vehicle Battery Modules
	[087] Experimental Investigation of Open Circuit Voltage of a Li-ion Battery at Different Operating Temperatures	[096] ShapeFfactor Analysis for the Convective Radiative Flow of Tri-hybridized Nanofluid with Three Different Nanoparticles over a Rotating Cone	[011] Computational Model of a Stirling Heat Pump with Linear Actuators and a Regenerative Heat Exchanger	[119] Drag Force and Mass Transfer in Gravity-driven Bubbly Flows on Inclined Channels using the UCLS Method	[053] MHD Natural Bio-convective Flow of Radiative Jeffery Nanofluid through a Vertical Permeable Cone with Viscous Dissipation Effects
	[161] Development of a Novel Thermal Management System for Li-ion Battery using Microchannel	[101] Numerical Simulation of Marangoni-Benard Convection in a Planar Periodic Domain	[067] Eulerian-Eulerian Modelling of Discharge Process in Spouted Bed Solar Receivers	[082] Numerical Investigation of the Effect of Microscale Cavities on Nucleate Boiling	[063] Hybrid Numerical Simulation of MHD Mixed Convection in Nanofluid-Filled Cavities with Application to Electronics Cooling
	[066] Numerical Investigation of Copper Metal Foam Integration in Hybrid Battery Thermal Management System for Enhanced Energy Density and Optimized Temperature Control	[126] Development of Grid Model Requirements for Direct Numerical Simulation of Natural Convection in an Infinite Gap			[068] Hybrid FDM-LBM Method for Thermal Management of Electronic Components via MHD Natural Convection Using MWCNT-Fe2O3/H2O Hybrid Ferrofluid
12:00-13:30	**LUNCH**				

DAY-1 (May 20, TUESDAY)

13:30-14:10	**KEYNOTE SESSION: Prof. Andrey Kuznetsov (MAIN HALL)** **Macroscale Turbulent Vortex Impingement on Porous/Fluid Interfaces** **Chair: Prof. Barbaros Çetin**			**KEYNOTE SESSION: Prof. Ayşe Gül Güngör (Room-B)** **Vortex Dynamics and Flame Interaction in Bluff Body Premixed Combustion** **Chair: Prof. Zafer Dursunkaya**	
14:20-16:00	**S2-A: Heat Pipes** **Chair:** Prof. Damena Agnofer **MAIN HALL**	**S2-B: Reacting Systems - I** **Chair:** Prof. Ayşe Gül Güngör **Room-B**	**S2-C: Numerical Heat Transfer - II** Chair: Prof. Ali Beskok **Room-C**	**S2-D: Multi-phase Modeling - II** **Chair:** Prof. Abdulmajeed Mohamad **Room-D**	**S2-E: Turbulent Flow** **Chair:** Prof. Marcelo de Lemos **Room-E**
	[019] Innovative Groove Designs for Enhanced Flat-Grooved Heat Pipe Efficiency	[132] Numerical Investigation of Hydrogen Addition on Flame and NOx Emissions in Methane Combustion for Industrial Kilns Using a New Reduced Mechanism	[144] Numerical Study of an Industrial Scale Continuous Container Glass Annealing Furnace	[124] Comparison of the UCLS and Coupled VOF–LS Methods for Hydrodynamics and Mass Transfer in Bubbles	[108] Implementation and Validation of an Improved k-ε Turbulence Model Based on Reynolds Number
	[044] Pseudo-3D Modeling of Grooved Heat Pipes	[091] Numerical Study of Combustion Instabilities in a Single Injector Combustor	[173] Natural Convection Within an Enclosure Fitted With Blocks	[169] Optimization of Closure Model Coefficients with Bubble-Induced Turbulence for Enhanced Two-Phase Flow Predictions	[121] Direct Numerical Simulation of Turbulent Flow Regime in a Dense Triangular Rod Bundle Cell at Re = 14,200
	[002] Thermal Management of a Distributed Heat Load Using Bent Aluminum Axially Grooved Heat Pipes	[043] 3D CFD Modeling of MHSI and Fire Propagation in an Aircraft Engine Bay	[007] Comparison of Isogeometric and Lagrangian Elements with Boundary Element Method for Orthotropic Heat Conduction Problem	[022] Experimental and Numerical Investigation of Heat Transfer Performance in a Condensing Heat Exchanger	[171] Progress in Turbulent Flow, Heat, and Mass Transfer Modeling in Porous and Hybrid Media
	[092] Effect of Groove-Fin Width Ratios on the Thermal Performance of Grooved Heat Pipes	[056] Large Eddy Simulation of Turbulent Non-reacting Flow Inside a Swirl-stabilized Combustor via Lattice-Boltzmann Approach	[220] Identification of Transient Fluid Temperature Using Thermometer Readings	[080] Growth Mechanism of Boiling Bubble in Microscale Within the Interplay of Ultrasonic and Thermal Field	[130] RANS Simulations of Gas Flow Within an Industrial Kiln for Sanitary Ware Manufacture
	[076] Assessment of Sintered Wick Heat Pipe Performance	[115] A Novel Approach to C2C Variation Analysis in Heavy-Duty Engines: Multi-Cylinder Combustion Modelling with Detailed Chemistry	[172] Discrete Green's Function Method for Laplace Equation with Nonlinear Boundary Conditions	[094] Particle Dispersion and Deposition in Evaporating Sessile Droplets	[071] Numerical Study on the Effect of Loading Density on the Efficiency of Reverse-Flow Cyclone Separators
16:00-16:20	**COFFEE BREAK**				

DAY-1 (May 20, TUESDAY)

16:20-17:00	**KEYNOTE SESSION: Prof. Damena Agnofer (MAIN HALL)** **Thermal Management of Electronics From Device Level to Data Centers** **Chair: Prof. Barbaros Çetin**		**KEYNOTE SESSION: Prof. Marcelo J.S. de Lemos (Room-B)** **Advances in Modeling and Simulation of Turbulent Flow, Heat & Mass Transfer in Heterogeneous Media** **Chair: Prof. Zafer Dursunkaya**		
17:00-18:40	**S3-A: Heat Exchangers** **Chair:** Prof. Tassos Karayiannis **MAIN HALL**	**S3-B: Aerodynamics/ External Flow - I** **Chair:** Prof. Matteo Bernardini **Room-B**	**S3-C: Numerical Heat Transfer - III** **Chair:** Prof. Marcelo de Lemos **Room-C**	**S3-D: Diverse Topics in Heat Tranfer - I** **Chair:** Prof. Quiwang Wang **Room-D**	**S3-E: Indoor/Outdoor Flows** **Chair:** Prof. Ali Cemal Benim **Room-E**
	[129] Transient Modeling of Heat Exchangers Using a Steady-State Approach	[128] Experimental Investigation of the Effect of Small Off-Surface Vortex Generator on the Aerodynamic Performance of NACA0012 at Low Reynolds Number	[135] LES Modeling of Free Convection Heat Transfer from Horizontal Cylinder	[30] CFD Modelling of Laser Cutting Process of UHS Steels, Experimental Validation and Optimum Cutting Parameters Selection	[141] Derivation of a Roughness Model for Urban areas by means of detailed CFD simulation
	[138] Performance Analysis of Fixed-Bed and Gyroid-Based Regenerative Heat Exchangers	[201] Investigating the Impact of Roughness Element Distributions on Shear Flow Dynamics in a Backward-Facing Step Channel	[142] Heat transfer coefficient between spherical particles in low-conducting fluid	[21] Investigation of thermal-induced distortions in LBPF process by utilizing finite element simulations	[72] Open Space Indoor Air Quality and Comfort: Ventilation Versus Buoyancy Strategies
	[170] Performance Analysis of Plate Heat Exchangers under Dynamic Environmental and Freezing Conditions	[42] Vortex Gust Encountered by a Flat Plate at 45˚ Wing Sweep	[175] Forced Convection of a Viscoplastic Fluid in a Pipe: Pressure Drop	[181] Computational Analysis of Heat and Material Flow During Chemical Recycling of Composites for Wind Turbine Blades	[105] Numerical simulation of wind fence effect on coal dust emission
	[150] CFD and Energy Analysis of Photovoltaic-Supported Electrically Driven Heat Exchangers	[123] Numerical Investigation of Heating Effect in 3D Shock-Wave/Boundary Layer Interactions Near Protrusions in Hypersonic Flow	[088] Numerical Investigation of Heat Transfer Characteristics in Liquid-Cooled Heat Sinks for SiC MOSFET Power Inverters	[84] Assessment of Thermophysical Nature and Process Utility Limits of Nanofluids through Non-Dimensional Parameters	[099] Numerical Investigation of Thermal Comfort for Passengers in a Helicopter Cabin
			[048] Experimental-numerical Method for Determining Heat Transfer Correlations in the plate-and-frame Heat Exchanger	[122] Assessment of MCw Plasma Gasification via Instantaneous Response of Thermochemical Decomposition versus Post Processed Syngas Chromatography	

DAY-2 (May 21, WEDNESDAY)

09:00-09:45	**PLENARY SESSION: Prof. Tassos G. Karayiannis (MAIN HALL)** **Aspects and Challenges in Flow Boiling in Small to Micro-Scale Heat Exchangers**				
09:45-10:00	**COFFEE BREAK**				
10:00-12:00	**S4-A: Aerodynamics/ External Flow - II** **Chair:** Prof. Matteo Bernardini **MAIN HALL**	**S4-B: Microfluidics & Biological Applications-I** **Chair:** Prof. Ali Koşar **Room-B**	**S4-C: Heat Transfer Enhancement** Chair: Prof. Abdulmajeed Mohamad **Room-C**	**S4-D: Machine Learning-based Modeling - I** **Chair:** Prof. Ali Cemal Benim **Room-D**	**S4-E: Diverse Topics in Heat Tranfer - II** **Chair:** Prof. Zafer Dursunkaya **Room-E**
	[157] Direct Numerical Simulation of a Hypersonic Transitional Boundary Layer in Chemical Non-equilibrium: Effect of Wall State	[211] Responsive Virtual Wall Liquid Crystal Microfluidics	[070] Investigation of the Effectiveness of Using Micro-lattice Structured Meta-material for Enhancing Heat Transfer	[038] Precursor Detection for Flashback Events in Hydrogen Combustor via Integrated Machine Learning and Nonlinear Analysis	[183] Preliminary macroscopic non-equilibrium model for heat, air, and moisture transfer in bio-based building materials
	[209] Implementation of Numerical Schemes for the Computation of Incompressible Flows in OpenFOAM	[015] Programmable 3D Microfluidic Bio-Reaction Reservoirs Integrated with a Portable Pressure Pump for Miniaturization of Bioassays: Point-of-Care Detection of Monkeypox via LAMP-on-a-Chip	[008] Experimental Investigation and Evaluation of Heat Transfer Enhancement Effectiveness of Aluminum Foams for the Liquid-Cooled Avionic Electronic Units	[061] Prediction of Thermal Parameters of Individual Tube Rows in Finned Heat Exchangers using Artificial Neural Networks	[136] Discrete element modeling of heat transfer in active zone of nuclear reactor HTR-PM with advanced radiative model and nonuniform burnout of TRISO pebble fuel
	[058] Investigation of Aerodynamic and Structural Features Wind Escape Floors in Super-Tall Super-Slender Buildings	[202] Enhanced Design and Performance Optimization of Membranelles Micro Flow Battery for Self-Powered Lab-on-a-Chip System	[013] Numerical Investigation of Thermal Conductivity Enhancement in Phase Change Materials using additively Manufactured Lattice Structures	[098] Physics-Informed Neural Networks for Heat Transfer and Fluid Flow problems	[133] Study of the Thermophysical Properties of a Multilayered Impacted Cork-based Material by Infrared Thermography
	[165] Numerical Investigation of Novel Derivatives of Owl-Inspired Airfoils for Low Reynolds Number Applications	[131] Microstructure-Level Investigation of Nanoparticle Transport in Collagen Hydrogels for Advancing Nanomedicine Design and Delivery Strategies	[009] Investigation of Convective Heat Transfer Enhancement on a Cold Plate using Serpentine Channel with Friction Stir Welding	[199] Application of Artificial Intelligence Model for Extended Jet Impingement Cooling on Wavy Target Surface	[194] Computational Thermography for Injury Detection and Monitoring in Rugby Players
	[166] 2D Simulations on a Flat Plate to Study the Effect of Porosity on Skin Friction Drag Reduction	[020] Mathematical Modeling of Momentum and Mass Transport in Liver-on-a-Chip Systems	[195] Thermal and Hydrodynamic Evaluation of Microchannel Heat Sinks with Inline and Staggered Pin Fins: Enhancing Electronic Cooling	[207] Machine-learning-assisted Optimal Airfoil Design at High Mach Numbers	[184] Sensitivity analysis of micro-scale based method for predicting the thermal conductivity tensor of heterogeneous bio-based building materials
		[095] Experimental Analysis for Detection of Microplastic Waste by Using a Novel Microfluidic System with an Integrated Object Tracking Algorithm		[077] Dimensionless Approach to Modeling and Predicting Coating Thickness in Continuous Galvanizing Lines Using CFD and Neural Networks	[225] Modeling Localized Heating Induced Size Effects in Semiconductor Devices
12:00-13:30	**LUNCH**				

DAY-2 (May 21, WEDNESDAY)

Time					
13:30-14:10	**KEYNOTE SESSION: Prof. Matteo Bernardini (MAIN HALL)** **High-Speed Turbulence: Unraveling Boundary Layer Dynamics with HPC and Direct Numerical Simulations** **Chair: Prof. Barbaros Çetin**			**KEYNOTE SESSION: Prof. Kerem Pekkan (Room-B)** **Blood Flow and Mass Transport Across Scales: Cardiovascular System Integration and Fiber-Level Investigations of Blood Oxygenators** **Chair: Prof. Zafer Dursunkaya**	
14:20-16:00	**S5-A: Reacting Systems - II** **Chair:** Prof. Matteo Bernardini **MAIN HALL**	**S5-B: Multi-phase Modeling - III** **Chair:** Prof. Zafer Dursunkaya **Room-E**	**S5-C: Thermal Management/Electronics Cooling** Chair: Prof. Damena Agnofer **Room-C**	**S5-D: Numerical Heat Transfer - III** **Chair:** Prof. Quiwang Wang **Room-D**	**S5-E: Diverse Topic in Heat Transfer - III** **Chair:** Prof. Abdulmajeed Mohamad **Room-E**
	[062] Numerical Investigation of Fuel Injection Port Design on Partially Premixed Methane-Air Combustor	[050] Numerical Analysis of Frost Formation on Finned Tube Heat Exchangers: Effect of the Humidity Level on Heat Transfer	[163] Novel Heat Carrier Nano-Fluidics (HCNFs); Structural Design and Thermal Analysis	[189] Computational study of Laminar Free Convection Heat Transfer inside a Vertical Convergent Channel Heated Isothermally	[026] Reliability Assessment of the Fuel Rail Assembly under Thermo-mechanical Stresses in the Engine Compartment of the Passenger Cars
	[055] Numerical Investigation of Flame Dynamics and Mixing Characteristics of a Partially Premixed Swirl-Stabilized Combustor	[198] Condensation Heat Transfer from Surfaces Modified by Coating of Microscale Particles using Thermal Spray Coating	[180] Thermal Management of High Bright Led Display System by Forced Convection and Tailored Cooling System Design with Analytical and Numerical Approaches	[176} Determination of the Local Nusselt Number in the Thermal Entrance Region of Flow between Two Parallel Plates	[054] A computer system for online determination of thermal and pressure stresses and remnant lifetime of pressure components
	[116] Assessing the Impact of Hydrogen Reaction Mechanisms on Combustion Behavior in Heavy-Duty LPDI Engines	[204] Numerical Investigation of Dropwise Condensation on Biphilic Surfaces	[188] Numerical and Experimental Investigation of Twister Type Additively Manufactured Flow Mixer Fins for the Liquid-Cooled Avionics Electronic Units	[139] Heat Transfer in Falling Films Over Inclined Walls: A Study of Smooth vs. Wavy Film Surfaces	[041] Numerical Examination of the Flow Channel Expansion/contraction on the Performance of Polymer Electrolyte Membrane Fuel Cell
	[039] Computational Analysis of Non-Reacting Flow in a Non-Premixed Burner Featuring a Plasma-Enhanced Bluff-Body Swirler	[210] Investigation of Contact Melting in Molten Salt Phase Change Units with Proposed Euler-Lagrange Iteration	[203] Improved Thermal Management of Electrical Vehicles Using Internal Magnetic Field of the Electric Motors	[146] Investigation on the Heat transfer Process of Silica-gel Particles inside a Cylindrical Reactor	[197] On the Polaritonic Figures-of-Merit of Ionic Crystals for Subwavelength Optics
		[064] Numerical Approach for Entropy Generation and Exergy Destruction of Isobutane Condensation Flow in Microchannels	[216] Cladded Porous Material Integrated Pin Fin Heat Sink Performance Evaluation	[081] Analysis of Pellet Defects through Numerical Simulations of flow and Heat Transfer in Underwater Pelletizers	[085] Interferometry Guided Investigation of a Thin Film
16:00-16:20	**COFFEE BREAK**				
16:20-18:00	**S6-A: Diverse Topics in Fluid Flow** **Chair:** Prof. Ayşe Gül Güngör **MAIN HALL**	**S6-B: Multi-phase Modeling - III** **Chair:** Prof. Qiang Liao **Room-B**	**S6-C: Machine Learning-based Modeling** **Chair:** Assoc. Prof. Ali Karakuş **Room-C**	**S6-D: Molecular Simulations** **Chair:** Prof. Ali Beskok **Room-D**	**S6-E: Geophysical System Modeling** **Chair:** Prof. Ali Cemal Benim **Room-E**
	[160] Effects of Rarefaction on Thermochemical Non-Equilibrium via Open-Source Software	[031] A Non-homogeneous 2D Model for Frost Growth on a Plate Fin Surface	[177] Investigation of Characteristics of Different ANN Concepts in Flow Prediction	[149] Towards understanding ion-specific adsorption and interfacial thermodynamics at water-mica interfaces	[118] Preliminary Findings from the Thermohydraulic Modeling of the Bozköy EGS Field (Aksaray), Türkiye
	[196] CFD Analysis of Propane Leak Dispersion and Ventilation Optimization in Refrigeration System	[152] A New Two-fluid Numerical Method for Simulating Mass Transfer in Immiscible Two-phase Flows: Validation and Comparison with Single-fluid Results	[179] Prediction of Heat Transfer Characteristics of Impinging Jets Utilizing ANNs Based on CFD Simulations	[57] Temperature-Dependent Characteristics of Interfacial Thermal Resistance Between Liquid Metal Gallium and Nanofillers: A Molecular Dynamics Simulation Study	[192] Numerical Simulation of a Rapid-Abandonment Oil Well
	[222] Simulation-Based Analysis of Oil Flow Behavior in Hydrostatic Bearings for Vertical Francis Turbines	[168] Efficient Eulerian Interfacial Area Transport for Modeling Droplet Coalescence and Breakup in Complex Flow Systems	[215] A Machine Learning Framework for Hemodynamic Analysis of Stenosed Arteries	[214] Molecular Dynamics Simulation of Epoxy-based Polymer Coatings: Microstructure and Water Transport Mechanism at Metal Interfaces	[162] Optimizing Deep Well Systems: Thermal Performance Analysis of Co-Axial Heat Exchangers
	[110] Application of Galerkin Reduced-Order Modeling Based on Proper Orthogonal Decomposition for Viscous Burgers' Equation	[065] An Immersed Boundary Approach for Encountered Geometries in Multiphase Flows with Pseudopotential Lattice Boltzmann Model	[106] Investigation of Single Droplet Dynamics in a Microchannel Using Physics-Informed Neural Networks	[079] A First-Principles Study on the Adsorption Mechanism of Water Molecules on the SrBr2 (100) Surface	[120] Probabilistic Estimation of the Enhanced Geothermal Systems (EGS) Potential of the Bozköy Field (Aksaray, central Turkey) using Monte Carlo Simulations
	[017] Development of Newton Raphson Based 1D Flow Network Solver				[178] Thermal-Hydrologic-Mechanic Effects on Heat Transfer Processes in Enhanced/Engineered Geothermal Systems
19:30-21:30	**GALA RECEPTION**				

DAY-3 (May 22, THURSDAY)

09:00-09:45	**PLENARY SESSION: Prof. Qiang Liao (MAIN HALL)** **Impact Dynamics and Solidification Behavior of Droplet Impact upon Subcooled Wall in an Electric Field**				
09:45-10:00	**COFFEE BREAK**				
10:00-12:20	**S7-A: Microfluidics & Biological Applications-II** **Chair:** Prof. Kerem Pekkan **MAIN HALL**	**S7-B: Rotating Machinery** **Chair:** Prof. Ali Cemal Benim **Room-B**	**S7-C: Energy Systems** **Chair:** Prof. Jan Taler **Room-C**	**S7-D: Aerodynamics/ External Flow - III** **Chair:** Prof. Ayşe Gül Güngör **Room-D**	**Room-E**
	[218] Wearable Platforms for Continuous Ultrasonic Imaging and Biomarker Monitoring	[140] Aerodynamic Analysis of Wind Turbine Airfoils with a Nested Krylov Subspace Solver Implementation in SU2	[158] Thermoelectric Cooler Design without Using Cold And Hot Face Temperatures	[151] Impact of Shock Boundary Layer Interaction on Swirl Characteristics in Supersonic Intakes with Bleed System [86] CFD Analysis of Aerospike Length and Blowing Effects on Aerodynamic Heating Reduction in High-Speed Flow	
	[036] ML Automated Microfluidic Circuit Design	[212] An Analysis of the Airflow Patterns of an Electrohydrodynamic Fan	[191] Numerical Analysis of Solar Volumetric Absorbers Using a Two-Energy Equation Model		
	[224] Effects of Kinematic Hardening of Mucus Polymers on the Airway Closure Phenomenon	[217] Computational Design of a High-Pressure Compressor and its Experimental Validation	[200] HVACs optimal scheduling for Renewable Energy Communities using integrated solar-powered heat pump and thermal energy storage	[029] HEMLAB Algorithm Applied to the ARD Capsule	
	[219] Flowing Liquid Crystal-Aqueous Interfaces that Respond to Lipid Adsorption	[107] Numerical Investigation of the Influence of Structural Parameters on the Efficiency of Vertical Axis Tidal Turbines	[010] Waste Heat Recovery from the Exhaust Gas of a Turbo-Diesel Tractor Engine Using Organic Rankine Cycle	[147] Edney Shock Interactions in High Enthalpy Rarefied Gas Flows	
	[025] Modeling of Microfluidic MEMS-Based Capacitive Pressure Measurement	[104] Numerical Investigation of the Performance of Centrifugal Pumps with Bionic Surface Structures	[148] Optimisation of Thermal Energy Storage Systems for Industrial Heating Applications	[145] Finite Element Analysis of Flow and Heat Transfer of an Incompressible Non-Newtonian Fluid in a Porous Cavity	
	[153] An Investigation to Enhance the Mixing Efficiency of SAR Micromixer with Obstacles	[167] Mixing Analysis in a Stirred Tank Equipped with Innovative Impeller			
	[156] A Coupled Model for Drug Release from a Non-swellable Microsphere to a Surrounding Tissue	[185] Computational Design of a High-Pressure Compressor and its Experimental Validation			

ABSTRACTS

Thermal Management of a Distributed Heat Load Using Bent Aluminum Axially Grooved Heat Pipes

Murat Parlak[1],Alper Apak[2], Ergun ÖRS[1]

[1]ASELSAN Inc. 06830 Gölbaşı, Ankara, Türkiye

[2]APAK Industry Inc. Eskişehir, Türkiye

parlak@aselsan, agunay@bilkent.edu.tr, ergunors@aselsan.com.tr

In this study, aluminum axially grooved heat pipes were used to cool a distributed heat source which is critical due to the confined space. The study includes design, analysis, production, testing, and verification of the system. This study was conducted in response to a real need. Due to the lack of space, it was decided to transfer the heat from the source to a large area and cool that area by convection. To increase the condenser area and prevent the 7 heat pipes from shadowing each other, it was necessary to bend the heat pipes in the adiabatic region to distribute the heat equally within the fins by giving a little twist in the adiabatic region of the heat pipes. In the CFD studies, the heat pipes are modeled as a high thermal conductive material, and the condenser, where fins are press fitted to the heat pipes, is cooled with a suitable fan to keep the volume as small as possible. In Türkiye, heat pipes are manufactured in the available facilities using acetone as the filling liquid. Various angles were used in the experimental study to observe the effect of gravity on performance. In the verification studies, the desired thermal results were achieved. Thus, a 360W heat load at 50°C outdoor temperature was cooled by a condenser to 70°C evaporator temperature.

Comparison of Isogeometric and Lagrangian Elements with Boundary Element Method for Orthotropic Heat Conduction Problems

Can Önol, Artun Alp Öztaş, Barbaros Cetin

Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

can.onol@ug.bilkent.edu.tr, alp.oztas@ug.bilkent.edu.tr,
barbaros.cetin@bilkent.edu.tr

Boundary Element Method (BEM) is a numerical method for solving partial differential equations with reduced dimensionality, from volumes (3D problems) to surfaces or from surfaces (2D problems) to contours through the discretization of the boundary. BEM is particularly well-suited for achieving high-accuracy approximations in linear problems, such as the Laplace equation, using a semi-analytic solution scheme for fundamental solutions of the problem. Using isogeometric analysis with the boundary element method results in an exact representation of geometry and field variables, resulting in a highly accurate result compared to conventional BEM. In this study, the main aim is to solve and compare a two-dimensional orthotropic heat conduction problem using the boundary element method with isogeometric and Lagrangian elements.

Session: S4-C

Paper ID: 008

Investigation and Evaluation of Heat Transfer Enhancement Effectiveness of Aluminum Foams for the Liquid-Cooled Avionic Electronic Units

Mustafa Ocak, Fahreddin Susar, Mümin Türkyılmaz

ASELSAN A.Ş. 06750 Akyurt, Ankara Türkiye

mocak@aselsan.com.tr, mturkyilmaz@aselsan.com

The increasing demand for higher processing capacity in avionic electronic units (AEU) significantly boosts the performance; however, it comes with thermal management problems. In this way, the increase of heat loads for high-performance AEUs inevitably makes use of the liquid-cooled chassis instead of the conventional air-cooled ones. Liquid-cooled heat exchanger fin structures have been well studied in the past, especially longitudinal vortex generator type fins have been pointed out for heat transfer enhancement capabilities in rectangular micro and macro channel flows. On the other hand, some other researchers have demonstrated that aluminum foam structures can exhibit effective cooling performance in liquid cooling applications by their porous and random lattice structure. Porosity of foam structures provides very high contact area with fluids compared to conventional fin structures. Regarding that, heat transfer enhancement effectiveness of aluminum foams at 5 PPI, 10 PPI and 20PPI pore densities are experimentally investigated under 0.25 lpm to 5 lpm volumetric flow rates. Results are compared with empty channel and a vortex generator type conventional liquid cooling fin structure. Pressure drop values, wall temperatures, fluid outlet temperature standard deviation values are employed to illustrate effectiveness of the aluminum fins. It is observed that aluminum foams enhance heat transfer by decreasing the wall temperatures (compared to empty channel) at least 24.8% up to 36,8% for 20 PPI, at least 23.4% up to 36.1% for 10 PPI, at least 24.8% up to 38.2% for 5 PPI, while the vortex generator fins only decrease wall temperatures at least 15% up to 30,4%. Inevitably both the aluminum foams and vortex generator fins cause pressure drop increase especially at lower volumetric flow rates relative to the empty channel. 5 PPI pore density foam seems to be the best alternative compared to other foams and vortex generator fins at 3 lpm volumetric flow rate if ~50% pressure increase can be tolerated for the liquid pumping system.

Investigation of Convective Heat Transfer Enhancement on a Cold Plate using Serpentine Channel with Friction Stir Welding

Mehmet Yiğit Zaif[1], Mehmet Yener[2], Barbaros Çetin[1]

[1]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[2]HARP Savunma Sistemleri A.Ş. 06800 Ankara, Türkiye

yigit.zaif@ug.bilkent.edu.tr, mehmet.yener@harp.com.tr, barbaros.cetin@bilkent.edu.tr

Enhancing heat transfer through elastic turbulence offers significant advantages at the microscale, particularly for advanced electronic cooling applications. While recent studies have primarily focused on the global convective heat transfer in curvilinear channels, the inherently chaotic nature of streamline flow in elastic turbulence yields complex and spatially varied local heat transfer characteristics. In this study, we propose a novel cold plate design tailored for the thermal management of 6U VPX cards, employing the Liquid Flow Through (LFT) method. A 3D model of the cold plate was developed and analyzed using Simcenter FLOEFD to simulate both fluid flow and thermal performance. The cold plate features a serpentine microchannel geometry optimized for promoting elastic turbulence, thereby improving heat dissipation efficiency. Furthermore, the design is developed with Friction Stir Welding (FSW) constraints in mind, ensuring structural integrity and manufacturability when bonding the channel to the base plate. The integration of elastic turbulence, LFT cooling, and FSW-driven manufacturability presents a robust solution for next-generation electronic cooling systems.

Session: S7-C

Paper ID: 010

Waste Heat Recovery from the Exhaust Gas of a Turbo-Diesel Tractor Engine Using Organic Rankine Cycle

[1]Mehmet Yiğit Zaif[1], A. Alperen Günay[1], Salih Bilgiç[2], Ufuk Saral[3], Barbaros Çetin[1]

[1]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[2]SimOfis 06530 Ankara, Türkiye

[3]Türk Traktör A.Ş. 06560 Ankara Türkiye

yigit.zaif@ug.bilkent.edu.tr, agunay@bilkent.edu.tr

sbilgic@simofis.com.tr, barbaros.cetin@bilkent.edu.tr

This study deals with the computational modeling of an organic Rankine cycle integrated into a 3.6L Turbocharged Diesel engine using different working fluids on the SimCenter AMESIM software. The plant (engine) will be modeled according to the actual parameters. The model will have accurate transient road data to measure the real-time simulation of the engine coupled with the ORC. The efficiency of the engine is expected to increase with the ORC. The expander type of the ORC is determined per the power output type of the ORC. The computational model determines all of the plant's design parameters.

Computational Model of a Stirling Heat Pump with Linear Actuators and a Regenerative Heat Exchanger

Ömer Tarık Mumcu[1], Yunus Selçuk[1], Ali Tunahan Tanrıöven[1]

Mehmet Yiğit Zaif[1], Cihan Turgut[2], Barbaros Çetin[1]

[1]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[2]ASELSAN A.Ş. 06830 Gölbaşı, Ankara, Türkiye

tarik.mumcu@ug.bilkent.edu.tr, yunus.selcuk@ug.bilkent.edu.tr, tunahan.tanrioven@ug.bilkent.edu.tr, yigit.zaif@ug.bilkent.edu.tr, barbaros.cetin@bilkent.edu.tr

This study will examine a heating system based on a double piston alpha type Stirling heat pump with linear actuators and a regenerative heat exchanger. In such a system, linear actuators control the Stirling cycle via compression and expansion, which is expected to increase the COP of the system. The energy transformations and transmission mechanisms describe the configuration and operating principles. The prototype's design parameters were determined using SimCenter AMESIM, a 1D modeling software. The prototype's performance is analyzed using sinusoidal/linear piston displacement and with or without a regenerator. Actions are taken to improve the system's performance.

Multi-physics Analysis and Design of Lightweight Composite Enclosures for Enhanced Thermal Management in Electric Vehicle Battery Modules

Thamasha Samarasinghe[1], Mihalis Kazilas[1], Stuart Lewis[2]

[1]Brunel University London Kingston Lane, Uxbridge, Middlesex UB8 3PH, UK

[2]TWI Headquarters, Granta Park Great Abington Cambridge CB21 6AL, United Kingdom

1933914@alumni.brunel.ac.uk, mihalis.kazilas@brunel.ac.uk, stuart.lewis@twi.co.uk

The rapid expansion of electric vehicles (EVs) demands advanced thermal management systems for lithium-ion (Li-ion) batteries to ensure safety, performance, and lifespan. This study introduces a novel composite battery enclosure with embedded copper thermal bridges, designed to enhance passive thermal regulation. Using three-dimensional computational fluid dynamics (CFD) simulations and multi-physics modeling in COMSOL, the research evaluates the enclosure's thermal dissipation, examining heat and mass transfer across varied cell configurations and inter-cell distances. Thermodynamic principles guide the design, optimising the heat transfer pathways to lower surface temperatures by up to 16.62% compared to traditional enclosures. The model integrates analytical and finite element methods to capture complex heat generation and dissipation dynamics within battery modules, validated experimentally through a custom manufactured rig. The composite enclosure, manufactured using liquid composite molding (LCM) with strategically embedded copper pins, balances structural integrity and thermal conductivity. This setup creates thermal bridges that efficiently channel heat away from critical areas, leveraging copper's high conductivity and the lightweight benefits of fiber reinforced polymer matrices. Experimental tests using infrared thermography confirm the model's predictive accuracy, showing a uniform temperature distribution and effective passive cooling without active energy input. This study advances the application of composite materials in EV battery casings, offering a sustainable and high-performance alternative to metal enclosures. The findings underscore the enclosure's potential in addressing the stringent thermal management requirements of Li-ion batteries in next generation EVs.

Session: S4-C

Paper ID: 013

A Numerical Investigation of Thermal Conductivity Enhancement in Phase-Change Materials Using Additively Manufactured Lattice Structures

Ahmet Koyuncu, Orkun Doğu, Atakan Kabukçu

ASELSAN A.Ş. 06830 Ankara, Türkiye

odogu@aselsan.com, akabukcu@aselsan.com

With advancements in technology, it has become possible to monitor various variables in our environment with wireless sensors easily. One of the downsides of these sensors is that they require a battery to operate, and you need to replace the battery when it depletes. Depending on the situation, replacing the sensors' batteries is not feasible. Thermoelectric generators (TEGs) can generate electricity from waste heat with no moving parts, and can be an available solution for the battery replacement problem. In this research, we aim to conduct a numerical study to assess the power generation of TEGs and pyramid-schemed TEGs for electrical sensors used in IoT. Numerical analysis was performed using COMSOL Multiphysics Simulation. The simulation results are verified by comparing our TEG models with the results from the literature.

Programmable 3D Microfluidic Bio-Reaction Reservoirs Integrated with a Portable Pressure Pump for Miniaturization of Bioassays: Point-of-Care Detection of Monkeypox via LAMP-on-a-Chip

Mehmet Tuğrul Birtek, A. Nazente Atceken, Savaş Taşoglu
Mechanical Engineering Department, Koç University 34450 İstanbul, Türkiye
natceken@ku.edu.tr, stasoglu@ku.edu.tr

Point-of-care (PoC) devices have revolutionized healthcare by enabling remote diagnostics and therapeutics, with microfluidic systems playing a pivotal role in their advancement. This study focuses on the detailed engineering and characterization of three-dimensional hydrophobic valves to form programmable bio-reaction reservoirs. Using 3D-printed soft lithography, we meticulously investigated the effects of channel dimensions and surface properties on the burst pressures of these reservoirs, which ranged from 6.4 to 38.2 mbar. The bio-reaction reservoirs were demonstrated in both series and parallel configurations, offering versatile platforms for the miniaturization and automation of biological processes. Our findings highlight the capability of these reservoirs to program flows in a variety of fluid samples, including water, plasma, blood and lysis buffer. Additionally, a portable pressure pump was developed to leverage the functionality of these hydrophobic valves, enabling precise control of fluid dynamics in PoC applications. The study culminated in the design of a microfluidic chip integrating two consecutive reservoirs for the autonomous execution of Loop-mediated Isothermal Amplification (LAMP) for the detection of Monkeypox virus. Primers were lyophilized within the bio-reservoirs, and the system achieved detection of Monkeypox RNA via smartphone imaging.

Session: S6-A **Paper ID:** 017

Development of Newton Raphson Based 1D Flow Network Solver

Çağrı Nart

TUSAŞ Motor Industry A.Ş 26210, Tepebaşı, Eskişehir, Türkiye

Cagri.Nart@tei.com.tr

In gas turbine engines; secondary air system (SAS) has a vital role such as providing air for cooling turbine blades, to preventing hot gas ingestion into disk cavities and sealing bearing chambers. SAS air mass flow rates must be calculated precisely to accomplish this difficult task. Due to the complex geometries through SAS flow path and difficulties in preparing CFD model, 1d flow modelling is preferred and used as an industry standard to perform internal air flow calculations. Flow domain is modelled with 1D flow elements such as K_{loss}, $C_{d,loss}$, Orifice, Pipe etc. This paper describes the theory of a inhouse computer program developed for performing steady state 1d flow network calculations with both compressible and incompressible fluids. The program solves conservation of mass, momentum and energy equations iteratively using related loss correlation for a 1d flow element. Generally total or static pressures are solved at network nodes as a primary variables and the other one is calculated as secondary variable. In this approach both total and static pressures are calculated simultaneously on a node than thermal calculation loop performed. Mass flow rates are modeled as a function of inlet total pressure and exit static pressure. Newton- Raphson method is chosen for solving these nonlinear system of equations due to converge and numerical stability advantages

Innovative Groove Designs for Enhanced Flat-Grooved Heat Pipe Efficiency

Gökay Gökçe[1], Barbaros Çetin[2], Zafer Dursunkaya[3]

[1]ASELSAN A.Ş. 06830 Gölbaşı, Ankara, Türkiye

[2]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[3]Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

ggokce@aselsan.com, barbaros.cetin@bilkent.edu.tr, refaz@metu.edu.tr

Grooved-heat pipes are efficient thermal management devices widely used in various industrial applications. Traditionally, research on grooved heat pipes has been confined to basic groove geometries, such as rectangular, trapezoidal, star-shaped, and omega designs, primarily due to the ease and cost-effectiveness of the manufacturing process. Moreover, intricate modeling requirements—phenomena like micro-channel flow, phase-change heat transfer (evaporation and condensation)—complicate the numerical analysis of novel groove shapes, further limiting exploration in this area. However, recent advancements in manufacturing technologies have made it feasible to produce complex but cost-effective groove designs. These innovations open opportunities to explore novel geometries that could significantly enhance the performance of grooved-heat pipes by optimizing capillary pumping and total heat transfer. In this study, a solution strategy, previously developed by the authors, is employed to evaluate the performance of grooved-heat pipes with non-traditional innovative groove geometries. This approach overcomes computational challenges by efficiently modeling the coupled heat and fluid flow processes, providing a robust tool to assess the potential benefits of advanced groove designs. Results show that these geometries offer improved thermal performance and highlight the advantages of moving beyond conventional geometries.

Mathematical Modeling of Momentum and Mass Transport in Liver-on-a-Chip Systems

Giray Yüksel[1], Özlem Tomsuk[2], Hüseyin Avci[2], Ender Yıldırım[3]

[1]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[2]Metallurgical & Materials Eng. Dept., Eskişehir Osmangazi University, Eskişehir, Türkiye

[3]Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

giray.yuksel@ug.bilkent.edu.tr, ozlem.tomsuk@ogu.edu.tr

havci@ogu.edu.tr, yender@metu.edu.tr

Organ-on-a-chips (OoCs) imitate the microenvironmental properties of human organs by integrating microfluidics into cell culture. Although OoCs hold significant promise in areas such as drug discovery, there is limited research on mathematical models that investigate transport phenomena within these microchips. This research presents a mathematical model that examines momentum transportation in the hydraulic circuit and mass transportation in a liver-on-a-chip system. The model is developed using MATLAB R2024a Simulink for the liver-on-a-chip system, which incorporates a peristaltic pump, a reservoir and debubbler tube, and silica-based tubing. The chip features two parallel channels separated by a PET membrane and a 3D cell culture. Momentum transportation is examined by applying Poiseuille Flow assumption and Hagen-Poiseuille Law, which allow the model to be transformed into an electrical circuit. Viscous dissipation of mechanical energy in silica-based tubing and microchip are modeled as resistances. Moreover, capacitive effects resulting from trapped air in the tubes are included. Mass transfer occurs between the channels and the tissue, and it is electrically modeled by the analogy between Fick's First Law and Ohm's Law. Diffusion is modeled as a resistance considering the apparent diffusion coefficient and surface area of the tissue. Furthermore, the mass accumulation in the tissue is modeled by a capacitor. The momentum and mass transport models are concatenated by a novel application of operational amplifiers. The peristaltic pump is modeled using artificial data, which is prepared with the consideration of flow rate and pump parameters.

Investigation of Thermal-induced Distortions in LBPF Process by Utilizing Finite Element Simulations

Yigit Karpat, Selena Şahin

Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

ykarpat@bilkent.edu.tr, selenas@ug.bilkent.edu.tr

Lattice structures are employed in aerospace and automotive industries to produce lightweight structures made of high-strength materials. Using additive manufacturing to produce such lattice structures has become a popular research area, and the laser powder bed fusion (LPBF) process has been commonly employed to produce such structures. When structures with small features are considered, thermally induced distortions on lattice structures are yet to be addressed. In this study, we have utilized finite element analysis to identify hotspots based on the temperature profiles observed on the layers during the build-up process. The analysis results have been compared for lattice structures with varying dimensions. Moreover, parametric optimization simulations were performed to adjust parameters that have significance over the geometric accuracy, such as scan speed and laser power. Our findings on thermal-induced distortions, especially for complex shapes like lattice structures, can provide valuable insights for additive manufacturing technology and improve distortion prediction for a given design.

Experimental and Numerical Investigation of Heat Transfer Performance in a Condensing Heat Exchanger

Cansu Deniz Canal, Ali Cemal Benim, Michael Diederich, Onur Karacay

Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

cansu.denizcanal@hs-duesseldorf.de, alicemal@prof-benim.com, michael.diederich@hs-duesseldorf.de, onur.karacay@hs-duesseldorf.de

The escalating global energy crisis necessitates the implementation of alternative energy solutions, such as biomass-fired boilers. Enhancing energy efficiency has emerged as a vital strategy, with waste heat recovery systems playing a central role in optimizing energy utilization. Advanced technologies, including condensing heat exchangers and enthalpy wheels, are widely employed in natural gas-fired boilers to recover latent and sensible heat from exhaust gases, thereby enhancing system efficiency and minimizing energy losses. Adapting these technologies for biomass-fired systems aims to achieve significant advancements in the efficiency and sustainability of biomass-based energy production. This study introduces a waste heat recovery system integrating a condensing heat exchanger and an enthalpy wheel to enhance the efficiency of biomass-fired boilers.

In the initial phase of this experimental investigation, a comprehensive evaluation of the condensing heat exchanger was conducted under various configurations and operating conditions. Simultaneously, a numerical model of the heat exchanger was developed and validated against the experimental data. This modeling approach aimed to accurately predict system behavior and minimize dependence on extensive experimental trials. The numerical study employed a three-dimensional computational framework using the commercial software ANSYS Fluent. A mesh independence study was performed to ensure the robustness of the results, and several turbulence models were utilized for the numerical simulations. A comparative analysis of these methods revealed a high degree of agreement, thereby confirming the reliability and accuracy of computational fluid dynamics (CFD) simulations in predicting the heat transfer performance of the condensing heat exchanger.

Modeling of Electromagnetic Field Distribution in a Digital Twin of a Laboratory-Scale Microwave Dryer for Clay Roof Tiles

Akshay Ujjani Narasimhaiah[1], Prajwal Dharmananda[1], Lucas Briest[1]
Nicole Vorhauer-Huget[1], Evangelos Tsotsas[1], Anne Tretau[2]
[1]Otto von Guericke University 39106 Magdeburg, Germany
[2]Materialforschungs-und-prüfanstalt Weimar 99423, Weimar, Germany
akshay.narasimhaiah@ovgu.de, prajwald2617@gmail.com, lucas.briest@ovgu.de
nicole.vorhauer-huget@ovgu.de, evangelos.tsotsas@ovgu.de, anne.tretau@mfpa.de

Simulation of a laboratory-scale microwave dryer for clay bricks was developed using COMSOL Multiphysics. The aim of the study is to investigate the temporal and spatial variation of the electromagnetic field strength inside the cavity as well as inside the clay brick (260x75x125mm) for different parameter settings. These include size and geometry of the brick, rotation of the turntable as well as operation of the mode stirrer. Parameters are varied in accordance with experimental settings summarized in *(Vorhauer-Huget et al. 2022). Maxwell equations are solved in the frequency domain assuming quasi-stationary conditions at each rotation point of sample and mode stirrer. This work investigates how the process parameters alter the penetration depth of the waves as well as the field-strength distribution. Novel insights deliver the base for development of a detailed understanding of the heating conditions of thermally thick materials, which will be important to avoid heterogeneous heating, material shrinkage and severe cracks.

Modeling of microfluidic MEMS-Based Capacitive Pressure Measurement

Seçkin Eroğlu[1], Ayda Önsoy[2], Can Önol[2], [2]Çağrı Öztürk[2], Barbaros Cetin[1], Ender Yıldırım[1]

[1]Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

[2]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

seckin.eroglu@metu.edu.tr, ayda.onsoy@ug.bilkent.edu.tr, can.onol@ug.bilkent.edu.tr, barbaros.cetin@bilkent.edu.tr, yender@metu.edu.tr

The emergence of microfluidics and biomedical micro electromechanical systems revolutionized in vitro diagnostics by enabling rapid, precise results from minimal sample volumes. Viscosity appears as a promising biomarker for diagnosing various pathologies. This study introduces modeling of a MEMS-based capacitive pressure sensor integrated to a microfluidic channel for measuring viscosity of body fluids as small as 0.1 mPa.s. The system includes a serpentine microchannel, which acts as a flow resistor, and two capacitive diaphragms for pressure sensing posed at the inlet and the outlet . The capacitive sensor and the serpentine channel were optimized to enhance the linearity and to ensure the required sensitivity by utilizing an analytical model, so that a change of 0.1 mPa.s of viscosity results in a change in the pressure drop in the order of 103 Pa under 12 µL/min flow with a serpentine channel length of 70 mm and hydraulic diameter of 100µm, corresponding to a linear capacitance change in the order of 10 fF. The analytical model was compared with a numerical model on COMSOL Multiphysics to ensure the accuracy. Fabricated sensors were also tested to experimentally verify the models. The sensors system, incorporating a serpentine channel and capacitive pressure sensors was also numerically modeled considering fluid-structure interaction (FSI). FSI model captures the diaphragm deflection, hence the capacitance, under fluidic pressure. Moreover, the model proves that the diaphragm deflection has negligible effect on the flow behavior, which showcases the potential of proposed viscosity sensing scheme in diagnostics. This study is supported by the Scientific and Technological Research Council of Türkiye (TÜBİTAK) under grant no. 22AG008.

Reliability Assessment of Fuel Rail Assembly under Thermo-Mechanical Stresses in the Engine Compartment of Passenger Cars

Serkan Kurt

Bosch A.Ş., R&D Center 16140 Bursa, Türkiye

serkan.kurt@tr.bosch.com

In this study, the reliability of the fuel rail assembly (FRA) in the internal combustion engine compartment is assessed under thermal loads. These thermal loads induce different deflections in the aluminum cylinder head and the steel rail body due to their differing thermal expansion coefficients. This discrepancy creates thermo-mechanical stresses in the rail body. Thermal loads act as oscillating loads in the engine compartment, which can lead to fatigue failure of the component. The fuel rail is a safety-critical component; if a fatigue crack occurs, gasoline could leak into the engine compartment, posing significant safety risks. Therefore, demonstrating the reliability of the rail body is a crucial activity. Finite Element Analysis (FEA) studies are performed to evaluate the effects of varying oscillating temperature loads and to observe the resulting thermo-mechanical stresses in potential critical regions of the rail body. Additionally, mechanical tests are conducted to generate the Wöhler curve for the rail body, which provides insights into its load capability (strength) and fatigue behavior. Following these analyses, reliability is derived through a comprehensive fatigue assessment. It is essential that the failure probability of the rail body remains below 1 part per million (ppm) throughout the service life of the product. This rigorous approach ensures that the FRA can withstand the thermal and mechanical stresses encountered in the engine compartment, thereby enhancing safety and performance.

HEMLAB Algorithm Applied to the ARD Capsule

Ozan Yıldız

Aeronautical Eng. Depatment, İstanbul Technical University 34469 İstanbul, Türkiye

yildizoz22@itu.edu.tr

The Atmospheric Re-entry Demonstrator (ARD) capsule of the European Space Agency (ESA) has been numerically analyzed using the HEMLAB algorithm to predict thermal and aerodynamic loads. The HEMLAB algorithm is a vertex-based finite volume solver for the compressible Reynolds-Averaged Navier-Stokes (RANS) equations, utilizing an efficient edge-based data structure. The turbulence model employed is the one-equation negative Spalart-Allmaras model. To further enhance computational efficiency, anisotropic mesh adaptation has been carried out using the metric-based anisotropic mesh refinement library, Python Adaptive Mesh Geometry Suite (pyAMG), developed by INRIA. The freestream conditions are selected based on the nozzle exit conditions of wind tunnel tests conducted by ONERA at the hypersonic S4 tunnel. For validation, the resulting heat flux, moment coefficient, pressure coefficient data, and surface contours are compared with the experimental results.

CFD Modelling of Laser Cutting Process of UHS Steels, Experimental Validation and Optimum Cutting Parameters Selection

Doğukan Çetin, Özgür Ekici

Mechanical Engineering Department, Hacettepe University 06800 Ankara, Türkiye

dogukancetin96@gmail.com, ozgur.ekici@hacettepe.edu.tr

Laser cutting is one of the popular metal cutting methods. Many researchers examined laser cutting process in terms of kerf surface formation, thermal stress effects and assisting gas jet effects on efficiency. However, they generally focused on lower thickness and hardness materials. In this study ultra-high strength steels, which are used for ballistic protection on combat vehicles and etc. are taken into consideration. UHS Steels are manufactured via quenching and tempering steps. In order to guarantee the ballistic properties of steel, it is essential to not to exceed tempering temperature during laser cutting. HAZ area is examined for varying feed rate and power values. A volumetric heat source method is adapted. Analyses are conducted via ANSYS Fluent and results are compared with experimental studies. For varying material, thickness, feed rate and laser power specimens cut via Bystronic Bystar 4025 CO2 laser cutting bench. Temperature distribution and cooling rate of warm zones are investigated with the help of thermocouples. For regions that are exceeding tempering temperature are mapped. Cooling rates of warms zones are compared with the Continuous Cooling Transformation diagrams of the steels. Cooling rates are intended to kept below a critical rate which transforms an un-tempered martensite structure at the kerf surface. A hydrogen embrittlement crack may occur at the kerf surface if the cutting parameters are not selected properly. Finally, this study will help us to determine optimum cutting parameters for obtaining best quality, minimum scrap rate laser cutting of UHS Steels.

A Non-Homogeneous Two-Dimensional Model for Frost Growth on a Plate-Fin Surface

Tingting Zhu, Hidde van der Bijl, Austin Labuschagne, Wilko Rohlfs

Mechanical Eng. Department, University of Twente, 7522 NB Enschede, Netherlands

t.zhu@utwente.nl, hidde.vanderbijl@hotmail.com, a.a.labuschagne@utwente.nl

The efficiency of air-source heat pump decreases notably when operating under frosting conditions, necessitating defrosting, which requires additional energy. Frost layers formed on fin surfaces increase thermal resistance, and block air passages resulting in pressure losses and deteriorating heat pump performance. To address this problem, the growth behaviour and regularity of the frost layer on the fin plate were studied using numerical methods in the present study. First, a one-dimensional model based on three governing equations for the frost temperature, density and height in the direction normal to the plates was formulated. This one-dimensional model makes use of a saturation assumption as a condition for desublimation of water vapour within the porous frost layer. This one-dimensional model was then extended to a two-dimensional variant by considering the effects of heat and mass transfer on the properties of the air passing through the channel and over the frost layer. This 2D frost layer was divided into a number of one-dimensional columns, over which air properties were modelled as constant along the column width, changing only from one column to the next. The model was compared to experimental data, and a mass transfer coefficient correction coefficient and a relation for the effective diffusion coefficient were introduced to ensure the model output more closely fit the experimental results used for validation. A good fit between experiments and the proposed model was found. A parametric study regarding the inlet air velocity and relative humidity was performed.

Multiphysics Synergy Analysis of Light, CO2 Mass Transfer and Fluid Dynamics in Microalgae Photobioreactors

Ruilong Wang, Ming-Jia Li

Beijing Institute of Technology, Beijing, China

wangruilong1997@163.com, mjli@bit.edu.cn

Photobioreactors (PBR) are critical systems for cultivating microalgae, offering promising applications in carbon sequestration and sustainable bioresource production. Achieving optimal reactor performance requires an integrated understanding of light distribution, CO2 mass transfer, and fluid flow dynamics. This study develops a comprehensive multiphysics evaluation method to assess photobioreactor performance by examining the synergistic interactions among these three physical fields. First, a novel synergy angle evaluation diagram considering the synergy angle between light and fluid flow (θ) and the synergy angle between CO2 mass transfer and fluid flow (φ) is introduced to quantify these interactions, enabling a systematic approach to reactor performance assessment and optimization. Then, by adopting the multiphysics synergy analysis method of illumination, CO2 mass transfer and fluid dynamics, the performance of various photobioreactor configurations, including flat-plate PBRs and column PBRs are evaluated with computational fluid dynamics. Finally, the evaluation results guide the structural optimization of PBRs, leading to significant improvements in microalgal growth and carbon sequestration efficiency. Several optimized structures of microalgae photobioreactors are proposed. This work provides a framework for analyzing and optimizing photobioreactor systems, contributing to the advancement of microalgae-based technologies for environmental and industrial applications.

ML Automated Microfluidic Circuit Design

Mehmet Tuğrul Birtek, Vural Aktaş, Bora Aktaş, Ahmed Choukri Abdullah, Savas Taşoğlu

Mechanical Engineering Department, Koç University 34450 Istanbul, Türkiye

vaktas19@ku.edu.tr, baktas19@ku.edu.tr, aabdullah22@ku.edu.tr, stasoglu@ku.edu.tr

Microfluidics enable high-precision and cost-effective processing of biological substances. However, designing and fabricating microfluidic chips typically require substantial expertise and numerous design iterations, posing significant barriers to entry for non-experts. We present, µFluidicGenius (µFG), an innovative machine learning-augmented microfluidic design tool accessible to any computer user. The tool allows users to simply drag and drop bio-reservoirs and linking channels in a designated space and specify desired flow rates. A 3D printable microfluidic circuit that deliver specified flow conditions is generated through utilization of advanced machine learning algorithms and mathematical modeling. Experimental evaluations demonstrated that µFG achieved a 92% precision in generating the desired circuits. µFG makes microfluidic chip design accessible to researchers, engineers, educators, and students by integrating a user-friendly interface and rapid computational methods. Our tool shows significant potential for future improvements by incorporating additional fluidic parameters into the training algorithm that will allow for more complex and varied microfluidic designs.

Precursor Detection for Flashback Events in Hydrogen Combustor via integrated Machine Learning and Nonlinear Analysis

Mustafa Mohamad, Rahul Agarwal

University of Calgary, 2500 University Dr NW Calgary, AB, Canada

mustafa.mohamad@ucalgary.ca, rahul.agarwal1@ucalgary.ca

As the global energy landscape shifts towards cleaner alternatives, hydrogen has emerged as a promising substitute for fossil fuels, offering a cleaner source of heat and power. However, the stable and safe combustion of hydrogen poses significant challenges. Notably, hydrogen flames exhibit a higher tendency for flashback compared to methane flames, which is a critical issue that must be addressed for the safe design of combustion chambers. This tendency is further complicated by the elevated burner temperatures, which can induce substantial thermal stresses, potentially compromising burner performance and longevity. Recent experiments have shown that thermoacoustic systems can exhibit nonlinear behavior. This has led to the adoption of nonlinear dynamical systems analysis methods to understand the unstable regimes in combustors. This approach avoids misinterpreting deterministic chaos as random noise in measured signals. This approach allows for the development of an early warning indicator for impending flashback events. Current methods, such as Support Vector Machines (SVMs), struggle to create a predictive framework using multiple warning indicators. This is due to the dependence on operating conditions, fuel combinations, and combustion chamber designs. In response to these challenges, we propose an integration of chaotic dynamical analysis with deep learning techniques. Initially, we apply chaotic analysis, leveraging phase space reconstruction, to unsteady thermoacoustic time series data (e.g., dynamic pressure) from a typical digital twin of industrial combustor. Subsequently, nonlinear combustion features are extracted from the thermoacoustic time series using recurrence matrices. These features are then used to train deep learning models, specifically ResNet networks, to predict the likelihood of impending flashback events. Thus, the development of data-driven methods for predicting flashbacks using machine learning tools is beneficial by two key reasons. Firstly, machine learning tools can extract a wealth of nonlinear features from time series data, outperforming traditional, handcrafted precursors of instability. Secondly, combining this machine learning model with dynamical systems theory can construct a more powerful prediction framework that possess the generalization capability needed across varied combustion systems for engineering applications

Session: S5-A

Paper ID: 039

Computational Analysis of Non-Reacting Flow in a Non-Premixed Burner Featuring a Plasma-Enhanced Bluff-Body Swirler

Fatemeh Bagherighajari, José Carlos Páscoa

Mohammadmahdi Abdollahzadehsangroudi

University Beira Interior Convento de Sto. António 6201-001 Covilhã, Portugal

pascoa@ubi.pt

This paper presents a novel configuration of a dielectric barrier discharge (DBD) plasma actuator, designed as a plasma swirler and integrated onto the surface of a bluff body in a non-premixed burner. The plasma swirler consists of DBD plasma actuators mounted at 90° in the radial direction, generating plasma-induced ionic wind in the tangential direction. A 3D numerical study of the turbulent, non-reacting flow regime in the combustion chamber is conducted, with the effects of the plasma actuators modeled using a phenomenological approach. Additionally, two alternative configurations of the plasma swirler, referred to as inward and outward plasma swirl generators, are analyzed. The study compares the tangential and radial velocity contours and profiles at various axial positions to evaluate the swirl generated by the different configurations. Flow streamlines visualizing the recirculation zones downstream of the bluff body, both with and without the plasma swirler, are also compared. The results demonstrate that the plasma swirler significantly enhances the characteristics of the turbulent recirculation zones. Variations in the swirl number and the spatial mixing index along the axial direction are further analyzed to quantify the impact of the plasma swirl generator on the flow dynamics.

Session: S5-E **Paper ID:** 041

Numerical Examination of Flow Channel Expansion and Contraction Effects on the Performance of a Polymer Electrolyte Membrane Fuel Cell

Enrico Zardini[1], [2]atemeh Bagherighajari[2], José Carlos Páscoa[2]

Mohammadmahdi Abdollahzadehsangroudi[2]

[1]University of Brescia, Piazza del Mercato, 15 - 25121 Brescia

[2]University Beira Interior Convento de Sto. António 6201-001 Covilhã, Portugal

e.zardini@studenti.unibs.it, pascoa@ubi.pt

This study numerically investigates the impact of flow channel expansion and contraction, specifically in "converging-diverging" and "tapered" configurations, on the performance of proton exchange membrane (PEM) fuel cells. A three-dimensional, multi-component, multiphase, and non-isothermal model was employed to simulate the PEM fuel cell performance under these configurations. Key performance metrics, including polarization curves, cathode reactant distribution, liquid water accumulation, and transverse flow patterns, were analyzed for various expansion/contraction flow channel designs and compared with parallel flow field configurations under reference conditions. The results demonstrate that expansion/contraction flow channels enhance reactant transport to the catalyst layer, increase reaction rates, and improve liquid water removal efficiency. At reference conditions, the converging-diverging configuration exhibited the highest performance, increasing net power density by 10% compared to the parallel flow field. This improvement underscores its effectiveness in facilitating reactant transport and liquid water expulsion, leading to superior overall performance.

Vortex Gust Encountered by a Flat Plate at 45° Wing Sweep

Arif Cem Gözükara[1], Orhan Hızalan[2], Ozan İbrahimağaoğlu[2], Okşan Çetiner[2]

[1]ASELSAN A.Ş. 06750 Akyurt, Ankara Türkiye

[2]Aeronautical Eng. Dept.,İstanbul Technical University 34469 Istanbul, Türkiye

cgozukar@aselsan.com.tr, hizalan@itu.edu.tr, cetiner@itu.edu.tr

This study investigates a swept wing encountering a discrete vortex gust using both numerical and experimental methods. The investigations consider a flat plate at a 45-degree sweep angle with an aspect ratio of 4 at Re = 10,000. The discrete vortex gusts are generated by the half rotation of an upstream plate with different half rotation periods in order to vary the intensity and the encounter time of the vortex gust. The results obtained from numerical simulations are compared with those of experiments that are conducted in the water channel of ITU Trisonic Laboratory using 2D2C Digital Particle Image Velocimetry (DPIV) for the visualization of the flow structures, in conjunction with simultaneous force measurements. The numerical simulations are performed by using both Unsteady Reynolds Averaged Navier-Stokes (URANS) and Delayed Detached Eddy Simulation (DDES) methods. The main objective of this study is to investigate the three-dimensional flow structure interactions in a massively separated flow over a swept wing. Previously, experimental and computational investigations were conducted for a 2D flat-plate wing aiming to assess the computational methods and use of these methods for simulation of fundamental parametric study of discrete large-scale vortex gust encounters. The focus was mainly on the lift coefficient variation and instantaneous flow structures, visualized as vorticity and/or velocity distribution at different cross-sections together with the streamlines. As another advantage over the experimental study, once validated, CFD investigations offered the visualization of the three-dimensionality of a nominally 2D flow and the correlation of the flow structures with the sectional pressure and shear stress distributions, valuable for flow separation prediction. Currently, under the defined conditions, both the numerical and experimental results show a remarkable match also for the lift transient and flow structures around the swept wing at zero-degree angle of attack. Since the discrete vortex gust encounter is a flow phenomenon which involves massive separation, computationally more expensive scale resolving methods such as LES and/or RANS/LES hybrid methods such as DDES is advised to be used for improving the fidelity of the simulation results. On the other hand, despite its relative simplicity, preliminary results obtained from URANS simulations are also in good agreement with the results obtained from DDES and experiments. Both computational and experimental results show that, compared to a 2D wing, for a swept wing encountering a discrete vortex gust, the lift transient is reduced and the peak value is delayed in all cases investigated.

Session: S2-B

Paper ID: 043

3D CFD Modeling of MHSI and Fire Propagation in an Aircraft Engine Bay

Ahmet Emre Kaynak[1], Ender Özden[1], Sıtkı Uslu[2], Emre Gümüşsu[1]

[1]TUSAS, Turkish Aerospace Industry 06980 Ankara, Türkiye

[2]Mechanical Engineering Department, TED University 06420 Ankara, Türkiye

ahmetemre.kaynak@tai.com.tr, sitki.uslu@tedu.edu.tr, sitki.uslu@tedu.edu.tr

Fire propagation is difficult for modeling due to multi-physical nature of the phenomenon including chemical reactions, species transport, participating media radiation and unstable flow regime. Many systems can use combustion, in other words, controlled fire for producing energy. Uncontrolled fire is mostly a result of an accident or a system malfunction. This work concentrates on uncontrolled fire propagation in an aircraft engine bay. There are two stages of the work. The first stage investigates numerical determination of the MHSIT (Minimum Hot Surface Ignition Temperature) of the fire. The second stage is related with propagation of the flame region inside the engine bay. 3D CFD analyses are conducted and results of the numerical predictions are compared with available experimental results.

Session: S2-A

Paper ID: 044

Pseudo-3D Modeling of Grooved Heat Pipes

Vahit Çorumlu[1,2], Zafer Dursunkaya[3], Barbaros Çetin[2]

[1]M. Celal Bayar University, Akhisar Vocational School

Electric and Energy Department, Manisa, Türkiye

[2]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[3]Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

refaz@metu.edu.tr, barbaros.cetin@bilkent.edu.tr

Grooved heat pipes are critical passive heat transfer devices widely employed in heat transfer and electronics cooling applications due to their superior reliability and capability of transferring high heat loads with small temperature differences. However, the analysis and design of grooved heat pipes may be challenging due to phase change and free surface, two-phase flow, and heat transfer. Our research group has recently developed a mathematical model for pseudo-3D analysis of grooved heat pipes under different operating conditions, which was named as Heat Pipe Analysis Toolbox (H-PAT). Currently, our mathematical model ignores the effect of the vapor flow within the heat pipe and does not take into account the pressure drop on the vapor side. If the vapor space is large, the viscous effects on the vapor side and at the interface may be safely neglected. However, the effect of vapor flow may be quite significant for heat pipes that have a confined region for vapor flow. In this case, the pressure drop on the vapor side and the viscous effects at the interface need to be included. In this study, our mathematical model has been extended to compute the liquid and vapor flows at a given cross-section by solving Poisson's equation with the appropriate boundary conditions via COMSOL Multiphysics.

Experimental-Numerical Method for Determining Heat Transfer Correlations in the Plate-and-Frame Heat Exchanger

Jan Taler, Ewelina Ziółkowska, Dawid Taler, Tomasz Sobota, Magdalena Jaremkiewicz, Mateusz Marcinkowski, Tomasz Cieślik

Institute of Power Engineering, Cracow University of Technology Cracow, Poland

tomasz.sobota@pk.edu.pl, magdalena.jaremkiewicz@pk.edu.pl

tomasz.cieslik@pk.edu.pl

Plate heat exchangers are used in heat substations for domestic hot water preparation. Water from the municipal water supply is heated by hot water from a district heating network. Cold mains water is often characterized by a high content of mineral salts, which are deposited on the plates' surface in scale. For this reason, plate heat exchangers become fouled and must be cleaned periodically and replaced with new ones after several chemical cleanings. The build-up of limescale on the plate surface is accompanied by a decrease in the heat transfer coefficient, which results in a decrease in the heat transfer from hot to cold water. In addition, the active cross-sectional area between the plates decreases, which causes an increase in the pressure drop on the heated hot water side. The thermal resistance of fouling is the difference between the inverse of the heat transfer coefficient of a fouled heat exchanger and the heat transfer coefficient of a non-fouled heat exchanger with clean plate surfaces:

$$r_f = 1/U_f - 1/U_c \tag{1}$$

where r_f is the thermal resistance of scale deposits [$m^2 \cdot K/W$], U_f is the overall heat transfer coefficient of fouled heat exchanger [$W/(m^2 \cdot K)$], and U_c is the overall heat transfer coefficient of the clean heat exchanger [$W/(m^2 \cdot K)$]. The heat transfer coefficient for a clean plate heat exchanger is determined by the formula:

$$1/U_c = 1/h_c + \delta/k_p + 1/h_h \tag{2}$$

where h_c and h_h are the heat transfer coefficients for cool and hot water, respectively, [$W/(m^2 \cdot K)$], k_p is the thermal conductivity of the plate [$W/(m \cdot K)$], δ is the plate thickness [m]. Based on flow-thermal measurements, the paper will present a numerical method for determining heat transfer correlations for calculating Nusselt numbers on the side of the cold and hot water in the plate-and-frame heat exchanger. The proposed method for determining the thermal resistance of scale deposits will be applied in a computer system to monitor thermal substitution fouling in Krakow's network.

Session: S1-D

Paper ID: 049

Numerical Analysis of the Frost Formation Over Protruded Flat Surfaces

Alper Abdusoglu[1], Kaan Demirhan[1], Enes Murat Yakut[1],

Altug Melik Basol[1], Mehmet Arik[1,2]

[1]EVATEG Research Center, Ozyegin University 34794 İstanbul, Türkiye

[2]Department of Mechanical Engineering, Auburn University, Auburn, 36849, AL, USA

kaan.demirhan@ozu.edu.tr, murat.yakut@ozu.edu.tr, arik@auburn.edu

Heat exchangers operating under severely humid conditions can be subject to excessive frost accumulation on the finned surfaces. At these conditions the leading edges of the fins are increased in thickness due to frost buildup. The increased leading-edge thickness can alter the heat transfer and frosting behavior of the heat exchangers in the ongoing operation. In this study, the frosting behaviour on protruded flat surfaces is numerically studied. Protruded flat surfaces are selected for their geometrical similarity with the fins with frost covered leading edges. Frost formation is simulated using a transient, two-dimensional Eulerian-Eulerian multiphase model in ANSYS Fluent. Frost accumulation is modelled through a mass source term implemented with user-defined functions. The coefficients of the source term account for the local temperature, humidity, and airflow velocity conditions and were tuned using flash mounted flat surface data. In this study simulations are carried out on protruded flat surfaces and the validity of the tuned model coefficients on predicting the frosting on protruded surfaces are evaluated by comparing the frost thickness and heat flux with the measurements conducted at the closed-loop frosting tunnel at Ozyegin University. Finally, the differences in the frosting physics on protruded flat surfaces are compared to flash mounted flat surfaces were discussed and model coefficients were modified accordingly and the resulting improvements in the predictions are evaluated.

Numerical Analysis of Frost Formation on Finned Tube Heat Exchangers: Effect of the Humidity Level on Heat Transfer

Kaan Demirhan[1], Alper Abdusoglu[1],Enes Murat Yakut[1],

Altug Melik Basol[1], Mehmet Arik[1,2]

[1]EVATEG Research Center, Özyeğin University 34794 İstanbul, Türkiye

[2]Department of Mechanical Engineering, Auburn University, Auburn, 36849, AL, USA

kaan.demirhan@ozu.edu.tr, murat.yakut@ozu.edu.tr, arik@auburn.edu

Finned tube heat exchangers are widely used in refrigeration systems. These heat exchangers are prone to frost accumulation under humid conditions. Frost accumulation can severely reduce the heat transfer capacity of the device by building a thermal barrier between the air and the fins and by reducing the air flow through the heat exchanger. This study numerically investigates the effect of air humidity level on the cooling capacity drop of a finned tube heat exchanger under frosting conditions. Simulations are conducted using the transient, two-dimensional Eulerian-Eulerian multiphase solver in ANSYS Fluent. Frost accumulation is modelled through a mass source term implemented via user-defined functions. The coefficients for the mass source term accounts for the local temperature, humidity, and airflow velocity conditions and are tuned using experimental data. The computational domain uses half of a single airflow passage between fins with symmetry boundary conditions. Numerical predictions on the frost thickness and the heat transfer rate are in close agreement with the experimental data gathered in the closed-loop frosting tunnel at Ozyegin University. The results show a significant influence of the humidity level on the evolution of the heat transfer rate under frosting conditions. The results are expected to aid in optimizing the design of finned tube heat exchangers for improved performance under frosting conditions.

MHD Natural Bio-Convective Flow of Radiative Jeffery Nanofluid Through A Vertical Permeable Cone with Viscous Dissipation Effects

Musawenkhosi Mkhatshwa

University of South Africa, Mathematical Sciences, 1710 Florida, South Africa

musawenkosi.mkhatshwa@ul.ac.za

The Jeffrey fluid model effectively represents viscoelastic behavior by differentiating between relaxation and retardation times. However, its thermal performance has been a limiting factor, hindering its broader use. To address this challenge, nanofluids, which possess superior thermal properties compared to conventional fluids, have been introduced. While nanoparticles provide significant benefits, their inclusion in conventional fluids can cause instability. Interestingly, the introduction of motile microorganisms into the nanofluid mitigates this limitation by boosting both thermal conductivity and mass transport, thereby stabilizing the suspension. This improved thermal conductivity is essential in industrial applications where conventional fluids are inadequate for achieving efficient cooling. As a result, this study explores the impact of suspended nanoparticles and viscous dissipation on the magnetized natural bio-convection flow of a radiative Jeffrey fluid containing motile microorganisms over a vertical permeable cone surface embedded in porous media under uniform surface heat flux and nanoparticle volume fraction flux conditions. By introducing appropriate dimensionless variables, the model equations have been transitioned into non-dimensional partial differential equations, which are numerically solved using the overlapping grid-based multi-domain spectral collocation method. The numerical results for flow profiles and engineering-relevant quantities are analyzed for various flow parameters. Key findings reveal that the Deborah number, magnetic field strength, and the influence of porous media all contribute to accelerating the fluid flow. Incorporating radiative heat flux into the Jeffrey nanofluid flow system enhances both thermal distribution and the rate of heat conveyance. The intense random motion of nanoparticles and the active movement of motile microbes significantly contribute to increasing the nanoparticle mass transfer rate and the density number of motile microbes, respectively. The results of this research have the potential to enhance the thermal efficiency of the working fluid and support advancements in the effectiveness of microbial fuel cells.

A Computer System for Online Determination of Thermal and Pressure Stresses and Remnant Lifetime of Pressure Components

Karol Kaczmarski, Jan Taler, Dawid Taler, Piotr Dzierwa, Marcin Trojan

Tomasz Sobota, Magdalena Jaremkiewicz

Department of Thermal Processes, Air Protection and Waste Utilization

Cracow University of Technology 31-155 Kraków, Poland

tomasz.sobota@pk.edu.pl, magdalena.jaremkiewicz@pk.edu.pl

The paper presents a new stress monitoring system for pressure elements of the power unit. It can monitor transient stresses from thermal and pressure loads in the power unit's start-up, shut-down, and load change. Circumferential thermal stress at the point of its concentration at the hole edge is determined based on the stress determined at a greater distance from the hole. The thermal stresses will be determined by measuring the temperature at one point near the component's inner surface or on the component's outer thermally insulated surface. The inverse problem of transient heat conduction is solved using the wall temperature measured at one point so the temperature distribution throughout the wall thickness is known. The thermal and pressure stresses are then determined where they are concentrated at the hole's edge. The thermal stresses calculated for the two aforementioned thermocouple locations were compared for experimental data. Calculations of thermal and pressure stresses at the edge of the boiler drum's opening during the boiler's start-up - shut-down cycle were carried out. A procedure for calculating the residual life of a pressure element based on thermal and pressure stresses determined online was presented. The developed stress and residual lifetime monitoring system may be applicable to thermal power plants. The large share of wind farms and photovoltaic cells forces the time-varying operation of thermal power units, which can lead to premature wear. The proposed residual lifetime monitoring system allows for the start-up and shut-down of thermal power boilers to ensure long-term and safe boiler operation.

Numerical Investigation of Flame Dynamics and Mixing Characteristics of a Partially Premixed Swirl-Stabilized Combustor

Muhammet Ömer Özdilek

Defense Technologies Program, İstanbul Technical University 34467 İstanbul, Türkiye

ozdilek15@itu.edu.tr

This study investigates the flame and flow dynamics within a swirler-stabilized combustion chamber using OpenFOAM. The primary objective is to analyze the behavior of CH_4/air mixtures in a combustion chamber stabilized with a swirl generator. The swirler case is modeled based on experimental and numerical studies in the literature. The swirler geometry is characterized by a swirl number of 1.4, and the flow global equivalence ratio is 0.8. The effects of turbulent chemistry interaction were investigated using the Large Eddy Simulation approach. To ensure that the numerical setups are independent of the mesh, numerical models were tested with different numbers of elements. In order to verify the accuracy of the numerical model, the velocity profiles in the central recirculation zone and swirl jet were compared with experimental data. This study lays the groundwork for continued research on swirl-stabilized combustion simulations to investigate flame dynamics using CH4/air mixtures, and the validated study will be modified with different fuel injection strategies for mixture ratio improvement studies.

Large Eddy Simulation of Turbulent Non-reacting Flow Inside a Swirl-stabilized Combustor via Lattice-Boltzmann Approach

Burakhan Şüküroğlu, Alihan Atilla Çınar, Ayşe Gül Güngör

Aerospace Engineering Dept., Istanbul Technical University 34469 İstanbul, Türkiye

sukuroglu20@itu.edu.tr, cinarali17@itu.edu.tr, ayse.gungor@itu.edu.tr

The Lattice Boltzmann Method (LBM), which models fluid with distribution of particles at mesoscopic levels, is a highly used alternative approach to conventional CFD methods due to its low computational cost and efficiency in parallel applications. Our goal is to use LBM to investigate the turbulent non-reacting flow inside the swirl-stabilized dump combustor. For this purpose, an in-house parallel hybrid flow solver that uses the hybrid LBM-finite difference method (FDM) is developed via the Fortran programming language. The in-house solver uses a D3Q19-type lattice structure for density and velocity field computation. The species transport and energy equations are treated by using the finite difference method. The large eddy simulation (LES) approach with the Smagorinsky subgrid-scale modeling is used. The two-dimensional laminar backward-facing step problem is studied as a benchmark case to investigate and validate the solver. The recirculation region length obtained with the in-house solver is consistent with the experimental data. Then LES of the non-reacting flow inside a three-dimensional swirl-stabilized dump combustor is carried out. Preliminary results showed that the swirling flow dynamics are captured by analyzing the vortex structures due to the formation of a recirculation zone downstream of the jet exit. The final paper will cover the numerical investigation of turbulent, multi-species, non-reacting flow inside the swirl combustor by using the hybrid LBM-FDM approach. Moreover, the accuracy and capability of the LBM-FDM approach will be evaluated by comparing the results with experimental data and other numerical analyses.

Temperature-Dependent Characteristics of Interfacial Thermal Resistance Between Liquid Metal Gallium and Nanofillers: A Molecular Dynamics Simulation Study

Jiaqing Zhang, Jiaze Xi, Wenxiao Chu, Qiuwang Wang

Xi'an Jiaotong University, Xi'an, CN

jq_zhang@stu.xjtu.edu.cn, 502021899@qq.com, wxchu84@xjtu.edu.cn

Low-melting-point liquid metal-based filler-type thermal interface materials (TIMs) have important applications in efficient thermal management under extreme conditions, such as the aerospace environments, due to their excellent thermal conductivity and good deformation capability. However, one of the key challenges faced by such composites in practical applications is the insufficient wettability between the liquid metal and the solid filler, which leads to a significant increase in the interfacial thermal resistance, thus limiting their heat transfer efficiency. In this paper, based on a nonequilibrium molecular dynamics simulation method, the effects of temperature and solid-liquid bonding strength on the interfacial thermal resistance between liquid metal gallium and two solid materials (Diamond-Ga and Cu-Ga) solid-liquid interfacial heat transfer. In the simulation, the Tersoff potential function is used to describe the interactions between diamond atoms, the MEAM potential function is used to describe the interactions between gallium atoms and copper atoms, respectively, and the Lennard-Jones potential function is used to describe the interactions between different types of atoms. The results show that the thermal resistance of this LM-NF (Ga@NF) solid-liquid interface decreases with increasing system temperature, which is mainly attributed to the enhancement of inelastic phonon-interface scattering and Umklapp scattering at high temperatures, where the affinity interface has a weak dependence on the temperature change while the non-affinity interface exhibits a strong temperature dependence. In addition, with the gradual increase of solid-liquid binding strength, the interfacial adsorption effect is significantly enhanced, which is the main reason why stronger solid-liquid interactions can effectively strengthen the interfacial heat transfer.

Investigation of Aerodynamic Features Wind Escape Floors in Super-Slender Buildings

Yeliz Alevsaçanlar[1,2], Nilay Sezer Uzol[3], Bekir Özer Ay[1]

[1]Architecture Department, Middle East Technical University 06800 Ankara, Türkiye

[2]Architecture Department, Çankaya University 06815 Ankara, Türkiye

[3]Aerospace Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

yelizaksu@cankaya.edu.tr, nuzol@metu.edu.tr, ozer@metu.edu.tr

The recent increase in the construction of supertall and super-slender buildings has presented new architectural and engineering challenges. Wind escape floors have emerged as a critical feature among the aerodynamic modifications used to reduce wind-induced oscillations and improve structural stability. Designed to allow wind to pass through, these floors reduce the effects of wind gusts and pressure differences. For integration of wind escape floors to balance structural performance and space efficiency, the prediction of aerodynamic loads becomes important. In this study, Computational Fluid Dynamics (CFD) simulations are performed for a basic rectangular tall building with and without wind escape floors. A super-slender tall building with 12:1 aspect ratio with wind escape floors and square plan is studied with different numbers and locations of wind escape floors to investigate aerodynamic effects. Aerodynamic simulations are done by using OpenFoam CFD Solver. The results for aerodynamic loads and surface pressure distributions are presented and discussed.

Pore-scale Study on Melting Characteristics of Phase Change Materials in Rectangular Porous Media Based on Thermal Resistance Analysis Methods

Ting Wang, Xiangxuan Li, Ting Ma, Qiuwang Wang

Key Laboratory of Thermo-Fluid Science and Engineering, MOE,

Xi'an Jiaotong University, Xi'an, Shaanxi 710049, P. R. China

3120103197@stu.xjtu.edu.cn

Phase change materials (PCMs) are extensively utilized in solar energy storage, building heating, and waste heat recovery fields, due to their high energy storage density and consistent phase change temperature. However, due to its low thermal conductivity, rapid charge and discharge cannot be achieved. This challenge is typically addressed by incorporating porous media into PCMs. Although the melting characteristics of PCMs in porous media have been widely studied, the flow and heat transfer mechanisms remain inadequately defined, necessitating further research. Therefore, this study simplified the structure of porous media to a rectangular configuration and adjusted the aspect ratio of the pores to analyze the melting characteristics of PCMs at the pore scale. Besides, a factor U was proposed to evaluate temperature uniformity. Combining the maximum temperature analysis, the transient results of the thermal resistance and the evaluation factors were analyzed based on thermal resistance analysis method. The findings indicated that optimizing pore size can significantly enhance heat transfer efficiency, which is crucial for improving the thermal performance of PCMs. Furthermore, the analysis of the structural characteristics of porous media provides a theoretical foundation for understanding the effects of pore structure on fluid dynamics and heat transfer characteristics. The research results not only deepen the understanding of the melting process of PCMs in porous media but also offer a theoretical basis for the structural design of porous media, promoting advancements in related technologies within the fields of thermal energy storage and thermal management.

Session: S1-B **Paper ID:** 060

A Numerical Framework for Conjugate Heat Transfer Using the Immersed Boundary Method with a Compressible Solver

ChungGang Li

Dept. Mechanical Engineering, National Cheng Kung University 701 Tainan City, Taiwan

cgli@gs.ncku.edu.tw

This research develops a numerical framework for simulating conjugate heat transfer (CHT) phenomena using the Immersed Boundary Method (IBM) with a compressible solver. CHT problems, which involve the interplay of fluid flow and heat conduction across solid-fluid interfaces, pose significant challenges due to complex geometries and interface treatment intricacies. To address these, we propose a novel approach that leverages IBM to handle fluid-solid interactions efficiently. The governing equation is unified for conduction and convection by modifying the velocity and pressure for the preconditioned Navier-Stokes equation, allowing the numerical algorithm for compressible flow at low Mach numbers to be directly applied to CHT problems. The framework integrates a Locally One-Dimensional CHT model to accurately determine the interface temperature as a weighted average of solid and fluid properties, ensuring energy conservation and numerical stability. This model simplifies the treatment of the fluid-solid interface, effectively addressing numerical instability, particularly in cases with large differences in thermal conductivity, while enhancing ease of implementation. Validation against benchmark problems demonstrates the framework's accuracy in predicting temperature and velocity fields. Furthermore, the method is applied to a practical engineering case involving a rotating fan and a complex heat sink, showcasing its ability to handle intricate geometries and substantial thermal conductivity differences, thereby broadening its applicability in thermal engineering.

A Numerical Framework for Conjugate Heat Transfer Using the Immersed Boundary Method with a Compressible Solver

Mateusz Marcinkowski, Tomasz Cieślik, Dawid Taler, Jan Taler

Cracow University of Technology, Warszawska 24, 31-155 Kraków, Poland

tomasz.cieslik@pk.edu.pl

This study presents a method for predicting the thermal parameters of individual pipe rows in finned heat exchangers, based on measured data (temperatures, flows) and artificial data (row number and exchanger ID). Artificial neural networks were used for prediction due to their ability to model complex nonlinear relationships. The Theil coefficient and mean percent absolute error were used as measures of prediction quality. The results indicate discrepancies between predicted and measured values of less than 3-5%, confirming the high accuracy of the developed artificial neural networks-based model. The results show that the use of neural networks can significantly improve the design, regulation and operation of thermal systems.

Numerical Investigation of Fuel Injection Port Design on Partially Premixed Methane-Air Combustor

Mehmet Kağan Adıgüzel, Ayşe Gül Güngör

Aerospace Engineering Dept., Istanbul Technical University 34469 İstanbul, Türkiye

ayse.gungor@itu.edu.tr

The location and design of a fuel injection port in a swirl-stabilized, partially premixed combustor significantly affect the mixing of fuel and oxidizer, which has a direct effect on flame stabilization, combustion efficiency, and emissions. This study aims to investigate the fuel injection port design effect for a swirl-stabilized, partially premixed methane-air flame. For this, two combustor configurations with different injector mechanisms are investigated. The simulations are performed using the open-source computational fluid dynamics solver, OpenFOAM. The large eddy simulation approach is employed to accurately capture turbulence-chemistry interactions along with the finite rate chemistry approach for combustion. The results are compared against each other and the experimental data. Preliminary results show that the design of the fuel injection port can be modified in a way that the recirculation can be enhanced, resulting in improved flame stability and reduced CO emissions due to fewer incomplete combustion zones. Final paper will present the details of the combustor design, numerical setup, validation of the computational approach and thorough discussion of the results.

Hybrid Numerical Simulation of MHD Mixed Convection in Nanofluid-Filled Cavities with Application to Electronics Cooling

Bilal El Hadoui, Youness Ighris, Mourad Kaddiri, Jamal Baliti

Sultan Moulay Slimane University 23000 Beni-Mellal, Morocco

mouradkadiri@usms.ma, jamal.baliti@usms.ma

The present study introduces a hybrid numerical approach developed by coupling the single relaxation time lattice Boltzmann with the finite difference method to study the problem of MHD mixed convection in a cavity filled with Al_2O_3/water nanofluids. The cavity contains a partially heated rectangular heater with a variable length representative of electronics cooling applications. The vertical wall on the right is cooled, while the other walls are adiabatic. The top wall moves uniformly, while the other walls are kept stationary. A parametric analysis is performed to examine the effects of the main parameters, namely Peclet number, Hartmann number, and nanoparticle volume fraction, on the optimization of fluid flow and heat transfer. The thermophysical properties of the nanofluid, such as viscosity, thermal conductivity, and electrical conductivity, are modeled based on experimental data for better accuracy. Results, highlighted by temperature and velocity profiles, flow intensity, heat transfer rates, and enhancement ratios gave evidence of the enhanced dynamic and thermal performance with using the nanofluid in the case of strong magnetic fields, and they are more enhanced by increasing the Peclet number. The experimental models developed for thermophysical properties revealed an improved accuracy compared to conventional theoretical models. They also indicated an optimum nanoparticle loading at which maximum heat transfer rate is obtained depending on other parameters.

Numerical Approach for Entropy Generation and Exergy Destruction of Isobutane Condensation Flow in Microchannels

Anıl Başaran[1], Ali Cemal Benim[2]

[1]Mechanical Eng. Department Manisa Celal Bayar University 45140 Manisa Türkiye

[2] Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

anil.basaran@cbu.edu.tr, alicemal@prof-benim.com

The extensive use of microchannel heat exchangers in various industries has sig-nificantly increased the importance of refrigerant condensation flow within mi-crochannels. The current study aims to provide a numerical model for entropy generation and exergy destruction rates of condensation flow inside the micro-channels. This study presents a series of numerical simulations of condensing R600a (isobutane) in microchannels. R600a is an eco-friendly compound and a good alternative to environmentally hazardous refrigerants. The investigation fo-cused on circular microchannels with different hydraulic diameters changing be-tween 200 and 600 μm. The various mass fluxes (ranging from 200 to 1200 kg/m²s) and inlet vapor qualities (from 0.3 to 0.9) were analyzed for a thorough evaluation. In the numerical simulations, the Volume of Fluid (VOF) method was utilized alongside the Lee model, which manages phase changes at saturation temperature. The entropy generation rate and exergy destructions of the conden-sation flow of R600a inside the microchannel were numerically predicted using the proposed numerical model. Predicted values were compared to the theoretical-ly calculated actual values. Considering all simulations, it was found that the mean absolute percentage error (MAPE) of simulated entropy at the outlet of the microchannel is 17.80%. The mean absolute errors of entropy generation and ex-ergy destruction rates were computed as 8.04 10-8 W/K and 2.39E-05 W, respectively.

An Immersed Boundary Approach for Encountered Geometries in Multiphase Flows with Pseudopotential Lattice Boltzmann Model

Biswajyoti Baishya, Dipankar Narayan Basu

Indian Institute of Technology, Guwahati, Mechanical Engineering, Guwahati, India

dnbasu@iitg.ac.in

The ubiquitous multiphase flow phenomena find essential scientific and industrial applications, requiring extensive experimentations for a comprehensive analysis of the associated physics. However, limitations regarding the experimental endeavors, such as the complexity of the setup, cost concerns, and the difficulties in capturing the transient interfacial phenomenon, led to the development of promising computational techniques, notably the lattice Boltzmann (LB) model, which has seen significant progress over the last two decades. One of the primary reasons behind the spurious force oscillations and spurious velocities in LB simulations with complex geometries is the stairstep approximation of the boundary when tackled with a halfway or fullway bounceback boundary condition scheme. Such spuriousness can lead to significant discrepancies in the obtained results and severely affect stability in the case of multiphase simulation because of the prevalent density differences. Thus, incorporating an appropriate immersed-boundary (IB) algorithm can be advantageous for addressing complex boundaries in multiphase flow simulations. Given the inherent uniform grid structure of the LB framework, which aligns well with the IB philosophy, the coupling of a suitable multiphase LB model with an IB algorithm seems feasible. In this context, the present work combines the partially saturated cell method (PSM) of the IB framework with the pseudopotential model (SC) of the multiphase framework, leveraging the diffusive nature of both models. The problem of a droplet splashing against a superhydrophobic surface is simulated, where the surface is treated as an immersed object. The wide ranges of Weber (We) and Reynolds (Re) numbers from 2.64 to 100 and 600 to 2500, respectively, are chosen, and the simulated results match satisfactorily with the available data from the literature.

Numerical Investigation of Copper Metal Foam Integration in Hybrid Battery Thermal Management System for Enhanced Energy Density and Optimized Temperature Control

Alireza Keyhani Asl, Noel Perera, Jens Lahr, Reaz Hasan

College of Engineering, Birmingham City University, Birmingham B4 7XG, UK

jens.lahr@bcu.ac.uk, reaz.hasan@bcu.ac.uk

A hybrid battery thermal management system (HBTMS) integrating phase change material (PCM), copper foam as longitudinal fins and layers, and liquid cooling has been numerically analyzed. The configuration consists of twelve 18650 lithium-ion batteries enclosed in an aluminum casing, with the copper foam serving a dual function: as longitudinal fins embedded within the PCM and as a layer inside the copper tubes within the cooling plates to enhance the liquid cooling performance. The numerical simulations utilized the enthalpy-porosity model to capture PCM behavior and the Darcy-Brinkman-Forchheimer (DBF) model to represent the copper foam dynamics. Metal foam fins were analyzed under the local thermal equilibrium (LTE) approach, while the foam layers were modelled using local thermal non-equilibrium (LTNE) conditions. Transient battery heat generation was incorporated through a lumped-capacitance thermal framework. This research addresses a critical gap by investigating the effective integration of metal foam in HBTMS to enhance both passive and active cooling. The voids within the metal foam structure not only improved conduction and convection heat transfer but also significantly reduced system weight, thereby increasing energy density. The results demonstrated a reduction of approximately 9 K in the maximum battery surface temperature compared to pure PCM cooling, while maintaining the maximum surface temperature difference below 1 K. Also, metal foam fins effectively regulated the PCM melting process by delaying its onset. Additionally, incorporating a metal foam layer reduced the number of required cooling plates, leading to a significant improvement in its energy density.

Eulerian-Eulerian Modelling of Discharge Process in Spouted Bed Solar Receivers

Cankut Erkaya[1], Arif Eren Özdemir[1], Can Akıcı[1], Görkem Külah[2], Murat Köksal[1]

[1]Mechanical Engineering Department, Hacettepe University 06640 Ankara, Türkiye

[2]Middle East Technical University, Mechanical Engineering, Ankara, Türkiye

canakici@hacettepe.edu.tr, gorkemk@metu.edu.tr

Directly irradiated spouted and fluidized particle receivers can potentially store thermal energy at higher temperatures than conventional molten salt receivers in concentrated solar power (CSP) applications, making them good candidates for next-generation CSPs. Furthermore, spouted and fluidized beds exhibit well-known peculiar characteristics such as high gas-solid heat transfer rates and uniform temperature distribution, which are also advantageous in solar thermal receiver applications. The design of prototype spouted and fluidized thermal receivers requires the development of gas-solid multiphase flow models validated by experimental data. In this study, a gas-solid flow model was developed for the discharge behaviour under convective cooling for spouted and fluidized bed thermal receivers using Eulerian-Eulerian (two-fluid) approach and its predictive capability was assessed using data sets from the literature as well as the results of the in-house experiments. The benchmark experimental data sets of Patil et al. (2015) for convective cooling in a gas-solid fluidized with 1 mm glass bead particles bed was used for validation and the interphase gas-solid heat transfer coefficient, the effective thermal conductivity of the solid phase, and wall boundary conditions were shown to be essential parameters for accurate modelling of cooling behavior. Furthermore, the Zehner and Schlunder (1970) model for the effective conductivity of the solid phase performed better than the kinetic theory approach. Based on the results of the Patil et al. (2015) study, the discharge behavior of a directly irradiated laboratory scale 0.15 m ID conical spouted bed receiver was simulated using two-fluid approach. The spouted bed receiver was built in a TUBITAK-funded project, and charging and discharging experiments were carried out for 0.95 mm CarboHSP particles using a 2kW metal halide lamp. The results show that two-fluid approach captures the general cooling trends for the duration of the simulations. The maximum temperature difference between simulation and experimental results was 7-8 °C, with similar cooling rates. The simulations also provide physical insight into the cooling behavior of CarboHSP particles.

Session: S1-E

Paper ID: 068

Hybrid FDM-LBM Method for Thermal Management of Electronic Components via MHD Natural Convection Using MWCNT-Fe_2O_3/H_2O Hybrid Ferrofluid

Youness Ighris, Bilal El Hadoui, Jamal Baliti, Mourad Kaddiri

Youssef Elguennouni, Mohamed Hssikou

Sultan Moulay Slimane University 23000 Beni-Mellal, Morocco

mouradkadiri@usms.ma, jamal.baliti@usms.ma

The emergence of hybrid nanofluids is currently attracting considerable interest among scientists as a promising solution for improving the thermal management of electronic components. This paper presents a study of the natural convection-enhanced thermal transport performances of a hybrid ferrofluid (80%Fe2O3 - 20%MWCNT) in a square cavity containing a partial heat source under the influence of a magnetic field. The study uses a hybrid method combining the lattice Boltzmann method and the finite-difference method. The effects of multiple parameters have been examined: heat source size ratio (5% ≤ Sr ≤ 25%), Rayleigh number (104 ≤ Ra ≤ 106), nanoparticle volume fraction (ϕ = 1%, 3%, 5%), as well as Hartmann number (0 ≤ Ha ≤ 100). Results are presented as isotherms, streamlines, velocity and temperature profiles, and average Nusselt numbers. The findings show that an increase in Rayleigh number, nanoparticle volume fraction, and heat source size ratio leads to an increase in Nusselt number, indicating an improvement in heat transfer performance. Conversely, an increase in the Hartmann number reduces the Nusselt number due to the magnetic field's stabilizing effect on fluid motion. The ideal configuration for fluid movement is obtained with a heat source having a size ratio of 15%.

Fluid-Structure Interaction Analysis of a Small-Scale Piezoelectric Wind Energy Harvester

Turan Mirzalı, Levent Aydınbakar, Osman Turan

Mechanical Engineering Department, Bursa Technical University, Bursa, Türkiye

mirzalituran108@gmail.com, levent.aydinbakar@btu.edu.tr

The demand for sustainable energy sources for low-power devices such as medical implants, wireless sensors, and MEMS devices has driven research into innovative energy harvesting methods. These devices traditionally rely on batteries, which have limitations such as a short lifespan, environmental impact, and high maintenance costs. As an alternative, micro power generators (MPGs) offer the potential to convert environmental energy sources like thermal, solar, wind, and vibrations into electricity. Among these, vibration-induced resonance-based (VIV) energy harvesting methods are noteworthy. However, their reliance on specific resonant frequencies limits efficiency to narrow frequency ranges. To address this limitation, flow-induced vibration-based piezoelectric wind energy harvesting systems have been developed. These systems use a piezoelectric cantilever placed in an enclosure to harness self-sustained vibrations generated by wind flow. This mechanism converts airflow into periodic strains, producing electric energy through the piezoelectric layer, eliminating resonance frequency dependence. Flow-induced vibration energy harvesting is a novel concept being studied in configurations such as piezoelectric membranes in vortex streets, flutter mills made of flexible plates, and oscillating foils inspired by fish swimming dynamics. This study models the flow-induced vibration of a piezoelectric beam within a circular cross-sectional channel using fluid-structure interaction (FSI) analysis. The flow, modeled as three-dimensional, time-dependent, and incompressible laminar, will be simulated using OpenFOAM, while structural analysis will utilize FEniCS. The two open-source solvers are coupled through preCICE to enable seamless FSI analysis. The analysis aims to determine the flow velocity that initiates vibration. Once the beam's displacement is calculated, Kirchhoff's law will be applied to estimate the induced voltage according to piezoelectric capacitance, and electrical charge.

Investigation of the Effectiveness of Using Micro-Lattice Structured Meta-Material for Enhancing Heat Transfer

Mustafa Alperen Saygı, Suhan Okuducu, Levent Aydınbakar, Osman Turan

Mechanical Engineering Department, Bursa Technical University, Bursa, Türkiye

mustafaalperen.saygi@gmail.com, levent.aydinbakar@btu.edu.tr

Enhancing heat transfer plays a critical role in energy efficiency, performance enhancement, cost reduction, and environmental impact mitigation across various industrial applications. Consequently, exploring innovative methods to enhance heat transfer efficiency has become a significant focus in engineering research. In recent years, micro-lattice structures (MLS) have gained attention due to advancements in innovative manufacturing techniques, such as additive manufacturing. The complex geometries of MLS enhance the interaction between fluid particles, thereby increasing momentum transfer and improving convective heat transfer. However, the intricate geometries of these structures also increase flow resistance, leading to higher pressure losses and greater energy demands. Therefore, assessing the effectiveness of MLS requires careful consideration of both heat transfer improvement and pressure losses. This study investigates the conditions under which sheet-based Gyroid-type MLS are effective for enhancing heat transfer in channel flows. Numerical analyses will be performed under constant wall temperature boundary conditions for laminar and turbulent flow regimes, considering various Reynolds and Prandtl numbers. The impact of MLS with different porosity levels on heat transfer performance and pressure losses will be systematically analyzed. The objective is to develop a general criterion based on key parameters to determine the effectiveness of sheet-based Gyroid MLS in different scenarios. The simulations will be conducted using OpenFOAM, an open-source computational fluid dynamics tool, under assumptions of three-dimensional, incompressible, and fully developed flow. The findings are expected to provide valuable insights for the design and application of MLS in energy-efficient systems.

Numerical Study on the Effect of Loading Density on the Efficiency of Reverse-Flow Cyclone Separators

Seray Kader Kurtoğlu, Tuba Arvas, Levent Aydınbakar, Osman Turan

Mechanical Engineering Department, Bursa Technical University 16310 Bursa, Türkiye

seraykaderkurtoglu@gmail.com, tubaarvas29@gmail.com

levent.aydinbakar@btu.edu.tr, osman.turan@btu.edu.tr

Cyclone separators are widely used for gas-solid particle separation across various industries due to their low operating costs, robustness, and adaptability to harsh environments. These devices rely on centrifugal forces generated by the spin effect, which creates a swirling airflow that separates airborne particles. Particles lose momentum, fall to the separator's base, and are discharged, while the clean air exits through the outlet. Applications span multiple industries, including abrasive and solid fuel processing, chemical and pharmaceutical manufacturing, food production, and paper industries. Cyclone separators offer effective removal of dust and particulates, reducing pollution, safeguarding equipment, and enhancing energy efficiency. Their simple design, lack of moving parts, and minimal maintenance requirements make them a preferred choice over more expensive fil-tration systems. This study investigates the impact of inlet velocity and loading density on the efficiency of reverse-flow cyclone separators. Using OpenFOAM, a three-dimensional, time-dependent turbulent flow simulation is conducted to model the intricate flow behavior inside the cyclone. Additionally, the discrete phase model (DPM) is employed to simulate the motion and behavior of particu-late matter under varying conditions. The objective is to identify the critical loading density at which cyclone efficiency is maximized. Results from this study will provide valuable insights into optimizing cyclone designs for industrial applications, ensuring better separation performance and reduced environmental impact.

Session: S3-E **Paper ID:** 072

Open Space Indoor Air Quality and Comfort: Ventilation Versus Buoyancy Strategies

Xiaoyan Ma[1], Rachid Bennacer[2], Longfei Chen[1], Josua Meyer[3]

[1]Hangzhou International Innovation Institute Beihang University Hangzhou, China

[2]ENS-Paris-Saclay, 91190 Paris, France

[3]Dept. Mechanical and Mechatronic Engineering, Stellenbosch University, South Africa

chenlongfei@buaa.edu.cn, mxy_kay@163.com

Energy consumption and IAQ (indoor air quality) in thermal buildings are attracting increasing attentions due to energy crisis. The present work reports an indoor cooling and ventilation system with physical separation and cooling boards under different conditions (natural convection, forced convection and different mixtures). Complex flow structures are obtained in two cases: the injection "assisting" case and the extraction "opposing" case. Results show mixing ability is the key parameter that controls the thermal field homogeneity for the cooling necessity. The 3D flow structure shows that a simple analysis of the assisting and opposing effects is limited to predict the flow and thermal fields. A wide range of Ra (from less than 1 to 105) and Re (from 0 to 60) has been performed, and different flow structures and thermal fields distinguished different phenomena. The intensity of mixing ability is quantified in terms of homogenisation. For some moderate injection intensity, a lower Nusselt number appears to give a worse situation than in the case of pure natural convection.

Session: S1-B **Paper ID:** 073

Exploring the Synergistic Impact of V-Ribs with Cylindrical Vortex Generators on Solar Air Heater Performance: A CFD Approach

Jyoti Pal[1], Sunil K. Singal[1], Varun Goel[2]

[1]Indian Institute of Technology Roorkee, Roorkee Uttarakhand, India

[2]NIT, Hamirpur, Himachal Pradesh, India

jyoti.jpal@gmail.com, sunil.singal@hre.iitr.ac.in, varun@nith.ac.in

Solar air heater technology is one of the most efficient and cost-effective solutions for applications such as space heating, drying, heating, and ventilation. Despite its advantages, the system faces limitations, including the low heat transfer capability of air and the formation of a laminar sublayer at the heating surface, which inhibits effective thermal performance. To address these limitations, various artificial roughness configurations and enhancement techniques have been explored. The present study explores the synergistic impact of V-ribs with cylindrical vortex generators underside the absorber plate which has emerged as an effective method to significantly enhance the thermal performance of the system. The design parameters include relative rib roughness ratio, e/Dh (0.026-0.049), cylindrical diameter to rib height ratio, d/e (3.62-4.02), relative rib width ratio, $W/w = 6$, angle of attack, $\alpha = 60$□ and Reynolds number, Re (3200-19200) as the operating parameter are considered. The 3D computational fluid dynamics analysis using ANSYS Fluent is performed to investigate the heat transfer and frictional flow characteristics, followed by experimental validation. The maximum thermo-hydraulic performance parameter (THPP) of 1.82 is obtained for e/Dh of 0.047, d/e of 2.67, $\alpha = 60^{\circ}$ at a Re of 5200. The simulation results obtained from the study can be useful for academic researchers in designing and optimizing the parameters of SAH for low-temperature applications.

Performance Analysis of Wavy Microchannels: A Comparative Study of Traditional and Manifold Microchannels for Electronic Cooling

Alişan Gönül[1], Ali Hameed Mumen Al-Zaidi[2], Tassos G. Karayiannis[2]

[1]Indian Institute of Technology Roorkee, Roorkee Uttarakhand, India

[2]Brunel University of London, Uxbridge Middlesex, United Kingdom

tassos.karayiannis@brunel.ac.uk

Microchannels are crucial in electronic cooling due to their high surface area-to-volume ratio, which enables efficient dissipation of high heat flux, thereby enhancing device performance and lifespan. Evaluating the effectiveness of alternative channel designs in managing heat flux is essential. This study compares wavy-wall microchannels with both traditional and manifold microchannel approaches. The analysis focuses on heat transfer coefficient, friction factor, thermo-hydraulic performance, and temperature uniformity. Parametric investigations were conducted on the effects of channel wavelength, aspect ratio, and inlet mass flux variations. Furthermore, the Taguchi method was employed to identify optimal design dimensions and evaluate the impact of input parameters on per-formance metrics. This comparison provides valuable insights into the operational efficiency of each system under various conditions.

Enhancement of Phase-Change Efficiency by the Synchronized Reciprocating Rotation and Heaving Motion of the Thermal Storage Unit

Guojun Yu
Dept. Thermal Energy & Power Eng., Shanghai Maritime University, Shanghai, China
gjyu@shmtu.edu.cn

Enhancing phase change energy storage efficiency is a critical research goal, driven by the inherently low thermal conductivity of most phase change materials (PCMs). Conventional strategies—such as optimizing heat exchanger structures or chemically modifying PCMs—have had limited success in fully overcoming this bottleneck. In this study, a novel phase change storage device that integrates reciprocating rotational motion about a vertical axis with synchronized vertical oscillations is proposed, aiming to intensify heat transfer and expedite phase transition. A mathematical model is developed within a non-inertial coordinate system to examine how these two coupled degrees of freedom influence overall phase change performance. Parametric analyses highlight the roles of rotation and oscillation in shaping temperature distributions and the phase evolution. Results reveal that a synergistic approach—coordinating both rotational and vertical oscillatory motions—can significantly enhance thermal performance. This work provides fresh insights into phase change processes under complex motion conditions and presents a promising design concept for advanced thermal energy storage systems.

Assessment of Sintered Wick Heat Pipe Performance

Kaan Yener[1,2], Zafer Dursunkaya[2], Barbaros Cetin[3]

[1]TUSAŞ, Turkish Aerospace Industry 06980 Ankara, Türkiye

[2] Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

[3] Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

kaannyener1@gmail.com, refaz@metu.edu.tr, barbaros.cetin@bilkent.edu.tr

With the rapid advancements in technology, managing thermal systems has become an increasingly critical challenge. Heat pipes, known for their reliability, maintenance-free operation, and independence from external energy sources, are widely used in aerospace and electronics cooling industries. Accurate modeling of heat pipes prior to experimental testing offers substantial advantages, such as time and labor savings, by enabling the prediction of system performance and operational limits. This study introduces a thermal resistance network model designed to quickly and accurately assess the thermal performance of sintered wick heat pipes. The model focuses on solving liquid flow within the wick, which is treated as a porous medium, while assuming constant pressure and temperature for vapor flow. A key innovation of this model is its independence from experimentally derived heat transfer coefficients. Instead, voids at the liquid-vapor interface are simplified and represented as truncated spherical sections. These sections are incorporated into the thermal resistance network, making the model not only accurate but also practical as a design tool for optimizing heat pipe configurations. The study also explores the effect of pore radius on heat pipe performance and operational limits. Results indicate that increasing the pore radius initially reduces the temperature difference between the ends of the heat pipe due to a decrease in the proportion of non-functional regions at the liquid-vapor interface. However, beyond a certain threshold, further increases in pore radius lead to greater temperature differences, as the impact of thin-film effects becomes more significant. Similarly, the maximum heat load that the heat pipe can carry increases with pore radius, although the rate of improvement decreases at larger radii. This work provides valuable insights into the thermal behavior of sintered wick heat pipes and offers a robust modeling approach that eliminates the need for extensive experimental data. The findings can be directly applied to optimize design parameters, improving the efficiency and reliability of thermal management systems in critical applications.

Dimensionless Approach to Modeling and Predicting Coating Thickness in Continuous Galvanizing Lines Using CFD and Neural Networks

Cansu Şimşir[1], Levent Aydınbakar[2], Osman Turan[2]

[1]Borçelik Çelik Sanayi AŞ 16601 Gemlik Bursa, Türkiye

[2]Mechanical Engineering Departmant, Bursa Technical University, Bursa, Türkiye

csimsir@borcelik.com, levent.aydinbakar@btu.edu.tr

Galvanized steels are widely used in industries such as automotive, construction, and transportation due to their enhanced corrosion resistance and long service life. In continuous galvanizing lines, coating thickness is controlled by applying a high-pressure air jet to the surface of steel sheets dipped into a zinc bath held at its melting temperature. Ensuring uniformity and accuracy of the coating thickness is critical in this process. Existing studies on coating thickness prediction typically focus on single-phase and two-phase models. Single-phase approaches usually employ RANS-based turbulence models, while two-phase models use VOF and LES methods to simulate the interaction between air and zinc. However, the significant computational time required for two-phase models limits their application in parametric studies. Most models in the literature rely on nozzle inlet pressure for predictions and neglect the impact of nozzle geometry variations on outlet velocities and coating thickness. This study aims to develop a generalized and geometry-independent mathematical model for coating thickness prediction, based on dimensionless parameters valid over a wide range of conditions. Analytical correlations considering pressure gradient and shear stress were formulated, and a validated single-phase CFD model was used for numerical analyses of 990 different cases. The resulting data were utilized to construct an initial predictive model through artificial neural networks, establishing relationships between process parameters and coating thickness. Experimental tests will be conducted to address the known limitations of single-phase simulations[6] and refine the initial model, aiming to achieve a high-accuracy prediction model suitable for industrial conditions.

A First-Principles Study on the Adsorption Mechanism of Water Molecules on the SrBr2 (100) Surface

Ming-Yang Gao, Chuan-Yong Zhu, Liang Gong

China Uni. Petroleum (East China), Qingdao West Coast New Area, Shandong, China

lgong@upc.edu.cn

Reynolds-averaged Navier-Stokes (RANS) turbulence models have been shown to be robust and accurate for a wide range of turbulent flows and widely used in industrial applications. However, its modeling accuracy is limited for flows that exhibit complex physical phenomena, such as adverse pressure gradients (APG). This study investigates the capabilities of RANS models to predict upstream history effects in turbulent boundary layers (TBLs) subjected to APG. We study three TBLs using RANS approach with the k-w SST and Reynolds stress models using OpenFOAM and compared to experimental and direct numerical simulation (DNS) data for different TBLs. History effects as a result of the significant change in pressure gradient are evaluated in terms of velocity profiles, pressure gradient parameters, and flow budgets; mean flow budget, turbulent kinetic energy budget, and Reynolds stress budget wherever applicable. Our findings demonstrate that the upstream history effects are captured considering skin-friction and pressure coefficients with a certain similarity to experimental and DNS results, while the accuracy of turbulent kinetic energy profiles vary significantly depending on the specific conditions and choice of RANS model. This study further investigates these variations to better understand their behavior and potential influence on the predictive capabilities of RANS models under complex APG conditions.

Growth Mechanism of Boiling Bubble in Microscale within the Interplay of Ultrasonic and Thermal Field

Yong Guo[1], Zongbo Zhang[1], Yan Li[2], Liang Gong[1]

[1] China Uni. Petroleum (East China), Qingdao West Coast New Area, Shandong, China

[2]Ocean University of China, Shinan District, Qingdao, Shandong, China

zzb001_0@163.com, yanli@ouc.edu.cn, lgong@upc.edu.cn

The growth mechanism is a fundamental issue in comprehending the process of flow boiling in the microchannel within ultrasonic field. Our previous experimental investigations have demonstrated that microscale bubbles subjected to ultrasound undergo periodic expansion and shrinkage under alternating positive and negative pressure. The current study unveils the growth mechanism of boiling bubbles under the interplay of ultrasonic and thermal fields through Lattice Boltzmann method (LBM). The numerical results indicate that continuous heat absorption brings about the bubble's high internal pressure, which effectively counteracts the compressive effect exerted by the ultrasonic positive pressure on the bubble. Consequently, instead of collapsing observed in the acoustic field at low temperatures, bubbles rapidly grow within oscillation dominated by periodic ultrasound pressure. Moreover, with the increase in heat flux, the bubble's internal pressure rises. The influence of ultrasound on the growth bubble diameter gradually weakens. Besides, the uneven acoustic radiation force induces capillary waves on the bubble surface. This study proposes a simulation method for capillary waves by incorporating its dispersion relation, and the capillary wave number obtained by simulation aligns closely with the theoretical values.

Analysis of Pellet Defects Through Numerical Simulations of Flow and Heat Transfer in Underwater Pelletizers

Sukhesh Naik K I, Abhilash Chandy
Indian Institute of Technology Bombay, Mechanical Engineering, Mumbai, India
achandy@iitb.ac.in

Underwater die-face pelletizers, in which the polymer melt is extruded via a capillary, then cut, stress-relaxed, and chilled by a medium, are widely used in the thermoplastics compounding industry. These processes greatly affect the quality of the finished pellet. Under normal processing conditions and with well-designed equipment, underwater pelletizing should produce uniform pellets in size, shape, weight, and other desired characteristics for further processing or aesthetics. To guarantee consistency, it's crucial to identify common problems that cause pellet defects during start-up, such as worn pelletizing components and improper operating conditions. Since most pellet defects have multiple causes, managing one variable at a time and allowing time between adjustments will help you fully assess the impact of each change before making additional adjustments. One common practical problem in these applications is referred to as "longs and elbows", which occurs due to low knife speed or high extrusion rates. To address this particular issue, three-dimensional (3D) fluid dynamics calculations of a polymer in an extruder, along with the turbulent flow of heating oil and the conjugate heat transfer through the die are carried out using a computational fluid dynamics (CFD) software. Validation is carried out by comparing predictions of temperature on the die plate surface and pressure drops across the die plate to measurements. Using a detailed analyses of polymer flow speeds, this study examines the effect of melt temperature on the probability of the formation of longs in the pelletizer system through a comprehensive set of parametric simulations. Such studies can potentially guide the thermoplastics industry in choosing the optimum operating conditions for lesser pellet defects and thereby higher manufacturing efficiency.

Numerical Investigation of the Effect of Microscale Cavities on Nucleate Boiling

Deniz Öztunç, Abdullah Berkan Erdoğmuş

ASELSAN AŞ. 06830 Gölbaşı, Ankara, Türkiye

denizoztunc@aselsan.com, berdogmus@aselsan.com

The numerical prediction of phase-change phenomena plays a crucial role in many heat transfer applications. Boiling is considered an effective means of heat transfer due to its high heat transfer capacity and constant temperature nature. Particularly in the nucleate boiling regime, the vapor bubble formation enhances heat removal from the surface and allows for more efficient cooling. The surface conditions of the boiling site have a pivotal impact on the nucleate boiling heat transfer. Therefore, to enhance heat transfer, the formation of microscale cavities on the boiling surface is a viable approach, since the cavities act as vapor traps. The level of improvement resulting from the cavity formation depends on the cavity geometry. A common method for the enhancement of heat transfer with cavities in numerical applications is not recognized in the literature. The present study numerically investigates the effect of cavity formation on nucleate boiling heat transfer, with a central focus on a rectangular microscale cavity geometry. The Finite Volume Method (FVM) is used to model nucleate boiling with and without the rectangular cavities. The geometries are modeled in 3D, and the working liquid is selected to be liquid nitrogen. The liquid temperature is set to the evaporation temperature of nitrogen in atmospheric conditions and the wall temperature is kept constant. The nucleate boiling characteristic is examined in steady-state and the results are compared based on two key parameters: heat transfer rate and the mass flow rate of the boiling gas.

Assessment of Thermophysical Nature and Process Utility Limits of Nanofluids through Non-Dimensional Parameters

Melda Ozdinc Carpinlioglu

Mechanical Engineering Department, Gaziantep Üniversitesi 27410 Gaziantep, Türkiye

melda@gantep.edu.tr

In this presentation, the experimental data gathered from the state-of-the-art, mainly on cooling and lubrication performance of nanofluids, are referred to. The simplest idea is such that thermophysical nature of nanofluids have reflections on the performance of the particular process together with the utility limits of nanofluids . In this respect, the influence of both preparation methodology in turn with stability-life time of nanofluids and process operational restrictions should be consulted . In spite of the enormous amount of research on the manner, we still do not have a consensus on the generalized functional relationships offering a link between thermophysical nature and process nature in terms of well-known non-dimensional parameters which are independent of the process type. The discussion has a starting step with the interrelated properties of absolute viscosity and thermal conductivity of nanofluids which are not only representative of thermophysical nature but also under the severe influence of both preparation and operational constraints. The provided correlations with defined non-dimensional parameters, including cooling performance of nanorefrigerants and tribological performance of nanolubricants, are outlined . The discussion ends with a relativity approach handling individual non- dimensional parameters as their ratios with those of base fluids to highlight the utility limits of nanofluids as a contribution for generalization.

Session: S7-D **Paper ID:** 086

CFD Analysis of Aerospike Length and Blowing Effects on Aerodynamic Heating Reduction in High-Speed Flow

İbrahim Berkalp Gişi, Yavuzer Karakuş

TUSAŞ, Turkish Aerospace Industries 06980 Ankara, Türkiye

yavuzer.karakus@tai.com.tr

Interferometry is a precise technique that utilizes the phase differences between two or more light waves traveling different optical path lengths for measurements. Among its various types, fiber optic interferometers stand out due to their easy setup and low cost. The light beam exiting from and returning to the fiber optic cable tip interferes with the reference beam reflected back from the same cable tip, creating a phase difference with the reference beam along its different path. This process carries information about many parameters such as distance, the refractive index of the medium the beam passes through, the presence of gas or liquid interfaces along the optical path, and the shape of the reflective surface it comes into contact with. In this study, the effects of the tilt of the reflective surface used in the setup, probe height, and the medium through which the light passes along the optical path on the interferometric data have been investigated through simulations and validated with experiments.

Experimental Investigation of Open Circuit Voltage of a Li-ion Battery at Different Operating Temperatures

Melih Uğur Koca, Altay Tekin, Tanılay Özdemir

Hacettepe Üniversitesi 06640 Ankara, Türkiye

melihuguroca934@gmail.com, tanilayozdemir@hacettepe.edu.tr

Li-ion batteries have crucial role in power demanding applications along with the energy storage systems due to their prominent specifications such as high energy density, long cycle life, and low self-discharge rate. Their usage in advanced applications, however, is bound to establishing a suitable battery management system. The open circuit voltage (OCV) has a significant importance while estimating the battery's state of charge (SOC) or even to predict the produced heat during the charge and discharge processes in battery management systems. This study focuses on determining the OCV values of a cylindrical Li-ion cell at various state of charge values and operating temperatures. In addition, the SOC range is extended down to -10% to be able to cover the behavior of an over-discharging cell. The measurements are performed under 280 K, 300 K and 324 K to investigate the effects of the ambient temperature on the OCV measurements at each SOC value. The results show that the OCV values are impacted by the operating temperature, especially at the both ends of the SOC for NCA Li-ion batteries. Over-discharge OCV values are also presented for further OCV-SOC-Temperature mapping.

Numerical Investigation of Heat Transfer Characteristics in Liquid-Cooled Heat Sinks for SiC MOSFET Power Inverters

Athul Pradeep, Arul Prakash Karaiyan

Indian Inst. Technology Madras, Dept. Appl. Mechanics & Biomed. Eng., Chennai, India

am23s038@smail.iitm.ac.in, arulk@iitm.ac.in

Modern electric vehicles (EVs) use high-power density inverters for longer driving ranges and reduced recharging times. Silicon carbide (SiC) metal-oxide-semiconductor field-effect transistors (MOSFETs) used in power inverters are gaining popularity due to their high efficiency and ability to withstand higher temperatures. These compact transistors generate a significant amount of heat at their junction during high power density operations. However, they have a junction temperature threshold of 175ºC; exceeding it can degrade the performance and reliability of power inverters. Therefore, effective thermal management must be ensured for optimal operation of these inverters. Liquid-cooled heat sinks are gaining popularity in high-power density inverters because of their higher heat dissipation capacity and reduced material volume as compared to conventional air-cooled plate fin heat sinks (PFHS). However, effective cooling strategies for high-power density inverters are still under investigation.

The present study numerically investigates the 3D steady-state conjugate heat transfer in a liquid-cooled serpentine rectangular minichannel heat sink (SRMHS) designed to cool four SiC MOSFETs (25 W to 50 W power loss each) using distilled water. The composite layers of the SiC MOSFETs are modeled to capture their thermal behavior accurately [1], as shown in Fig. 1(a). The baseline SRMHS is designed to achieve a 40% reduction in material volume compared to available PFHS. The fluid domain under study is also illustrated in Fig. 1(b). Fluid mixing within the minichannel is enhanced by removing material from the baseline design and reincorporating them as vortex generators in two equal-area configurations: Rectangular Winglet Pair (RWP) and Delta Winglet Pair (DWP), as demonstrated in Fig. 1(c) and (d). Dimensions of the proposed designs are selected such that they can be manufactured using the Direct Metal Laser Sintering (DMLS) method [2]. Computational Fluid Dynamics (CFD) analysis is performed using a finite volume (FVM)-based solver (ANSYS Fluent), with second-order spatial discretization schemes and the Coupled algorithm for pressure-velocity coupling. The k-ω SST model is employed to accurately capture longitudinal vortices generated by vortex generators. The fluid inlet temperature is set to 20°C, with flow inlet velocity ranging from 0.1 m/s to 0.6 m/s. A uniform heat generation boundary condition is applied to reflect the point-source nature of SiC junctions.

Preliminary results show that the proposed designs successfully maintain the SiC junction temperature well below the threshold (175ºC), while air-cooled PFHS fail to achieve the same at high power loss. The addition of vortex generators reduces the maximum junction temperature by 5.5°C compared to the baseline SRMHS design, but with a considerable increase in pressure drop, as shown in Fig. 2. To achieve this reduction in junction temperature, the pressure drop required for SRMHS with DWP is comparatively lower than that of RWP, thereby reducing the fluid pumping power. Additionally, at higher velocities, the incorporation of vortex generators enhances the heat transfer coefficient by 35% compared to the baseline design. The performance of vortex generators is evaluated using the thermal performance factor (TPF), which highlights the trade-off between heat transfer and pressure drop. The SRMHS with DWP exhibits a higher TPF, suggesting enhanced heat transfer with a relatively lower pressure drop. These findings highlight the potential of SRMHS with vortex generators to provide effective thermal management solutions for EV power inverters.

Numerical Study of Combustion Instabilities in a Single Injector Combustor

Musa Onur Ozturkmen, Yusuf Özyörük

Aerospace Eng. Department, Middle East Technical University 06640 Ankara, Türkiye

yusuf.ozyoruk@ae.metu.edu.tr

The interactions between the unsteady heat release rate and pressure fluctuations in a combustion chamber may lead to thermoacoustic instabilities depending on the phase relationship between the two. In many instances, the linear stability characteristics of the chamber are determined by solving the nonhomogeneous wave equation in the frequency domain that takes these interactions into account, ignoring the convective effects. However, mean flow may have some impact on the phase between the unsteady heat release rate and pressure oscillations, and the stability characteristics may be altered. To investigate the convective effects, two different formulations—the linearized Euler equations and the acoustic perturbation equations—are utilized. One of the goals is also to assess how various formulations that include convective effects predict the thermoacoustic response of a combustor. In this study, finite difference discretization is employed for the two-dimensional axisymmetric versions of these formulations. An experimental test case from the literature is considered to evaluate the formulations for the prediction of stability characteristics against the longitudinal wave components.

Effect of Groove-Fin Width Ratios on the Thermal Performance of Grooved Heat Pipes

Ramazan Aykut Sezmen[1], Barbaros Cetin[2], Zafer Dursunkaya[1]

[1] Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

[2] Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

aykut.sezmen@roketsan.com.tr, barbaros.cetin@bilkent.edu.tr

Heat pipes are heat transfer devices that utilize phase change mechanisms to carry high amounts of heat with low temperature differences between two reservoirs. Due to this main advantage over conventional methods, heat pipes are popular in numerous industrial applications, particularly electronics cooling. Parallel to increasing number of applications, modelling techniques and predicting the thermal performance of heat pipes having different geometries became an important issue. In this study, in order to understand the effect of the groove and fin width variation for the same groove pitch value, a parametric performance analysis is conducted for a wide range of groove and fin widths. Heat transfer and the phase change mechanisms of working fluid are modeled using thermal resistance network for the flat grooved heat pipes. Temperature and contact angle differences between two ends of the heat pipe are used as the performance parameter for the better understanding the effect of the investigated parameters.

Numerical Analyses for Detection of Microplastic Waste by Using a Novel Microfluidic System with an Integrated Object Tracking Algorithm

Okan Külekçioğlu[1], Doruk Durmaz[1], Bushra Begum Khalak[1], Ela Bahşi[1], Selin Kasap[2]
Güleda Onkal Engin[3], Emine Ülkü Sarıtaş[4], Emine Yegan Erdem[1,4]

[1]Bilkent University, Mech. Eng., Ankara, Türkiye

[2]Bilkent University, Electrical and Electronics. Eng., Ankara, Türkiye

[3]Civil Engineerin, İstanbul Technical University, 34469 İstanbul, Türkiye

[4]Bilkent University UNAM, 06800 Ankara, Türkiye

yeganerdem@bilkent.edu.tr

Detection of microplastics in fluids, particularly water, has become increasingly important for many applications due to the harmful impact on pollution and health. This project aims to use a microfluidic channel with continuous flow to detect deformation in microplastics and implement an object tracking integrated image processing algorithm to sense and classify them from other materials in the chosen fluid. Initially, numerous potential designs are tested with simulations of fluid mechanics for particle tracing, and microplastic-microchannel impact. Chosen primary designs are tested for further understanding of microplastics behavior and plastic deformation.

Microplastics have been an area of focus for the last decade. To understand microplastic flow in microfluidics a study used porous regions in microchannels to induce collision between microplastics and channel wall [1]. It was found that high velocities result in less collisions and entrapment, and used larger pore ratios to raise capillary number and prevent further deposition. Another method of detection using nile red staining and fluorescence microscopy [2]. Using low flow rates and microplastic concentrations, microplastics deposit throughout the microchannel. The detection was possible due to the fluorescent intensity as organic particles and plastics are differentiable under this form of microscopy.

Since this project differs from more conventional methods of microplastic detection, there are a set of parameters that have been assigned to determine feasible designs. For particle tracing, the design would need to generate high velocity profiles where main and side channels mix to generate contact regions with wall spikes. Each design must achieve these velocities with multiple contact regions without causing blockage and accumulation on spikes and channel walls. In addition to these design parameters, the high velocities that are chosen have to result in plastic deformation with heated microplastics on impact with spikes.

From 20 designs, four final designs were chosen and determined to be most feasible. The first design included two side inlet channels and a main microchannel which has a straight and flat path and 100µm zigzags on both sides. Simulations showed that despite some particles hitting the sidewalls, many were passing through the channel with no impact. This design was still considered as it produced some initial interaction with the walls. To enable more impact points a spiked stair design was produced with 100µm zigzags located on the top side of each step with a total of 5 steps and 3 side channels before three steps. This design was chosen as simulations showed multiple successful impacts at high particle velocity and fluid flow rates while also not generating any clogs preventing continuous flow.

After finding success with steps design, a third design with the flat a flat main channel with small baffles extruded into the channel after eight side channel inlets. This baffled design was able to generate multiple interactions with the zigzags and baffles. The final design merged the baffled design with zigzags and the step design. This was done to explore optimal performance with microplastic and channel wall interaction. Following simulation, the baffled design produced multiple channel wall interactions with no clogging. After all designs showed adequate impact points and feasibility. Microplastic particles were analyzed for deformation comparisons, particularly with metals. Under the glass temperature of microplastics, 100C for polystyrene, the deformation analysis resulted in microplastic experiencing plastic deformation, contrary to metal particles. This further eliminates any error with differentiating between microplastics and metals after deformation in the channel.

References

[1] Haiyang He, Ting Wu, Yi-Feng Chen, Zhibing Yang, Science of The Total Environment, Volume 858, Part 2, 2023. 858, Part 2, 2023, 159934, ISSN 0048-9697.

[2] Mesquita, P., Gong, L., & Lin, Y. (2022). Micromachines, 13(4), 499..

Particle Dispersion And Deposition In Evaporating Sessile Droplets

Atacan Gürlek[1], Bushra Begum Khalak[1], Seden Zengin[1], Ela Bahşi[1], Reyhan Sever[2], Güneş Kibar[3], Emine Yegan Erdem[1,4]

[1]Bilkent University, Mech. Eng., Ankara, Türkiye

[2]Middle East Technical University, Mechanical Engineering, Ankara, Türkiye

[3]Adana Alparslan Türkeş Science and Technology University, Adana, Türkiye

[4]Bilkent University UNAM, 06800 Ankara, Türkiye

yeganerdem@bilkent.edu.tr

Sessile droplets commonly occur in fluid and gas interactions. Sessile droplets containing micro and nanoparticles interact within the droplet. When fully evaporated, particles deposit, producing a particular distribution on surfaces. This project aims to analyze and control the particle and fluid interactions, droplet behavior at fluid and gas interactions, and particle distribution variation on different surfaces, all of which will give a novel understanding of sessile particle-laden droplets. The four different textured surfaces contain hydrophobic and hydrophilic surfaces with varying wall profiles.

Earlier research has been done to gain an understanding of particle-particle interactions that can initiate and oppose the formation of the well known "coffee ring" particle distribution [1]. It was found that increasing particle size results in particle deposition at a droplet's center and the coffee ring dispersion otherwise. Methods of controlling particle dispersion and deposition throughout evaporation have been proposed but not implemented. Modifying substrates for thermal conductivity, selective heating, changing ambient parameters, and substrate texturing are a few potential methods [2].

Two textured surfaces are being tested. Silicon dioxide and Teflon coated structures fabricated with straight walls. Textures were tested with contact angle measurements and post-evaporation particle distribution analysis. All textured surfaces were tested with 5μL DI water droplets. Every surface was tested with three droplets for better accuracy. All samples were put aside until all fluid evaporated. After evaporation, all these samples were imaged from the top view to observe and determine the particle distributions. A particle distribution tracing algorithm was developed to scan each image collected and calculate the area occupied by particles.

The first textured surface experimented on was silicon dioxide-coated straight wall samples. Both Wenzel and Fakir droplets were observed in most samples. Wenzel droplets showed gradual decreases in the contact angle with the pillar length increments, except the results from the size ratio of 2.4. Despite the exception, for most of the sizes used, it can be concluded that the length of the pillars is inversely proportional to surface hydrophobicity. Samples with shorter pillar lengths produced a number of Fakir droplets. All the Fakir droplets behave similarly to each other, with higher contact angles than Wenzel droplets. Some samples did not produce any Fakir droplets.

Following these results, textured samples with Teflon-coated straight walls were tested. Unlike the silicon dioxide samples, the droplets produced were primarily Fakir droplets. The Wenzel droplets showed the same occurrence with higher size ratios having more Wenzel droplets due to the gaps. Fakir droplets were observed in nearly all size ratios and pillar lengths, allowing for a more comparable analysis. This trend is due to Teflon's expected hydrophobicity. All Fakir droplets increase in contact angle as sample size ratios and pillar lengths decrease.

After collecting sample images, the distribution tracing algorithm measured the areas produced by each textured surface. For silicon dioxide-coated straight wall surfaces with Wenzel droplets, it is found that the correlation between the contact angle and the spread of the droplet is negative. The particles are more spread out and concentrated along the edges of the droplet. For teflon coated straight wall samples, the same analysis was conducted with the Fakir droplets. Despite the expectation, the spread area is higher as the contact angle increases, showing a wider spread after the dispersion of the Fakir droplet. The particles were also concentrated in the center and less on the edges of the droplet.

References

[1] B. M. Weon and J. H. Je. Physical Review E, 82(1), 2010.
[2] D. Zang, S. Tarafdar, Y. Y. Tarasevich, M. Dutta Choudhury, and T. Dutta. Physics Reports, 804:1-56, 2019.

Experimental Analysis for Detection of Microplastic Waste by Using a Novel Microfluidic System with an Integrated Object Tracking Algorithm

Bushra Begum Khalak[1], Doruk Durmaz[1], Okan Külekçioğlu[1], Ela Bahşi[1], Selin Kasap[2], Güleda Onkal Engin[3], Emine Ülkü Sarıtaş[4], Emine Yegan Erdem[1,4]

[1] Bilkent University, Mech. Eng., Ankara, Türkiye

[2]Bilkent University, Electrical and Electronics. Eng., Ankara, Türkiye

[3]Civil Engineerin, İstanbul Technical University, 34469 İstanbul, Türkiye

[4]Bilkent University UNAM, 06800 Ankara, Türkiye

yeganerdem@bilkent.edu.tr

Microplastics, MP, have become a very large pollution contributor over the last few years, particularly during the emergence of COVID-19. Surges in demand and usage of plastic products led to high levels of disposal in water. This project developed a microfluidic design with zigzag structures and a heater to initiate microplastic collision and plastic deformation. The results are observed by an object tracking algorithm to help detect microplastic samples without any mechanical separation.

In the last few years, advancements in microplastic detection and separation have become more prominent. One study of microplastics flowing through microchannels with electrodes to generate a DC electrical signal [1]. Accumulation at the anode helps distinguish MPs from other materials. Some existing processes include filtration using comb-shaped regions at the tip of a long multiple fold channel [2]. Using optical spectrometry on each comb filter, microplastics are detected.

A microfluidic system is developed using multiple design simulations in COMSOL to determine feasible channel wall zigzag structures. Microplastics flowing through the channel will be introduced to their glass transition temperature and soften to deform when impacting the zigzags. Side inlet channels are also placed at multiple points along the main channel to instigate more interactions with channel walls.

Two materials will be used for microfluidic chips, to deduce usable designs and conduct proper observations. Initially chips will be made from PDMS, as it will be more affordable and simple to fabricate for design comparison. However, PDMS will not produce glass temperatures efficiently due to its insulative properties. After design selection, each chip will be fabricated on silicon wafers to be for proper deformation analysis.

Samples of microplastics and other material will have different percentages of deformation. As the microplastics flow through the main channel inlet and outlet, before and after images will be collected. These images will be compared using an object tracking algorithm to help detect and classify particles throughout continuous flow.

Experiments began with fabricated PDMS chips. These chips were fabricated using soft lithography with PDMS polymers, curing agents, and an SU-8 mold. Experimental setups included a syringe pump, a ceramic heater, a thermocouple, an optical microscope, and a smartphone camera. The first tested design included steps with 100 µm zigzags on the top side. Side channels are placed under multiple steps to generate high velocities with inlet flow rates of the side channels being higher than of the main channels. The main channel flow rate was kept the same for all designs evaluated. With these parameters, microplastics were observed to interact and impact channel side walls. This showed the design was able to produce the desired collisions and videos of these flows were recorded and inputted into the object tracking algorithm to evaluate deformation percentages.

This same process was performed on two other channels with zigzags, of 35 µm, and baffles at every side channel entrance. The final design tested had the same side wall profile with steps added to the baffled design. The design produced collisions with side walls and was then observed in the MATLAB Blob Analysis tool for object tracking. The analysis was done by evaluating the difference in pixelated areas of the MP. It found that as the MP flow through the channel, the area and shape of the MP changed. The success deformation can classify the particle as a MP.

References

[1] Elsayed, A.A., Erfan, M., Sabry, Y.M. et al. Sci Rep, 11, 10533, 2021.

[2] Zabihihesari, A., Khalili, A., Farshchi-Heydari, M., Eilaghi, A., & Rezai, P. New Journal of Chemistry, 2021

Shape Factor Analysis for the Convective Radiative Flow of Tri-Hybridized Nanofluid with Three Different Nanoparticles Over a Rotating Cone

Jafar Hasnain

Department of Computer Sciences, Bahria University 441000 Islamabad, Pakistan

jafar_hasnain14@yahoo.com

Nanofluids have lately emerged as leading alternatives or enhancements to conventional heat transfer fluids. Dispersed in a base liquid, the tri-hybridized nanofluid contains three types of nanomaterials with different chemical and physical properties. Real objects used in engineering contexts can be three-dimensional in form and have a changeable cross-section, as well as have surfaces that are inclined from a vertical angle. So, the present work aims to examine the effects of three differently shaped nanoparticles in a base fluid (water) flowing over a heated, rotating cone pointing vertically downward. Copper in the blade shape, titanium oxide in the cylindrical shape, and alumina in the platelet shape are used in the study. The flow is affected by a transverse magnetic field to the flow and thermal radiation. The heat transfer characteristics are also discussed with the heat source/sink phenomena and the boundary conditions with prescribed surface temperature or heat flux at the cone surface. The flow equations are solved numerically using the shooting approach after transforming them into ordinary differential equations using dimensional analysis. The results are plotted in the form of comparative graphs to examine the behaviour of fluid velocity and temperature with different parameters.

Physics-Informed Neural Networks for Heat Transfer and Fluid Flow Problems

Onur Ata, Atakan Aygun, Ali Karakus

Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

onurata@metu.edu.tr, alikarakus.me@gmail.com

Physics-informed neural networks (PINN) have emerged as a viable alternative to the classical numerical methods while solving partial differential equations. Their ability to obtain the solution without generating any mesh or using a discretization method without any labeled data have drawn attention in recent years. PINNs struggle to obtain the correct solution when the problem has multiscale behavior or highly nonlinear behavior. In this talk, we will show the solutions for thermal convection problems in different flow regimes. We will show the results for different thermal convection problems and show the applicability of PINNs on heat transfer and fluid flow problems.

Numerical Investigation of Thermal Comfort for Passengers in a Helicopter Cabin

Mehmet Külte, Sertaç Çadırcı

Dept. Mechanical Engineering, İstanbul Technical University, İstanbul, Türkiye

kulte17@itu.edu.tr, cadircis@itu.edu.tr

Different types of helicopters operate around the world, depending on their areas of use and operational capabilities. Additional systems are needed to ensure that operational conditions do not have a negative impact on the pilot and passengers and that the operation is carried out in a healthy manner. Environmental control systems (ECS) are used to provide thermal comfort for the cabin and cockpit in helicopters. The temperature and distribution of the air provided to the cabin and cockpit directly affects the thermal comfort of passengers and pilots. In this study, the thermal comfort of passengers in the cockpit of a helicopter is examined in terms of the Equivalent Temperature (ET), a measure of non-evaporative heat loss from the human body and the Effective Draft Temperature (EDT), an index that provides a measure of the local warmth or a cooling sensation. The flow is 3D, unsteady and the working fluid is air with constant thermo-physical properties. The study is conducted for the helicopter's ground environment, and the helicopter's outer wall heat transfer coefficient was applied as boundary condition. Helicopter's exterior temperature is 50°C and air temperature provided from ECS outlets is 10°C. Thermal comfort of the passengers' head, shoulders, chest, legs and feet is calculated separately with EDT and ET parameters. Thermal comfort analyses are performed for ECS outlet air with blowing angle changing between 0° and 40°. As a result of the study, local air velocity and local temperature have a strong effect on comfort in environments where homogeneous air distribution cannot be achieved, such as a helicopter cabin. Thermal comfort analysis has shown that air blowing angle between 15° and 25° provide the best thermal comfort.

Numerical Simulation of Marangoni-Benard Convection in a Planar Periodic Domain

Hakan I. Tarman

Mechanical Eng. Department, Middle East Technical Universit 06800 Ankara, Türkiye

tarman@metu.edu.tr

Thermal convection in a planar domain with periodic horizontal extent bounded below by a heated rigid plate and above by a deformable free surface with spatially varying surface tension is numerically simulated under the effect of gravity in a variational formulation. In the horizontal direction, the assumption of periodicity allows the use of Fourier series expansion. In the vertical direction, Legendre Lagrangian interpolant expansions are used for the velocity and temperature, while orthogonal Legendre polynomial expansion with order one less than that for velocity is used for the pressure. A lower-order expansion for pressure eliminates the problem of the pressure boundary condition at the lower plate. Since the governing equations, including the free boundary, are nonlinear, a Newton-like iterative method is used based on successive linearization with respect to the velocity, temperature, pressure, and the position of the free boundary. The linearization is performed on the continuous form of the variational integrals over the domain with the free boundary. High-order polynomial expansions and the Newton-like iteration procedure lead to accurate results and fast-converging iterations.

Numerical Investigation of the Performance of Centrifugal Pumps with Bionic Surface Structures

Adrian Pla Cerda

Layne Bowler Pup Company Inc, 06370 Ankara, Türkiye

adria.plcer@gmail.com

Centrifugal pumps are widely used in water supply systems and contribute significantly to operational costs due to their high energy consumption, which impacts accessibility for the population. Even a small percentage increase in efficiency could result in substantial benefits in terms of operational costs. Applying non-smooth surface bionic structures in rotating machines, such as groove structures, pit structures, and other non-smooth surfaces, has demonstrated their effectiveness in modifying the boundary layer, leading to noise reduction, drag reduction, erosion mitigation, and efficiency improvement. Additionally, implementing a non-smooth surface can, in some cases, be achieved with minor modifications bringing the opportunity to use an update for existing installed equipment. However, the use of these geometries is still in the development phase, with most research conducted in laboratories. The optimal type of bionic structures and their distribution along the surface warrant further exploration. This research investigates the impact of bionic structure geometry on hydraulic performance by studying a centrifugal pump with a bionic vane. Based on an existing radial impeller from a double-entry vertical pump, the pressure surface wall is modified using various structural arrangements. The flow field through the non-smooth surface bionic structure is analyzed via numerical simulation using the open-source CFD code OpenFOAM and compared with validated simulation results from the performance curves of the standard design. The results show the relationship between different geometric arrangements and the effect of the internal flow parameters affecting the drag, and consequently the efficiency of a long pump operating range.

Numerical Simulation of Wind Fence Effect on Coal Dust Emission

Sofya Yarikova, Andrey Epikhin

Ivannikov Inst. System Programming Russian Academy Sci., 109004 Moscow, Russia

andreyepikhin@ispras.ru, yarikova_sp@ispras.ru

Various methods and technologies can be used at open coal terminals to reduce dust emissions, depending on the specific conditions. One such method is the use of porous wind fences, which are already widely used in Canada, China, Japan and other countries. Various works have shown that the dust control efficiency of porous windbreaks can be more than 80%, but usually the authors only investigate the wind erosion mechanism of one coal stock pile. In real terminals the situation is much more complicated - dust is generated by many different industrial processes such as loading and unloading coal, transport, etc. There is also a lack of research into the effect of the distance between the fence and the piles, which in some cases can have a negative effect. In this study, a numerical simulation of the wind flow through a porous fence during coal loading and unloading is carried out. The effect of the fence position on the removal of coal dust particles is determined. The Eulerian-Lagrangian method is used for numerical modeling in the open source package OpenFOAM. Velocity profiles in different cross sections and particle masses are compared. The most and least efficient configurations for fence and coal stockpile locations are determined from the results of the calculations.

Investigation of Single Droplet Dynamics in a Microchannel Using Physics-Informed Neural Networks

Mohammadali Fakhri, Mohamad Ali Bijarchi, Mohammad Hassan Saidi

Department of Mechanical Engineering, Sharif University of Technology Tehran, Iran

bijarchi@sharif.edu, saman@sharif.edu

Interactions between a droplet and an immiscible liquid shrouding it, profoundly influence its dynamics and shape deformation. The aforementioned characteristics are essential in droplet manipulation utilized in a wide range of industrial applications. To comprehend the effect of various forces synergistically impact the droplet manipulation phenomenon, a two-phase flow of a single droplet within a microchannel as a case study is considered in the present investigation. To do so, herein, The Physics-Informed Neural Network (PINN), as an innovative method for solving complicated multiphysics problems described based on coupled partial differential equations (PDEs), is employed. The PDE system for two-phase flow consists of the continuity equation, the Navier-Stokes equations, and the Volume of Fluid (VOF) model equation. The continuity and Navier-Stokes equations govern the flow of main and droplet fluids through the microchannel, whereas the VOF model is utilized to describe the interactions between the droplet and the main flow, as well as track the interface. The fully connected neural network is trained using the Adam optimization algorithm. The results are validated with Computational Fluid Dynamics (CFD) methods. This work provides a novel framework, offering promising implications for applications where accurate interface tracking is critical for effective droplet manipulation, and it also provides a valuable insight into the use of PINNs for two-phase flow simuliations.

Numerical Investigation of the Influence of Structural Parameters on the Efficiency of Vertical Axis Tidal Turbines

Sofya Yarikova, Andrey Epikhin, Azat Valiev

Ivannikov Inst. System Programming Russian Academy Sci. 109004 Moscow, Russia

andreyepikhin@ispras.ru, yarikova_sp@ispras.ru, avaliev@ispras.ru

Vertical axis tidal turbines are used to generate electricity from water energy as a renewable energy source. There are several numerical approaches for modeling vertical turbines, but most of them are quite difficult and resource intensive due to complex unsteady flow structures. Three-dimensional RANS numerical modeling requires a lot of computational resources, so it is necessary to use simpler hybrid models that can calculate the hydrodynamics and flow field of the turbine with fine accuracy. In this study, a three-blade vertical tidal turbine is considered. The influence of geometrical parameters such as blade profile and chord size, radius of the turbine on its performance characteristics is investigated. Three-dimensional computational fluid dynamics simulations are performed using different approaches in OpenFOAM. The Arbitrary Coupled Mesh Interface (ACMI) and the Actuator Line Model (ALM) method implemented in the modified turbinesFoam library are used. Comparison of the ALM with sliding meshes and experimental data shows the effectiveness of the proposed method for calculating the characteristics of vertical axis tidal turbines. The influence of design parameters on the efficiency of a vertical turbine has been established.

Implementation and Validation of an Improved k-e Turbulence Model Based on Reynolds Number

Andrey Epikhin, Daria Romanova

Ivannikov Inst. System Programming Russian Academy Sci. 109004 Moscow, Russia

andreyepikhin@ispras.ru

Numerical simulations of the jet flow are conducted in the range of Reynolds numbers from 5100 to 86000. The calculations are performed using the OpenFOAM package and the k-epsilon turbulence model. Based on the modeling and comparison with experimental data, it is found that this turbulence model with standard values of coefficients does not accurately simulate the jet flow process. The choice of initial conditions for k and epsilon allows to approximate the modeling results in the near field (up to 12 nozzle diameters), but in the far field the jet dissipation process is incorrectly computed. The velocity decay is faster than in experiments. To solve this problem, a methodology is proposed that allows to find the optimal values of the k-e coefficients of the turbulence model by minimizing the deviation of the calculated data from the experimental data. The points along the centerline of the jet are selected as the data used for comparison with the experimental values. As a result, a Reynolds number based k-e model of turbulence is developed and validated in this study. The main advantage of the proposed model is its dependence on one input parameter - Reynolds number. The proposed model is implemented in OpenFOAM. Validation is performed on different jet flows, ERCOFTAC cases, including other types of flows, such as backward step, mixing layer, and flow over hill. It is found that the proposed model significantly refines the solution for incompressible jets, while not degrading it for other problems, which allows its use in other applied problems.

Application of Galerkin Reduced-Order Modeling Based on Proper Orthogonal Decomposition for Viscous Burgers' Equation

Onur Küçükoğlu, Nilay Sezer Uzol

Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

kucukogluonur@gmail.com

The increasing complexity of numerical simulations in engineering and scientific research demands efficient computational methods, particularly for high-fidelity computational fluid dynamics models requiring satisfactory spatial and temporal resolutions. Reduced-order modelling (ROM) provides a powerful alternative by approximating the solution of partial differential equations (PDEs) in a lower-dimensional space, significantly reducing computational costs while preserving essential dynamics. Among various ROM techniques, Galerkin Reduced-Order Models (Galerkin ROMs) based on Proper Orthogonal Decomposition (POD) have proven effectiveness due to mathematical rigour and versatility, a powerful method for addressing computational challenges in complex dynamical systems. Burgers' Equation, which involves convection and diffusion terms, is selected to demonstrate the methodology as a benchmark problem. The theoretical background of POD and its application with the Galerkin projection method are presented, providing a low-dimensional representation of the system while retaining essential dynamics. Numerical simulations of Burgers' Equation are conducted to generate the dataset required for GROM application during the offline data collection stage, employing consistent spatial and temporal discretization techniques. This work presents the computational efficiency and implementation of POD-based GROM with a demonstration work for flow problems. Challenges such as non-linear term treatment and boundary condition handling are addressed, and the potential for future advancements in real-time simulations and cost-effective flow field modelling are discussed.

A Novel Approach to C2C Variation Analysis in Heavy-Duty Engines: Multi-Cylinder Combustion Modeling with Detailed Chemistry

Serdar Güryuva

FORD OTOSAN 41470 Kocaeli, Türkiye

sguryuva@ford.com.tr

The stringent tail-out emissions limits for heavy-duty diesel engines necessitate complex and heavy after-treatment systems. This increased complexity, coupled with multi-mode engine calibration for emissions and fuel economy, significantly reduces tolerances in engine performance and engine-out emissions. While engine-level performance testing and emissions measurements can be conducted with high precision, assessing cylinder-to-cylinder (C2C) variation remains a challenge. Accurate C2C variation analysis requires measuring individual cylinder air flow rates and exhaust gas recirculation (EGR) levels, along with cylinder-specific performance data using pressure sensors. Due to packaging constraints, direct instrumentation is impractical; hence, a multi-cylinder combustion analysis with detailed chemical kinetics is the most viable approach. This paper demonstrates the engine test correlated single-cylinder combustion analysis methodology and its multi-cylinder application in the development of the Ecotorq 12.7L EU6 engine. The multi-cylinder model evaluates air flow, EGR distribution, and swirl variations under part-load and full-load conditions. Despite similar total air flow rates across cylinders, variations in EGR fractions and swirl ratios cause performance and emissions discrepancies, which require design modifications to the integrated intake manifold. This study represents a novel application of multi-cylinder combustion analysis for a heavy-duty engine, overcoming the challenges of system complexity and high computational demand.

Assessing the Impact of Hydrogen Reaction Mechanisms on Combustion Behavior in Heavy-Duty LPDI Engines

Talat Gökçer Canyurt, Serdar Güryuva, Cengizhan Cengiz

FORD OTOSAN 41470 Kocaeli, Türkiye

sguryuva@ford.com.tr

The European Union has implemented stringent CO_2 emissions regulations for heavy-duty vehicle (HDV) manufacturers to reduce greenhouse gas emissions from the transport sector. Non-compliance with these regulations can result in substantial financial penalties. To meet these regulation targets, HDV manufacturers must introduce zero-emission vehicles powered by either electric powertrains or hydrogen combustion engines. When hydrogen (H_2) engines are adapted from diesel counterparts with minimal design changes, the combustion analysis tools play a major role to evaluate the risks of abnormal combustion and high metal temperatures. One critical challenge in this process is selecting the appropriate reaction mechanism. The literature offers numerous H_2 chemistries, each with varying ignition times and laminar flame velocities, often tested in controlled environments such as shock tubes. These mechanisms must accurately predict real-world combustion behaviour, which can be difficult due to the complex nature of hydrogen combustion in an engine environment. This study evaluates several H_2 reaction mechanisms in combustion simulations of a production-ready H_2 LPDI heavy-duty engine. The model is developed under cold-flow conditions to accurately capture the mixing of H_2 and air and is validated with experimental cold-flow data. The combustion mechanisms are assessed using in-house single-cylinder research engine data, aiming to correlate performance and evaluate abnormal combustion risks. The findings reveal that most reaction mechanisms from the literature lead to unrealistic abnormal combustion patterns in the H_2 engine, producing unsatisfactory results that do not align with the observed behaviour in practical testing. These results highlight the need for more accurate reaction mechanisms and the importance of experimental validation to optimize hydrogen engine performance and ensure compliance with emissions regulations.

Development of a Commercial Gas Cooking Oven Design with Superheated Steam Technology and Waste Heat Recovery Focused on Green Transformation

Remzi Timur, Zafer Kahraman, Murat Hacı, Kadir İçibal, Hakan Serhad Soyhan

ÖZTİRYAKİLER A.Ş 34500 İstanbul, Türkiye

zkahraman@oztiryakiler.com.tr

Commercial gas cooking ovens, widely used in commercial kitchens due to their high capacities for cooking large quantities of food, cause higher emission rates than other kitchen appliances. This study aims to develop a prototype design of a commercial gas cooking oven that integrates superheated steam technology and waste heat recovery systems to enhance energy efficiency, environmental sustainability, and healthy cooking. The innovative design leverages superheated steam technology to reduce fat and salt usage in food preparation, promoting healthier cooking practices. At the same time, the waste heat recovery system optimizes energy consumption to reduce carbon emissions. The prototype oven design, developed through an R&D-based systematic approach, underwent simulations and design validations aligned with EN 203-1, EN 203-2-2, and EN ISO 14067 standards. The multifunctional combination of superheated steam, convection, and combi modes demonstrated compatibility with green transformation goals. Significant R&D achievements were made through university-industry collaboration, including adapting an Exhaust Gas Recirculation (EGR) system, commonly used in vehicles, to the prototype commercial gas oven. EGR systems typically reduce nitrogen oxide (NOx) emissions by recirculating a portion of the exhaust gases back into the combustion chamber, lowering combustion temperatures. In the prototype, waste gases generated during cooking processes were utilized to preheat the air-fuel mixture using a heat exchanger principle, resulting in energy savings. Simulation data revealed reductions in flue gas temperature and improved energy efficiency compared to a standard commercial gas cooking oven, contributing to green transformation in the industry.

Preliminary Findings from the Thermohydraulic Modeling of the Bozköy EGS Field (Aksaray), Türkiye

Aydın Çiçek[1], Ali Cemal Benim[1], Norbert Klitzsch[2]

[1]Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

[2]RWTH Aachen University 52074 Aachen, Germany

aydincicek2003@gmail.com, nklitzsch@eonerc.rwth-aachen.de

The installed conventional geothermal power capacity of Türkiye is 1,727 MWe as of December 2024. However, the extractable EGS potential estimations range between 15,000 to 25,000 MWe although available power potential is over 250,000 MWe. In this study, we simulated the heat and fluid transport of the Bozköy geothermal field, the hottest EGS site in central Türkiye. The reported bottom hole temperature at almost 4,000 m exceeds 341 ºC. During field studies, we collected various metamorphic rock samples belonging to potential reservoir. We used them to measure porosity, density, specific heat capacity and thermal conductivity in the laboratory. For the other parameters required for the simulation, such as permeability and the physical properties of the injected water, we assumed based on typical literature values at reservoir conditions. Temperature and pressure corrections were applied to the parameters measured in the laboratory. The data were used to parameterize a reservoir model that we used for simulating flow rates of 50, 75, 100, 125 and 150 kg/s for a period of 20 years. In our presentation, we present the reservoir model and the simulated thermal performances.

Drag Force and Mass Transfer in Gravity-Driven Bubbly Flows on Inclined Channels Using the UCLS Method

Nestor Vinicio Balcazar Arciniega, Joaquim Rigola, Assensi Oliva

Heat and Mass Transfer Technological Center,

Polytechnic University of Catalonia 08003 Barcelona, Spain

joaquim.rigola@upc.edu

Gravity-driven bubbly flow is relevant in nature and industry, for instance, bubble chemical reactors and mass-transfer unit operations in chemical process engineering or thermal equipment in power and refrigeration systems. This research presents a numerical study of drag force and mass transfer in gravity-driven bubbly flows on inclined channels. Direct Numerical Simulations (DNS) of bubble swarms are performed using the multi-marker Unstructured Conservative Level-Set (UCLS) method. In this framework, the multiple marker approach avoids the so-called numerical coalescence of fluid interfaces in bubble swarms, whereas interface capturing and surface tension computation are accurately performed by the UCLS method. Furthermore, the finite-volume method discretizes 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 convective term within the momentum transport equation, level-set advection equations, and mass transfer equation are discretized by unstructured flux-limiters schemes to avoid numerical oscillations around the interface and minimize the numerical diffusion. Such a combination of numerical techniques preserves the numerical stability of bubbly flows' DNS with high Reynolds numbers (100-1000) and high-density ratios (100-1000). The numerical code is parallelized for execution on supercomputers, up to thousands of CPU cores. Validations against empirical correlations from the literature are reported, including rising bubbles in a vertical pipe. Once the numerical code is validated, numerical and physical findings are reported for CD and Sh in single bubbles and bubble swarms rising on inclined channels with circular cross-sections. The inclination angle ranges from 15 to 90 degrees. On the other hand, Eötvös number (3-10), Morton number Mo (lower than 10e-8), which corresponds to the so-called wobbling regime, and low Schmidt number Sc=1 to keep the hydrodynamic and mass transfer boundary layer thickness of the same order.

Probabilistic estimation of the Enhanced Geothermal Systems (EGS) potential of the Bozköy field (Aksaray, central Turkiye) using Monte Carlo simulations

Ebru Bayar[1], Aydın Çiçek[1], Ali Cemal Benim[1], Norbert Klitzsch[2], Alper BABA[3]

[1]Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

[2]RWTH Aachen University 52074 Aachen, Germany

[3]Dept. International Water Resources, Izmir Institute of Technology 35430 Izmir, Türkiye

aydincicek2003@gmail.com , nklitzsch@eonerc.rwth-aachen.de,
alperbaba@iyte.edu.tr

Türkiye's total capacity for hydrothermal systems is estimated at around 4,500 MWe, of which 1,728 MWe is already installed (as of January 2025). In addition to the hydrothermal capacity, there is also the total capacity of the Enhanced Geothermal Systems (EGS), for which estimates range between 250,000 MWe and 400,000 MWe. Some studies assume that 15,000 to 25,000 MWe of this capacity can be realized on the basis of a feed-in tariff of 15 $ cents/kWh. In our study, we focus on an unproductive high-temperature geothermal field such as the Bozköy Geothermal Field (BGF) in Aksaray (central Türkiye). It has two deep, economically unproductive geothermal wells, but with 341 ^{0}C at a depth of about 4,000 m, it has the highest hole temperature ever measured in Türkiye. However, to our knowledge, there is no assessment of the EGS potential of the BGF to date. We estimate the total EGS capacity of the BGF using a modified USGS heat-in-place method based on a probabilistic Monte Carlo method. For this purpose, we developed an efficient Python program, which we validated by simulations with the commercial program Crystal Ball. We then simulated the EGS capacity for a 50 km2 license area and a 30-year operation with a single-flame power plant. We used a depth range of 3 to 5 km, a turbine efficiency of 70% and a conservative recovery factor of 1 to 2%. These simulations result in an electricity production of ~189 ~128 and ~76 MWe for the percentiles P10 (possible), P50 (probable) and P90 (proven). The additional sensitivity analysis shows that the temperature, reservoir volume, turbine conversion efficiency and recovery factor are key factors affecting the performance, while the heat capacity and density of the rock have a much smaller impact. Our verified simulation results show that the developed Python program can provide fast and reliable potential predictions for future EGS projects.

Direct numerical simulation of turbulent flow regime in a dense triangular rod bundle cell at Re=14200

Alexey Noskov, Vasiliy Oskirko, Roman Zaharov, Dmitry Oleksyuk

National Research Centre "Kurchatov Institute" 105005 Moscow, Russia

noskov_as@nrcki.ru, oleksyuk_da@nrcki.ru

The results of direct numerical modeling of adiabatic turbulent fluid flow in a cell of dense triangular rod packing at Re=14200 and their comparison with experimental data are presented. The calculations were performed using the STAR-CCM+. Dense rod packing is considered as a limiting case of rod placement in a fuel assembly, in which the non-uniformities of velocity, tangential stresses on solid walls and temperature in the presence of heating are most pronounced. As a result of the calculations, instantaneous and average characteristics of the turbulent flow are determined. The results of comparison of calculated and experimental data showed qualitative and quantitative coincidence of velocity fields, shear stress profiles and other characteristics. Additionally, this work compared the results of the DNS calculation with the results obtained using various RANS turbulence models.

Assessment of MCw Plasma Gasification via Instantaneous Response of Thermochemical Decomposition versus Post Processed Syngas Chromatography

Melda Özdinç Çarpınlıoğlu

Mechanical Engineering, Gaziantep University 27410 Gaziantep, Türkiye

melda@gantep.edu.tr

Solid waste management is an issue having been included either explicitly or having had interacting reflections implicitly in a number of the Sustainable Development Goals of the United Nations. The selected titles in this respect are such that 7: Affordable and Clean Energy , 12: Responsible Consumption and Production ,9: Industry ,Innovation, Infrastructure ,11: Sustainable Cities and Communities and 15: Life on Land. In recent years microwave plasma MCw is a referred innovative technology for solid waste gasification due to the process advantages it offers . Here an assessment of microwave MCw plasma gasification as a thermochemical decomposition process of solid granulated particles what we called as fuel into so called syngas what we called as alternative fuel is presented in reference to the previously conducted operational tests of MCw GASIFIER (TUBITAK –Research Grant Number 115-M-389) . Syngas generation is by means of an instanteneous thermochemical decomposition as a hybrid transient process governed by the so-called gasification time , t g. Instantaneous measurements of volumetric content and temperature of syngas during t g, the amount and material content of ash collected at the end of decomposition and generated amount of syngas, msyn are defined as instantaneous response data. Syngas chromatographic data on the collected and stored syngas samples for a storage time t s such that t s < t g are defined as the post processed data . Instantaneous response is just monitoring process from the start to the end of decomposition with the real environmental conditions while post processing is mainly chromatographic analysis on the mass and energy content of generated syngas sample under reference conditions of STP . A variety of volumetric and gravimetric parameters of syngas , ash and fuel are defined to describe the interactive influence of input power and air rate on gasification with a particular emphasis to the methodology of data as a contribution to the state of art . The introduced methodology which can be treated as independent of the system and operational conditions concerned , can provide a base for the comparative evaluation of ongoing enormous number of experimental research on waste to energy.

Numerical Investigation of Heating Effect in 3D Shock-Wave/Boundary Layer Interactions Near Protrusions in Hypersonic Flow

Muhammed Osama Balkhi, Bayram Çelik

Astronautical Eng. Department, İstanbul Technical University, 34469 Istanbul Türkiye

celikbay@itu.edu.tr, balkhi22@itu.edu.tr

One of the primary challenges in hypersonic flights is the intense aerodynamic heating due to shock-wave boundary layer interactions. The flow across surface roughness or protrusions interferes with the freestream flow, leading to a three-dimensional interaction creating flow features such as a bow shock and horseshoe vortex in front of the protrusion. This affects the local convective heating on the vehicle's surface, particularly on the reattachment region. In this study, we use an open-source density-based second-order accurate finite volume Navier-Stokes solver to understand the flow field in such interactions better. The effect of isothermal and adiabatic walls will be investigated, and a detailed interpretation of the flow field will be presented.

Comparison of the UCLS and coupled VoF - LS methods for hydrodynamics and mass transfer in bubbles

Nestor Vinicio Balcazar Arciniega, Joaquim Rigola, Assensi Oliva

Heat & Mass Transfer Tech. Center., Polytechnic Uni. Catalonia Terrassa, Spain

joaquim.rigola@upc.edu, asensio.oliva@upc.edu

Mass transfer in gravity-driven bubbles is relevant in nature and industry, for instance, bubble chemical reactors, mass-transfer unit operations in the chemical processing industry, and thermal equipment for power and refrigeration plants. This research reports a critical comparison of the Unstructured Conservative Level Set (UCLS) Method and the Coupled Volume of Fluid - Level Set (VoF - LS) Method for the Direct Numerical Simulation (DNS) of hydrodynamics and mass transfer in bubbles on 3D unstructured meshes. The UCLS method presents excellent mass conservation of fluid phases and accurate computation of surface tension force, as well as efficient implementation on unstructured meshes and parallel computing platforms. On the other hand, the coupled VoF-LS method incorporates excellent mass conservation of fluid phases using the geometric Volume-of-Fluid method and accurate computation of surface tension force by using signed distance functions, geometrically reconstructed from the minimum distances to the interface. DNS of mass transfer in bubbles is performed to assess the accuracy of interface-capturing methods. Concerning the numerical approach, the finite-volume method discretizes transport equations on 3D collocated unstructured meshes. Unstructured flux limiters schemes discretize the convective term of transport equations. Validations against empirical correlations from the literature are performed for both methods. Finally, the advantages and disadvantages of UCLS and VoF - LS methods on unstructured meshes are analyzed.

Development of Grid Model Requirements for Direct Numerical Simulation of Natural Convection in an Infinite Gap

Alexey Noskov, Vasiliy Oskirko, Roman Zaharov, Dmitry Oleksyuk

National Research Centre "Kurchatov Institute" 105005 Moscow, Russia

noskov_as@nrcki.ru, oleksyuk_da@nrcki.ru

Direct Numerical Simulation (DNS), due to the absence of empirical simplifications and turbulence models, provides results with a high degree of accuracy. The main contribution to the uncertainty of results comes from the implementation of the numerical scheme and insufficient spatial resolution of the computational grid. The latter leads to incomplete resolution of the entire spectrum of turbulent scales, which can significantly affect the reliability of the simulation. This study demonstrates the influence of spatial grid resolution on DNS results using the numerical simulation of turbulent heat transfer in a gap under natural convection conditions. Based on the research findings, recommendations are formulated for selecting the minimum acceptable control volume dimensions when solving heat transfer problems under natural convection conditions using DNS methodology.

Experimental Investigation of the Effect of Small Off-Surface Vortex Generator on the Aerodynamic Performance of NACA0012 at Low Reynolds Number

Farid Bakır[1], Abderrahim Larabi[2], Amrid Mammeri[3], Michael Pereira[1], Tarik Azzam[2]

Hamid Oualli[2], Florent Ravelet[1]

[1]Ecole Nationale Supérieure d'Arts et Métiers 75013 Paris, France

[2]Ecole Militaire Polytech., Laboratoire de Mécanqiue des Fluides, 16046 Algiers, Algeria

[3]Valeo Thermal Systems, France

farid.bakir@ensam.eu, amrid.mammeri@valeo.com,

michael.pereira@ensam.eu, tarik.azzam@gmail.com

Experimental study aiming to improve the aerodynamic performance of NACA0012 wing model at low Reynolds number of Rec = 2.6 × 104 was carried out using a micro cylinder rod acting as a passive vortex generator. The diameter of the control element is set as, d/c = 1.34% the chord length and positioned upstream of the leading edge at various distances from 1 × d to 4 × d. Wind tunnel measurements using HWA and flow field characterization by PIV were performed to assess the wing performance and to visualize the effect of the small rod on the flow structures around the aerofoil. In the low angles of attack region, α < 10◦, the control device causes a significant change in the behaviour of lift force as the non-linearity variation observed for the baseline case at such low Reynolds number is suppressed. In the other hand, in the higher angles of attack region, α > 10◦, a pronounced increase in the lift coefficient is observed for the appropriate rod location. Furthermore, PIV analysis has shown that the tiny rod provides a momentum from the freestream to the boundary layer so that the separation point is delayed downstream giving narrower wake which results in a significant drag reduction. Similarly, it is important to note that there is a large variation in the wake width in comparison with baseline case. In fact, the size of the separation region is significantly reduced by 50% when flow control is applied. This has a direct consequence to recover the pressure section peak near the leading-edge as well as the momentum balance between upstream and downstream. Through this qualitative analysis, it brings an additional insight to previous aerodynamic force analysis that could be the main reason for the pronounced aerofoil performance enhancement observed at the post-stall region for the microrod-aerofoil configuration.

Session: S3-A

Paper ID: 129

Transient Modeling of Heat Exchangers Using a Steady-State Approach

Mariusz Granda, Marcin Trojan, Jan Taler, Dawid Taler

Department of Thermal Processes, Air Protection and Waste Utilization

Cracow University of Technology 31-155 Kraków, Poland

dawid.taler@pk.edu.pl

This paper investigates the application of a steady-state model on the low heat capacity fluid side in the transient analysis of a heat exchanger. The focus is on a double-pipe system with water and air as working fluids. CFD modeling and mathematical modeling were used to solve selected transients using the Runge-Kutta and Euler methods. Transient analysis using a steady-state model relies on solving system of linear equations (here by the Gauss-Seidel method) that neglect the heat capacity. This method significantly reduces computation time, making it possible to monitor the transient operation of the exchanger online.

RANS Simulations of Gas Flow Within an Industrial Kiln for Sanitary Ware Manufacture

Eugenio Schillaci, Jesus Ruano, Joaquim Rigola, Jiannan Liu

Carlos David Perez Segarra

Universitat Politecnica de Catalunya 08034 Barcelona, Spain

jesus.ruano@upc.edu, joaquim.rigola@upc.edu, jiannan.liu@upc.edu, cdavid.perez.segarra@upc.edu

In the decarbonization processes of the industry, the integration of renewable energies, the substitution of hydrocarbons for fuels without environmental impact, the reduction of consumption, and the optimization of production processes are crucial within energy transition models at national, European, and global levels. Implementing green hydrogen as fuel in industrial production kilns, such as those required in ceramic ware manufacturing processes, would be a disruptive milestone in this decarbonization process due to their high energy-demanding operation [1]. In the case of the design of industrial burners, it is known that the increase in H2 within a small percentage in the combustion of CH4 (up to approximately 30%) allows the use of the same burners and kiln geometry, maintaining the same performance level. However, when that percentage increases, the burner itself and the geometry of the kiln must be redesigned to adapt to the jet's new characteristics and the corresponding distribution of air currents and temperatures. The adoption of new fuels would lead to the necessity of finding a new optimized distribution of hot air in kilns. In this work, the design of new green combustion technology furnaces is supported by global simulation of gas currents (fuel and air) inside the kiln to study and optimize their distribution. In the current work, gas flows are simulated using RANS k-w to provide a clear numerical picture of thermal and fluid dynamic processes occurring within the kiln geometry taken as a reference. This article follows a previous work from the authors [anonimyzed reference], where geometry and domain of reference were presented together with preliminary results. Here, additional numerical features are added to better estimate the temperature and flow distribution, seeking a good agreement with the temperature distribution profile along the kiln provided [1]. Some new features considered are compressible flow, improvements in the definition of boundary conditions, heat sources, and sinks to model the interaction between the air surrounding the ceramic ware and the ceramic elements. Moreover, a mesh convergence analysis is performed to assess grid independence. The validation of a full RANS simulation in a reference scenario will allow to perform new parametric

simulations with different fuel blendings at the burners, thus analyzing the impact of introducing green fuels. Moreover, the information extracted from the whole kiln can be employed as input for more detailed simulations to optimize the local behavior of burners and combustion air inlets. Finally, it will allow full analysis of conjugate heat transfer problems involving the fluid moving around a set of sanitary ceramic products and the pieces themselves, optimizing flow behavior locally.

References

[1] Carlos Cuviella-Suárez, David Borge-Diez, and Antonio Colmenar-Santos. Water and Energy Use in Sanitary-ware Manufacturing: Using Modelling Processes for Water and Energy Accounting and Decarbonisation. Springer Nature, 2021.

Microstructure-Level Investigation of Nanoparticle Transport in Collagen Hydrogels for Advancing Nanomedicine Design and Delivery Strategies

Ali Aykut Akalın[1], Ege Dağıstan[2], Altuğ Özçelikkale[1]

[1]Dept. Mechanical Eng., Middle East Technical University 06800 Ankara, Türkiye

[2]Mechanical Sci. & Eng., Uni. of Illinois Urbana-Champaign Illinois 61801, USA

akalinaykut@gmail.com, ege.dagistan@metu.edu.tr, aozcelik@metu.edu.tr

Nanomedicine involves therapeutic and diagnostic agents formulated as nanoparticles (NPs) that can be designed in various sizes, shapes and functionalities. Despite the major promise of nanomedicine for targeted drug therapies and vaccination technologies, its clinical translation remains limited by challenges in efficient delivery to the target tissues. In particular, the extracellular matrix (ECM), primarily composed of collagen along with other proteins, is a heterogeneous, porous, nanofibrous network that poses a significant barrier for NP delivery due to complex particle-fluid-structure interactions hindering advective and diffusive transport of fluids and NPs. Applications in drug delivery, tissue engineering, and disease modeling e.g. via in vitro microphysiological systems, are currently limited by the lack of a clear understanding of the transport characteristics of NPs in the tissue ECM. In this study, we consider microstructure-level modeling of fluid and NP transport in collagen hydrogels as in vitro ECM mimics to investigate the effects of ECM density, structural anisotropy and NP size on transport characteristics. Finite element method (FEM) simulations in COMSOL Multiphysics are combined with Brownian dynamics modeling in MATLAB to predict fluid and nanoparticle transport within collagen hydrogels. Key transport coefficients, including permeability and effective diffusivity, are evaluated for varying collagen concentrations (1.5 mg/mL to 6 mg/mL), fiber orientations (parallel, transverse, or random), and nanoparticle diameters (50 nm to 400 nm). In addition, permeability and effective diffusivity for selected conditions are experimentally measured for collagen hydrogels in bulk and microfluidic perfusion assays using fluorescence recovery after photobleaching (FRAP) and timelapse microscopy analysis. The predictions of the computational models are found to be in good agreement with the experimental measurements. These findings contribute to a deeper understanding of NP transport mechanisms, paving the way for improved nanomedicine design and delivery strategies.

Numerical Investigation of Hydrogen Addition on Flame and NOx Emissions in Methane Combustion for Industrial Kilns Using a New Reduced Mechanism

Jiannan Liu, Carlos David Perez Segarra, Eugenio Schillaci

Jesus Ruano, Joaquim Rigola

Universitat Politecnica de Catalunya 08034 Barcelona, Spain

jiannan.liu@upc.edu, cdavid.perez.segarra@upc.edu, jesus.ruano@upc.edu

Industrial kilns primarily operate based on the combustion of natural gas, which contributes significantly to carbon emissions and has a considerable environmental impact. Hydrogen is a promising alternative fuel due to its zero carbon emissions during combustion. Blending green hydrogen with natural gas offers a practical solution, utilizing existing infrastructures while reducing carbon-based emissions like carbon monoxide (CO) and carbon dioxide (CO2). However, this approach increases nitrogen oxides (NOx) emissions due to hydrogen's higher combustion temperature. NOx pollutants significantly impact air quality and public health. Therefore, transitioning from pure methane to pure hydrogen combustion in industrial kilns requires addressing the dual challenge of reducing both carbon and nitrogen-based emissions to meet environmental and sustainability goals. Understanding the chemical mechanisms underlying NOx formation in methane-hydrogen combustion is essential for developing effective emission control strategies. Since Zeldovich proposed the thermal NO formation mechanism, research on NOx formation has advanced significantly, revealing a complex interplay of multiple pathways that dominate under different combustion situations. Glarborg et al. (2018) categorized the primary routes for NO formation as the thermal NO, prompt NO, N2O mechanism, NNH mechanism, oxidation of HCN, oxidation of HNCO, and oxidation of NH3. Studies have contributed to modeling NOx formation through reduced mechanisms tailored to specific combustion scenarios. Gao et al. (2013) extracted a 36-step reaction mechanism for four major NOx formation pathways from the detailed GRI-Mech 2.11 mechanism. In the present study, a reduced mechanism for methane-hydrogen combustion was constructed and validated, with the ability to predict NOx formation. The proposed new mechanism, called ZG81, consists of 81 reactions and 28 species and was constructed based on the reduced Z45 mechanism by Zettervall et al. (2021) and the 36-reaction NOx formation mechanism summarized by Gao et al. Its accuracy was demonstrated by comparing the calculated laminar flame velocity of methane-hydrogen mixtures with experimental data. The open-

source software Cantera v3.0.0 is used to simulate the one-dimensional, adiabatic, steady, unstretched, laminar, and planar flame propagation. The mechanism's capability to predict NO formation was further validated using a non-premixed methane-hydrogen diffusion flame (JHC flame). The JHC flame is a MILD combustion experiment conducted by Dally et al. (2002), investigating flameless combustion by introducing a methane-hydrogen fuel jet into a hot air co-flow. The reactingFoam solver of the OpenFOAM CFD software toolbox is used to model turbulent flame simulations. The turbulent flow is described by the standard k-ε turbulence model in RANS simulations. The Partially Stirred Reactor (PaSR) model is implemented for modeling the turbulence-chemistry interaction process. The reduced ZG81 mechanism, is compared to the Z45 mechanism and the detailed GRI-Mech 2.11 mechanism to evaluate their relative performance and accuracy. Based on the proposed mechanism, the effect of various blending proportions, excess air ratios, and inlet air distributions on the methane-hydrogen flame in industrial kilns were investigated. The combustor is a cylindrical vessel with mutually perpendicular jets for fuel injection, along with primary and secondary air inlets. It is mounted on the kiln wall, with the jet flame ejecting horizontally into the external air domain. The optimization of flame behaviour and NOx emission reduction in the blending flame was investigated through regulating the airflow between the staged inlets.

References

P. Glarborg, J. A. Miller, B. Ruscic, and S. J. Klippenstein, "Modeling nitrogen chemistry in combustion," Progress in Energy and Combustion Science, vol. 67, pp. 31–68, 2018.

X. Gao, F. Duan, S. Lim, and M. Yip, "NOx formation in hydrogen–methane turbulent diffusion flame under moderate or intense low-oxygen dilution conditions," Energy, vol. 59, pp. 559–569, 2013.

N. Zettervall, C. Fureby, and E. Nilsson, "Evaluation of chemical kinetic mechanisms for methane combustion: A review from a CFD perspective," Fuels, vol. 2, pp. 210–240, 2021.

B. Dally, A. Karpetis, and R. S. Barlow, "Structure of turbulent non-premixed jet flames in a diluted hot coflow," Proceedings of the Combustion Institute, vol. 29, pp. 1147–1154.

Study of the Thermophysical Properties of a Multilayered Impacted Cork-based Material by Infrared Thermography

Célia Sanz[1], Thomas Lahens[2], Jean-Christophe Batsale[2], Théo Chavatte[1], Fabrizio Sarasini[2], Sommier Alain[1], Stefeno Sfarra[4]

[1]Univ. Bordeaux, CNRS, Bordeaux INP, I2M, UMR 5295, F-33400, Talence, France

[2]Arts et Métiers Inst. Technology, CNRS, Bordeaux INP F-33400 Talence, France

[3]Department of Chemical Engineering Materials Environment & UDR INSTM, Sapienza University of Rome I-00184 Rome, Italy

[4]Dept. Industrial and Information Eng. Econ., Uni. L'Aquila, L'Aquila, I-67100, Italy

celia.sanz@u-bordeaux.fr, thomas.lahens@u-bordeaux.fr, theo.chavatte@u-bordeaux.fr, fabrizio.sarasini@uniroma1.it

Cork is used in various applications, ranging from the automotive to the aerospace industry. Indeed, it has very good energy absorption ability, interesting thermal and acoustic isolation properties and it is lightweight. Sandwich-structured composite materials using cork as a core are being studied because they combine the qualities of both components. Therefore, these materials have good energy absorption while being strong and stiff in bending. In addition, because cork comes from a renewable resource, their environmental impact is reduced compared to synthetic materials. Due to their inherent properties, cork-based composite materials are good candidates for impact applications. The thermal property distributions of such shock-impacted materials are generally strongly correlated with their structural properties. It is then proposed here to use active thermography to evaluate the response field to impact loading of a thick cork sample sandwiched between two composite skins reinforced with natural fibres. The in-depth and in-plane properties of the sample are being investigated to assess the damage in the impacted region. The methods for characterizing the in-depth properties consist in heating uniformly the front face of the sample and analysing the front face response by thermography. Due to the multiscale nature of the sample, several types of thermal excitations (uniform flash or step) are proposed. The flash method provides information on the outermost thin composite layer (i.e., the external skin). The step heating method provides information on the effusivity of the thick cork layer (i.e., the internal core) and the thermal resistance of the composite skin. The analysis of in-plane properties requires non-uniform heating on the front face (either by moving a laser spot or by patterns of various shapes). We can then detect cracks in the thin layer opening onto the front face and attempt to estimate their in-plane resistance. Details on methods and results related to healthy and damaged parts of the sample will be given.

LES Modeling of Free Convection Heat Transfer from Horizontal Cylinder

Alexey Noskov, Vasiliy Oskirko, Roman Zaharov, Dmitry Oleksyuk

National Research Centre "Kurchatov Institute", 105005 Moscow, Russia

noskov_as@nrcki.ru, oleksyuk_da@nrcki.ru

The study of horizontal cylinder cooling under free convection is important in many engineering problems, including nuclear power engineering, for example, cooling containers for spent nuclear fuel during storage or transportation. This paper presents the results of LES modeling of horizontal cylinder cooling under free convection in water. As a result of the calculation, the values of the Nu number and the temperature on the cylinder surface were determined. The coolant velocity fields were also obtained. The calculation results show good agreement with the experimental study results.

Discrete Element Modeling of Heat Transfer in Active Zone of Nuclear Reactor HTR-PM with Advanced Radiative Model and Nonuniform Burnout of TRISO Pebble Fuel

Andrei Malinouski

a_malin@hmti.ac.by

Calculating heat transfer and temperature distribution in granular bed implies accounting for many heat transfer mechanisms, In high risk applications, like nuclear reactors, there are non-trivial conditions that challenges for accuracy of prediction. High temperature in bed, high power density, and potentially stochastic distribution between nuclear fuel elements makes it important to look into deviations in temperature on the scale of individual fuel elements several centimeters in radius, in case of pebble bed reactors. One of the key advantages of pebble bed design is claimed inherent safety in loss of coolant incident. We use discrete element method, that calculates parameters like temperature and position of every pebble, to obtain reliable temperature distribution. Our contribution consists of more robust and accurate view-factor based model for radiative heat transfer in bed and imitating distribution of burnout in pebble bed fuel to examine non-uniformity of temperature in nominal regime and loss-of-coolant failure. We found that for HTR-PM design maximal temperature in loss of coolant failure may vary by 80 K between pebbles, but stays within safety margins for TRISO fuel. We propose to use proposed method for applications in high-temperature heat transfer in dense granular media.

Performance Analysis of Fixed-Bed and Gyroid-Based Regenerative Heat Exchangers

Ahmet Kasidecioglu, Altug Melik Basol, Ozgur Ertunc

Fluid Dynamics and Spray Laboratory, Mechanical Engineering Department

Özyeğin University İstanbul, Türkiye

ahmet.kasidecioglu@ozu.edu.tr, altug.basol@ozyegin.edu.tr, ozgur.ertunc@ozyegin.edu.tr

This study evaluates the thermal performance of fixed-bed and gyroid-based regenerative heat exchangers designed for gas heat recovery applications with identical porosity and thermal storage material mass. The investigation examines critical thermal performance parameters, including regenerator effectiveness, dimensionless length, utilization factor, heat transfer coefficients, and flow dynamics. Conjugate heat transfer along with Reynolds Averaged Navier Stokes equations-based flow simulations are employed to analyze the influence of geometric configurations and flow conditions on key thermal characteristics and overall energy regeneration. Numerical findings are systematically compared with experimental data from an independent study to validate the results. The analysis aims to assess whether gyroid-based structures, with their intricate channel geometries and enhanced fluid mixing, can be a candidate for replacing or complementing the industry standard of packed-bed designs. Their feasibility and effectiveness are explored, offering valuable insights for optimizing regenerators to enhance efficiency in industrial heat recovery systems.

Session: S5-D

Paper ID: 139

Heat Transfer in Falling Films Over Inclined Walls: A Study of Smooth vs. Wavy Film Surfaces

Yuanxiang Chen, Assensi Oliva Llena, Yue Liu, Jesus Castro Gonzalez

Centre Tecnologic de Transferencia de Calor (CTTC), Universitat Politecnica de Catalunya-BarcelonaTech (UPC) ESEIAAT Terrassa Barcelona, Spain

asensio.oliva@upc.edu, jesus.castro@upc.edu

This paper presents a numerical simulation of heat transfer in falling film flow over inclined walls. Falling film heat exchangers are critical in various industrial applications, including nuclear reactors, cooling towers, and chemical processing. The inherent instability of falling films generates surface waves and, in some cases, separation vortices, potentially enhancing heat transfer between the film surface and the wall. This research investigates the impact of wavy surface features on the thermal performance of falling film heat exchangers. The simulations were conducted using the OpenFOAM solver interFoam, modified to implement a temperature equation. The modified solver was validated against experimental results and agreed well with empirical correlations. Two scenarios were examined: one with an artificially smoothed film surface to force undisturbed flow downstream and another where artificial disturbances were imposed at the inlet to trigger interfacial waves. The results reveal the influence of surface waves on flow dynamics and heat transfer within thin liquid films.

Aerodynamic Analysis of Wind Turbine Airfoils with a Nested Krylov Subspace Solver Implementation in SU2

Ezgi Orbay Akcengiz, Nilay Sezer Uzol

Aerospace Eng. Depertment, Middle East Technical University 06800 Ankara, Türkiye

eorbay@metu.edu.tr, nuzol@metu.edu.tr

The aerodynamic design of wind turbine rotors depends on accurate lift and drag polars over a wide range of Reynolds numbers, which are important for optimizing blade performance. Modern wind turbine blades, with non-linearly twisted and tapered profiles, operate under flow conditions characterized by low Mach numbers and Reynolds numbers ranging from 1 to 15 million. They consist of a family of airfoils distributed along the rotor radius, each with different aerodynamic characteristics. These variations significantly influence the blade's overall performance and need accurate and efficient characterization of each airfoil. Due to the limited availability and high cost of wind tunnel experiments, especially for high Reynolds number flows and complex phenomena such as stall and post-stall behaviors, Computational Fluid Dynamics (CFD) plays an important role in airfoil characterization. A critical aspect of CFD simulations is the computational cost associated with solving the governing equations. In steady-state simulations, a significant portion of the execution time is consumed by the linear solvers used to resolve the system of equations (Economon, 2016). Krylov subspace iterative methods are one of the most common methods to solve linear systems for their ability to increase convergence rate in large-scale simulations. This study aims to improve the efficiency of CFD simulations by implementing a hybrid Krylov subspace iterative method within the SU2 solver. The Flexible GMRES (Saad, 1993) method is coupled with BiCGSTAB as a variable preconditioner, further preconditioned by LU-SGS, to improve convergence rates for steady-state simulations governed by the RANS equations. The methodology is validated on the DU00-W-212 airfoil under steady-state conditions at $Re=3\times10^6$ and M=0.141, across various angles of attack. Preliminary results show that the proposed nested FGMRES/BiCGSTAB method achieves faster convergence, reducing the required iterations by approximately 50% compared to standard FGMRES. Notably, the method handles stall and post-stall conditions more efficiently, where achieving convergence is more challenging due to flow unsteadiness. In the final paper, the aerodynamic performance of the airfoils used in the NREL 5-MW reference wind turbine (Jonkman, 2009) will be investigated using the proposed methodology, aiming to provide detailed understanding and obtaining fast and accurate airfoil database generation for accurate blade design.

Derivation of a roughness model for urban areas by means of detailed CFD simulation

Michael Vögtle, Rainer Stauch,Hermann Knaus

Machines and Systems, Hochschule Esslingen

University of Applied Sciences 73728 Esslingen am Neckar, Germany

michael.voegtle@hs-esslingen.de, rainer.stauch@hs-esslingen.de

Hermann.Knaus@hs-esslingen.de

The site assessment of wind turbines in complex terrain is becoming increasingly important. Accurate site assessments for wind turbines in complex terrains are crucial for optimizing renewable energy efficiency. However, conducting meteorological measurements is challenging due to their time-intensive and costly nature. Computational Fluid Dynamics (CFD) simulations with appropriate models in the meso-micro scale offer a viable alternative. Common dimensions of the computational domain for simulation models of site assessment studies are a length and width of approximately 10 km. Orography is described by highly resolved digital height models and the topography is captured by digital land use data. At these scales it is no longer possible to resolve the topography in detail. While established canopy models exist for forests, there is a lack of standardized approaches for urban areas to account for their complex influence on wind flow. This gap is critical as urban areas increasingly affect wind turbine site assessments, driving the motivation for this study. The objective of this study is to develop a roughness model for urban areas. Such a roughness model could be incorporate into simulation models for site assessments, where the influence of surrounding urban areas is relevant but could not be resolved in detail. With the help of the roughness model, their influence can be considered without the necessity for detailed geometric modelling. This allows new urban geometries to be taken into account without the need for extensive computational resources. The process created should facilitate the future consideration of urban areas in larger simulation models at minimal calculation times. The relevant effects and input parameters for the roughness model have to be analysed in a detailed simulation and transferred to the model for urban areas. Since model parameters of urban areas are not yet known, detailed micro-scale models are created based on the meso-scale initial model. The process of determining the parameters for the roughness model and optimising them to ensure the quality of the model is the core of this study. The geometry of the detailed simulation model includes real building geometries based on a high-resolution 3D building model. The reason for the use of real

building geometries are the avoidance of numerical artefacts due to the use of generic equidistant roughness elements and the possibility of comparison with real measurement data from another research project. In order to be able to investigate the roughness effect of the interfering bodies separately from that of the orography, the latter was modelled as a plane surface. A subsequent consideration of the orography by superposition is feasible. The dimensions of the micro-scale model are approximately 1.5 km by 1 km with a height of 0.15 km. A logarithmic wind profile is defined for the velocity at the inlet boundary to provide a fully developed inflow condition. The key aspect of the development of the roughness model is the determination of the roughness length z_0. From experimental measurements, this roughness length is calculated from measurement data at a certain height. In the simulation, the post-processing of the flow field in the entire computational domain allows the determination of the roughness length. Therefore, the reference height z for calculating the roughness length z0 has to be determined by the analysis of the flow field:

$$z_0 = \frac{z_{ref}}{\exp\left(\frac{\kappa \cdot \bar{u}(z_{ref})}{u_\star}\right)}$$

where κ is the von Kármán's constant and z is the reference height above ground. $u_\star$ is the shear stress velocity, for which two different calculation approaches are compared in this study. Another approach for determining the roughness length z_0 is based on the displacement height. The three different approaches to determining the roughness length z_0 are evaluated and compared. The purpose of the roughness model is to mimic the influence of the urban area on the flow. This includes pressure drop, flow deceleration and production terms for turbulent quantities. The pressure drop was analysed between the beginning and end of the urban area and also over its height. The post-processing is applied to the velocity distribution and the turbulent kinetic energy (TKE). In order to consider the inhomogeneity of the urban area within the roughness model, the urban area is divided in zones by an equidistant grid. The grid must be large enough to avoid effects of individual roughness elements, but fine enough to represent the macroscopic structure of the urban area. Therefore, the grid size is based on the average building size and the building density. After sampling, local roughness values are calculated for each grid cell. The obtained results allow to create structured models which consider the inhomogeneity of the urban area. Furthermore, different inflow directions were analysed. The anisotropy of the urban area and its influence to the roughness quantities are investigated. Particular attention is paid to the vortex formations in the wake of buildings. Initial evaluations indicate a substantial influence. The derived roughness model helps to improve the simulation-based process of site assessment by reducing computational effort while considering the influence of urban areas. Thus, it supports the efficient deployment of wind energy sites next to urban areas, contributing to further development of sustainable energy solutions.

Heat transfer coefficient between spherical particles in low-conducting fluid

Andrei Malinouski, Oscar Rabinovich[1], Heorhi Barakhouski[2]

[1]A.V. Luikov Heat and Mass Transfer Institute of the National Academy of Sciences of Belarus 220072 Brovki, Minsk, Belarus

[2]Physics Faculty, Belarusian State University 220030 Minsk, Belarus

a_malin@hmti.ac.by, orabi@hmti.ac.by

Prediction of effective thermal conductivity of granular materials is important in many applications, from thermal management in electronics to celestial soils. Discrete elements method (DEM) is the most advanced approach, able to capture features like effects of mechanical load, mixtures of different materials and so on. Pivotal for this approach is knowing heat transfer coefficient between adjacent particles. However, most of novel articles and computational codes use simplified formula to predict conductive heat transfer coefficient. The formula is accurate only for solid materials which are simultaneously of high thermal conductivity relative to fluid and of high rigidity, which is not always the case. In this work, numerical modeling of accurate conductive heat transfer coefficient between elastic spherical particles is conducted by both means of finite volume discretization and numerical solution of integral equations. The resulting formula explicitly presents heat transfer coefficient in a form suitable to use in DEM codes. Non-dimensional approach allows to formulate generalized correlation taking into account thermal transfer along particle contact and/or through fluid-filled gap between particles. Effect of thermal flux density dependence on fluid pressure for gap sizes approaching mean free path in gas is also accounted for. Unlike in previous works, correlations are suitable for solid materials with moderate or low thermal conductivity, and are presented as explicit formulas and are convenient to use in DEM calculations for heat transfer in packed and fluidised beds.

Numerical Study of an Industrial Scale Continuous Container Glass Annealing Furnace

Hazar Şişik, Oğuzhan Işık, Gönenç Can Altun, Ersin Yıldız, Altuğ Melik Başol
Mechanical Engineering Department, Özyeğin University 34794 İstanbul, Türkiye
hazar.sisik@ozu.edu.tr, altug.basol@ozyegin.edu.tr

Annealing is a heat treatment process used in glass manufacturing. It is used to relieve the residual stresses that have formed during the forming process due to rapid cooling of glass. In container glass manufacturing annealing process is carried out in continuous annealing furnaces. Heat transfer inside the furnace is inherently a transient process where glass is brought to the furnace to heat it up to the required annealing temperature and cooled back down in a carefully adjusted rate that would not cause residual thermal stresses formation in the material. The transient nature of the process requires careful adjustment of the process parameters that would result in the required heating and cooling rates in the furnace. In this regard numerical models and simulations can help find the necessary process parameters depending on the container glass shape. In this study the heat treatment process of a container glass geometry in a real continuous annealing furnace in the manufacturing plant of Şişecam in Mersin, Türkiye is numerically studied. Furnace operating parameters were derived from the actual operation and imposed as boundary conditions in the thermal furnace model. The model is based on two different solvers which operate in an iterative manner. One of the solvers treats the heat transfer in the furnace domain and the other solver calculates the glass temperatures while they are moving on the conveyor belt. The radiative heat transfer inside the furnace is solved using an in-house developed Monte Carlo ray tracing-based surface-to-surface radiation model. The radiation heat transfer is solved in full geometrical detail with all the 10,000 glass objects inside the furnace. The convective heat transfer to glass is modelled using a porosity-based approach. Simulations results were validated with the real zone temperature data taken from the furnace and glass temperatures at the furnace exit. Finally, simulations with two different conveyor speeds were conducted and the resulting glass temperatures along the furnace were compared and implications on the process quality were discussed.

Finite Element Analysis of Flow and Heat Transfer of an incompressible Non-Newtonian Fluid in a Porous Cavity

Razi Khan[1], Jorge Tiago[2]

[1]CEMAT, Instituto Superior Técnico, Av. Rovisco Pais 1, 1049-001 Lisboa, Portugal

[2]Department of Mathematics and CEMAT, Instituto Superior Técnico, Av. Rovisco Pais 1
1049-001 Lisboa, Portugal

razi.khan@tecnico.ulisboa.pt, jorge.tiago@tecnico.ulisboa.pt

We investigate the heat and mass transfer of an unsteady incompressible non-Newtonian Powell-Eyring fluid flow in a permeable porous cavity. The semi-implicit scheme in conjunction with finite element method is used to solve the non-dimensional equations. The buoyancy force variations and the influence of Darcy number on heat and mass transfer is thoroughly addressed. Attention is focused on how the presence and absence of Darcy number influence the flow and heat transfer within a cavity filled with non-Newtonian fluid. We analyze the impact of flow parameters involved specifically Darcy number, Renolds number and Grashof number on velocity, temperature and heat transfer rate. We also focus our attention on analyzing the Nusselt and average Nusselt numbers.

Investigation on the Heat Transfer Process of Silica-gel Particles inside a Cylindrical Reactor

Yue Liu[1],Jian Zheng[2], Carlos David Pérez-Segarra[1], Yuanxiang Chen[1], Jesús Castro[1]

[1]Centre Tecnologic de Transferencia de Calor (CTTC), Universitat Politecnica de Catalunya-Barcelona Tech (UPC), ESEIAAT 08222 Terrassa (Barcelona), Spain

[2]GD Midea Heating&Ventilating Equipment Co., Foshan,528311 Guangdong, China

cdavid.perez.segarra@upc.edu, yuanxiang.chen@upc.edu, jesus.castro@upc.edu

Adsorption refrigeration technology has attracted significant global interest since it can utilize low-grade energy sources, such as solar heat and industrial exhaust heat. Additionally, it employs environmentally friendly refrigerants with low global warming potential (GWP) value and ozone depletion potential (ODP) value. In an adsorption system, the reactor is the most critical component due to its dominant impact on the system's performance. This work presents a preliminary investigation on the pure heat transfer process of silica gel packed bed inside a cylindrical reactor, emphasizing the measurement of the heat transfer coefficient (HTC) between the silica gel packed bed and the solid heat exchanger with different heating source temperatures. Moreover, the flow characteristics of the secondary heating water in the annulus reactor were also explored. An 0-D model was developed to design the experimental test campaign, which uses empirical correlations and is parametrized with the data of the experimental data bench. The results indicate that when the temperatures of the heating water range from 60 to 80°C, the convective HTC of the secondary water varies from 300 to 500 W/(m2 K), and the HTC between the silica gel packed bed and the solid heat exchanger varies from 15 to 30 W/(m2 K), which is smaller than those values reported and employed by other researchers. Based on the obtained experimental data, the numerical results of the 0-D model reached a good agreement with the experimental data. The results in this work will lay the groundwork for further study of the adsorption characteristics of silica gel-water working pair in the cylindrical reactor and help optimize the design of the packed bed reactor.

Edney Shock Interactions in High Enthalpy Rarefied Gas Flows

Elif Fatma Avcılar, Bayram Çelik
Dept. Astronautical Engineering, İstanbul Technical University 34469 İstanbul, Türkiye
avcilar20@itu.edu.tr, celikbay@itu.edu.tr

This study aims to computationally model and analyze the Edney shock interactions by using an open-source Navier-Stokes solver. The previous numerical studies on such interactions at high Mach numbers that are available in the literature have generally focused on the effects of rarefaction without considering chemical reactions. Apart from these studies, this study incorporates reactions to achieve a more realistic representation of high-enthalpy flows. The obtained results are compared to experimental results of Moss, where a Mach 10 flow with a Reynolds number of (1.66×10^5) flow is generated in the ONERA R5Ch low-density wind tunnel. For our numerical model, a cylinder is used to generate the primary bow shock of the system, along with a relatively weak external oblique shock that is generated by a wedge with a 20° deflection angle. In order to model the interactions in both continuum and slip regimes, we varied the Knudsen number to be smaller than (10^{-1}). Our computational result will contribute to the literature by revealing qualitative and quantitative comparisons of the reacting flows in continuum and slip regimes.

Optimisation of Thermal Energy Storage Systems for Industrial Heating Applications

Oriol Sanmarti, Santiago Torras, Carlos David Pérez-Segarra, Jordi Vera Fernandez

Centre Tecnologic de Transferencia de Calor (CTTC), Universitat Politecnica de Catalunya-Barcelona Tech (UPC), ESEIAAT 08222 Terrassa (Barcelona), Spain

oriol.sanmarti@upc.edu, santiago.torras@upc.edu, cdavid.perez.segarra@upc.edu, jordi.vera.fernandez@upc.edu

With advancements in technology, it has become possible to monitor various variables in our environment with wireless sensors easily. One of the downsides of these sensors is that they require a battery to operate, and you need to replace the battery when it depletes. Depending on the situation, replacing the sensors' batteries is not feasible. Thermoelectric generators (TEGs) can generate electricity from waste heat with no moving parts, and can be an available solution for the battery replacement problem. In this research, we aim to conduct a numerical study to assess the power generation of TEGs and pyramid-schemed TEGs for electrical sensors used in IoT. Numerical analysis was performed using COMSOL Multiphysics Simulation. The simulation results are verified by comparing our TEG models with the results from the literature.

Towards understanding ion-specific adsorption and interfacial thermodynamics at water-mica interfaces

Alper Tunga Çelebi, Matteo Olgiati, Florian Altmann, Laura Mears, Markus Valtiner

Institute of Applied Physics, Vienna University of Technology
Wiedner Hauptstrasse 8-10 1040 Vienna, Austria

celebi@iap.tuwien.ac.at

Adsorption of ions at the solid-liquid interface plays a critical role in various natural processes and applications, including metal corrosion, electrochemical energy storage, and clay swelling. In these processes, the electrolyte composition significantly impacts the structure and behavior of ions and water at the interface. By performing molecular dynamics (MD) simulations, we explore ion adsorption, hydration, and electric double layer (EDL) structures in aqueous electrolytes containing Cs+, Li+, and Ca+2 ions confined between two negatively charged mica surfaces. Our simulation results reveal that Cs+ ions exhibit the strongest screening effect on the mica surface compared to Li+ and Ca+2 ions at the same concentration, indicating a stronger ion adsorption. Interestingly, the number of adsorbed Cs+ ions exceeds the surface charge of mica. This refers to a phenomenon called as "overscreening". As a result, the surface becomes positively charged, and the diffuse layer of EDL becomes co-ion dominated. However, this is not the case for Li+ and Ca+2 which they attach less strongly to the surface and do not overcharge the mica. Additionally, high-resolution atomic force microscopy (AFM) imaging which enables to visualize the lateral distribution of individual mono- and multi-valent ions on the surface of muscovite mica. AFM imaging not only resolves the crystal structure of muscovite mica in aqueous solutions but also captures the transient distribution of adsorbed ions from salt-rich solutions across varying concentrations. The ion screening behavior at the interface strongly influences water structuring. Water molecules can penetrate between the adsorbed Li+ layer and the surface, strongly hydrating Li+ ions whereas this is not possible for Cs+. This behavior is controlled by the entropic contribution of the water and ionic species. Understanding the competition as a function of type and concentration of ions allows to unravel the interfacial thermodynamics directly from MD and AFM data.

Session: S3-A

Paper ID: 150

CFD and Energy Analysis of Photovoltaic-Supported Electrically Driven Heat Exchangers

Jordi Vera Fernandez, Oriol Sanmarti, Santiago Torras, Carlos David Pérez-Segarra
Centre Tecnologic de Transferencia de Calor (CTTC), Universitat Politecnica de Catalunya-Barcelona Tech (UPC), ESEIAAT 08222 Terrassa (Barcelona), Spain
jordi.vera.fernandez@upc.edu, oriol.sanmarti@upc.edu
santiago.torras@upc.edu, cdavid.perez.segarra@upc.edu

The integration of renewable energy sources into industrial heating systems is a critical step toward achieving sustainable energy solutions. This study focuses on the analysis of an electrically powered heat exchanger supplied by photovoltaic panels, aimed at enhancing energy efficiency in industrial heating applications. A detailed computational fluid dynamics (CFD) analysis will be conducted to evaluate the thermal and fluid dynamic performance of the heat exchanger under various operating conditions. The study will also assess the energy contribution of photovoltaic panels, including their efficiency, output variability, and the overall impact on the thermal energy supplied to the system.

Impact of Shock Boundary Layer Interaction on Swirl Characteristics in Supersonic Intakes with Bleed System

Muhammed Enes Özcan, Nilay Sezer Uzol

Aerospace Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

nuzol@metu.edu.tr

The aerodynamic performance and stability of external compression intakes are strongly influenced by swirl and its distribution across the engine face. The two primary parameters defining flow quality are the total pressure profile and the angular properties of the velocity field, quantified by the swirl angle variation. This study investigates the effects of a bleed system on swirl characteristics in a supersonic intake operating at freestream Mach numbers of 1.6, 1.8, and 1.9, with engine face Mach numbers (MEF) ranging from 0.3 to 0.5. Computational Fluid Dynamics (CFD) simulations are performed for evaluation of swirl angle distributions under both bleed-on and bleed-off conditions using a custom-designed 5x8 rake by using ANSYS Fluent. The results demonstrate that activating the bleed system significantly reduces peak swirl angles and modifies swirl patterns. For MEF = 0.3, the maximum swirl angle decreases from 6.8° (without bleed) to 1.2° (with bleed). At freestream Mach 1.8 and MEF 0.3, swirl directivity without bleed is nearly zero to -0.2 across all rings, indicating a twin swirl pattern. With the bleed system, swirl directivity increases to approximately +1 in the first four rings, while the last ring remains at -0.2. This transformation converts the disruptive twin swirl pattern into a positive bulk swirl, enhancing flow stability and angular momentum uniformity. The findings highlight the critical role of bleed mechanisms in improving aerodynamic stability and extending operational ranges of supersonic intakes. By mitigating adverse swirl effects, bleed systems optimize performance and flow uniformity, crucial for high-speed applications.

A new two-fluid numerical method for simulating mass transfer in immiscible two-phase flows: validation and comparison with single-fluid results

Nour El Houda Djaballah, Benoit Trouette, Savaş-Can Selçuk

Eric Chenier, Stephane Vincent

MSME, Gustave Eiffel University, CNRS UMR 8208
Paris-East Créteil University F-77454 Marne-La-Vallée, France

benoit.trouette@univ-eiffel.fr, can.selcuk@univ-eiffel.fr, eric.chenier@univ-eiffel.fr

stephane.vincent@univ-eiffel.fr

Bubbles (or droplets) interactions with an immiscible and continuous phase are prevalent in several industrial applications. Their efficient heat- and mass- transfer properties are of interest and are mainly due to the generation of large interfacial areas. In this paper, we investigate the mass transfer of a two-phase bubbly flow. The major challenge raised by such a flow comes from the abrupt discontinuity of the species concentration at the bubble-liquid interface. Appropriate numerical models are therefore required to capture such features accurately and efficiently. Two categories of numerical methods are considered for the advection-diffusion of species concentration. First, we used a two-fluid formulation that considers each phase separately. This approach employs an extended version of the so-called Ghost Fluid Method that imposes the concentration jump and the flux-continuity across the interface. The second approach is based on a one-fluid formulation, which considers a mixture at the interface and incorporates an additional and non-trivial term to account for the interfacial diffusion. For both approaches, an original advection scheme based on a geometrical transport is proposed to ensure the consistent transport of the interface. The accuracy of each method is then studied via a set of direct numerical simulations of mass transfer between a bubble and a surrounding liquid phase. We consider both static and moving bubbles in a deformable and non-deformable scenarios. We perform convergence studies against analytical solutions by studying the bubble's interface topology, the surrounding flow-dynamic and the local mass-transfer rate through the Sherwood number.

An Investigation to Enhance the Mixing Efficiency of SAR Micromixer with Obstacles

Giray Yüksel, Ayşe İrem Avlık, Emine Yegan Erdem

Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

giray.yuksel@ug.bilkent.edu.tr, irem.avlik@ug.bilkent.edu.tr, yeganerdem@bilkent.edu.tr

With advancements in technology, it has become possible to monitor various variables in our environment with wireless sensors easily. One of the downsides of these sensors is that they require a battery to operate, and you need to replace the battery when it depletes. Depending on the situation, replacing the sensors' batteries is not feasible. Thermoelectric generators (TEGs) can generate electricity from waste heat with no moving parts, and can be an available solution for the battery replacement problem. In this research, we aim to conduct a numerical study to assess the power generation of TEGs and pyramid-schemed TEGs for electrical sensors used in IoT. Numerical analysis was performed using COMSOL Multiphysics Simulation. The simulation results are verified by comparing our TEG models with the results from the literature.

Experimental analysis and dynamic energy model of a Gas Liquid Energy Storage (GLES) for electric and thermal storage

Miriam Di Matteo

Dept. Mech. & Aerospace Eng., University of Rome "La Sapienza" 00185 Rome, Italy

miriam.dimatteo@uniroma1.it

Energy storage systems have been identified as a pivotal strategy for integrating renewable sources, thereby ensuring energy stability and flexibility of the electricity grid. The present work analyses a Gas Liquid Energy Storage (GLES) prototype, an experimental mechanical storage system, by performing on-field tests and building-up a dynamic simulation model. In detail, renewable electricity excess drives a volumetric gear pump for oleo dynamic compression of gaseous N_2 within the cylinder. Similarly, in the expansion phase, the N_2 pressure gradient pushes the oil though the pump impellers, generating electricity again. Contrary to conventional Compressed Air Energy Storage systems (CAES), this technology is based on a closed thermodynamic system. The mathematical model implemented in MATLAB/Simulink, has been validated by an experimental campaign. This latter consisted in temperature, pressure and volume measurements for three different oil flow rates (namely 1.8, 3.6 and 5 l/min) and two pre-charge pressures (i.e. 2.65 bar and 25.2 bar). As a result, absolute errors of ±1.2 °C and ±1.4 bar for temperature and pressure have been registered, respectively. Moreover, an energy density greater than 0.32 kWh/m^3 has been achieved, and round-trip efficiency, mechanical work and heat have been used as indicators for the performance assessment. In the end, the higher the oil flow rate, the lower the heat release is. That result suggests the potential exploitation of the GLES for both electrical and thermal storage purposes. This work also establishes a foundation for further experimental analysis involving the replacement of N_2 with CO_2 and Kr, to increase the stored electricity, due to their higher molecular weight.

A coupled model for drug release from a non-swellable microsphere to a surrounding tissue

Giampaolo D'Alessandro

University of Southampton, University Road, Southampton SO17 1BJ United Kingdom

dales@soton.ac.uk

Use of polymeric microspheres is an effective way to obtain a controlled release of drugs. The problem here considered regards a non-swellable microsphere loaded uniformly with a drug and surrounding by a semi-infinite tissue. Inside the microsphere the drug is dissolved from a solid phase (the matrix) to a liquid phase filling the matrix pores. Therefore, a diffusion-dissolution model known as Harland model is considered inside the microsphere. Moreover, inside the tissue the drug exists in both a bounded and unbounded phase through binding-unbinding reactions. Consequently, the equations related to the two phases inside the tissue lead to a coupled system. Additionally, due to the internal boundary conditions the equations of the two layers are coupled too. Anyway, using an uncoupling procedure available in the literature the tissue equations can be uncoupled revealing the dual phase lag behavior of the drug diffusion in this layer. With the same approach the internal boundary conditions can be also uncoupled allowing the equations of the two layers to be solved analytically in different steps.

Direct Numerical Simulation of a Hypersonic Transitional Boundary Layer in Chemical Non-equilibrium: Effect of Wall State

Marco Fratini, Giacomo Della Posta, Matteo Bernardini

Dept. Mech. & Aerospace Eng., University of Rome "La Sapienza" 00185 Rome, Italy

marco.fratini@uniroma1.it, giacomo.dellaposta@uniroma1.it, matteo.bernardini@uniroma1.it

This study presents a direct numerical simulation (DNS) of a transitional hypersonic boundary layer in chemical non-equilibrium, focusing on the influence of wall conditions. The simulation investigates a Mach 8 flow over a flat plate, representative of the post-shock region of a hypersonic wedge traveling at Mach 21. The wall is modeled as cold and isothermal with two catalytic conditions: non-catalytic and fully catalytic. The stagnation enthalpy is high enough to trigger molecular oxygen dissociation in the boundary layer's high-temperature region caused by aerodynamic heating, making chemical non-equilibrium effects significant. Thermodynamic equilibrium is assumed for the vibrational degree of freedom, which is a reasonable approximation due to relatively high air density characteristic of hypersonic cruise flight at low altitudes, which makes molecules collision more frequent and therefore decreases vibrational relaxation characteristic times. Laminar-turbulent transition is achieved through a suction and blowing boundary condition applied to a narrow strip of the wall. The objective of this numerical study is to explore how wall catalyticity affects the development and transition of the boundary layer, as well as the stream-wise evolution of wall shear stress and wall heat flux. High-fidelity data will thus provide foundational insights into the complex interactions between chemical non-equilibrium and boundary layer dynamics in hypersonic flows.

Thermoelectric Cooler Design without Using Cold And Hot Face Temperatures

Ahmet Bahadır Dağlı, Necip Berker Üner
Chemical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye
nuner@metu.edu.tr

Peltier modules are now easily accessible materials for accomplishing thermoelectric cooling for electronics, condensers and essentially any small surface or volume without using moving parts or a refrigerant. Common bismuth telluride-based modules are well characterized through experiments in which face temperatures were measured. In the case of applications, measurement of face temperatures lead to additional costs and custom manufacturing to place thin temperature sensors with small form factors, especially for the hot side on which heat dissipation must be managed at the same time. Moreover, in some specific applications, such as liquid cooling, temperatures are usually measured away from the cold surface and rather in the liquid bulk. Interestingly, a design strategy for predicting the cooling performance of Peltier modules without continuous measurement of hot and cold face temperature is lacking. Ideal calculations should be essentially based on the cooling performance of the hot side and Peltier circuit characteristics. This report presents the modeling framework accompanied by supporting experiments to predict cooling performance based on performance metrics of the module, structural parameters and the efficiency of cooling of the hot face. First, a heatsink and fan combination was characterized through steady-state temperature measurements at constant voltage, where hot side temperature is deduced from cold side measurements. These measurements are shown to enable the determination of UAf, the overall heat transfer coefficient multiplied by the effective cooling area of the heatsink with the selected fan. We show that by changing fan speeds, data can be extended to different heatsinks and fans through a generalization that involves fin geometry and efficiency. This framework is then put to use for designing a sub-zero condenser for vapors. A cubical structure was manufactured with four Peltier modules, leading to an ultimate bath temperature at room temperature. By the use of an unsteady-state model, heat leakage from the ambient was characterized. This information was utilized for estimating the actual heat duty of the cooling bath as a function of temperature and condensation performance was measured by removing water vapor in humid stream. We will present the design methodology, along with details related to temperature control, which allow simplified calculation of realistic cooling capacities in various thermoelectric cooling applications.

CFD Coupled With a Lognormal Model for Modeling an Evaporation-Condensation Type Aerosol Generator

Zeynep Birce Reyhan, Necip Berker Üner

Chemical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

zeynep.reyhan@metu.edu.tr, nuner@metu.edu.tr

III-V semiconductors are among the most versatile materials for optoelectronic and photonic applications due to their air stability and tunable direct bandgaps, which span a wide range from 0.35 to 6 eV. Gas-phase synthesis offers a promising approach for producing ligand-free III-V nanocrystals, an essential feature for electronic applications. However, most gas-phase methods rely on vapor-phase precursors, which are expensive, highly toxic, and pyrophoric. An alternative involves generating elemental precursors using an evaporation-condensation generator (ECG), commonly implemented as a tubular furnace. Developing an ECG capable of delivering a stable aerosol output with high mass yield is a critical need. In this study, mathematical modeling and simulations of an ECG were performed by integrating momentum, heat, and mass transport with aerosol dynamics. Antimony, a key component of various III-V compounds, was selected as the test material. Building on experimental results for an ECG operating under reduced pressure, simulations were conducted at 6 Torr across varying gas flow rates using computational fluid dynamics (CFD). Once the transport phenomena were modeled, a detailed lognormal aerosol dynamics model was developed, incorporating terms for advection, surface growth, diffusion, coagulation, and thermophoresis. These terms were integrated into the CFD framework to achieve a comprehensive mathematical representation of aerosol formation. The results, including aerosol mass flow rates and generator yields, demonstrated order-of-magnitude agreement with experimental data. Details of the CFD and lognormal model will be presented, along with results and comparison with experimental data.

Effects of Rarefaction on Thermochemical Non-Equilibrium via Open-Source Software

Muhammet Erdem Akbaş, Bayram Çelik
Dept. Astronautical Engineering, İstanbul Technical University 34469 İstanbul, Türkiye
akbas20@itu.edu.tr, celikbay@itu.edu.tr

The present study focuses on the effects of rarefaction and slip boundary conditions on the flow field analyses conducted with Compressible Navier Stokes Equations with Park's Two Temperature Model on high Mach Number re-entry flow simulations. Calculations are performed using an open-source thermochemical non-equilibrium Navier-Stokes solver, hy2Foam. The first part of the study aims to validate and compare the hy2Foam solver with the literature. Such conditions are important since similar conditions are investigated with proprietary software in the literature, but open-source, multi-specie reacting gas studies are limited. Also, validation of hy2Foam with such complex and rarefied flow physics is not present in the literature. The second part of the study aims to investigate the limitations of Computational Fluid Dynamics at slip regime, the region where the use of Direct Simulation Monte Carlo is computationally expensive due to relatively low mean free path. Although several numerical studies are available in the literature at such conditions, none of them aim to investigate the rarefaction effects on CFD calculations at high altitudes exclusively. Also, such findings in the literature are mostly constructed with proprietary software, but the use of open-source software is of critical importance to enhance global scientific contribution. Such a study will summarize and compare the suitability and limits of hy2Foam.

Development of a novel thermal management system for Li-ion battery using microchannel

Alireza Sheikhi Darani, Mohammad Hassan Saidi

Dept. Mechanical Eng., Sharif University of Technology 14588-89694 Tehran, Iran

saman@sharif.edu

Electric vehicles can help reduce the serious threats of energy crises and environmental pollution. Battery thermal management systems have crucial roles in electric vehicles as keeping the temperature of battery cells at operating temperature has a direct impact on increasing driving range, extending cell lifetime, and improving system safety. In this investigation, a novel thermal management system based on microchannel and phase change materials for 18650 lithium-ion cells is suggested. This system uses the spare space created between cells located in the battery pack to convey the coolant via microchannels parallel to the axis of cylindrical cells. Additionally, the remained void space is filled with phase change material to improve the performance of the system. Not only does this system utilize spare space for cooling but it also, cools the whole battery pack uniformly. The computational model for this system is established to evaluate the performance of this system in various situations.

Thermal Management of a Distributed Heat Load Using Bent Aluminum Axially Grooved Heat Pipes

Ahmet Tahir Kalkışım[1], Tassos G. Karayiannis[3], Ahmet Yavuzdoğan[2], James Tyacke[3]

[1]Electronics and Automation Dept., Gümüşhane University 29100 Gümüşhane, Türkiye

[2]Geodesy Department, Gümüşhane University 29100 Gümüşhane, Türkiye

[3]Mech. & Aerospace Eng. Department, Brunel University, UB8 3PH London, UK

atkalkisim@gumushane.edu.tr, tassos.karayiannis@brunel.ac.uk

yavuzdogan@gumushane.edu.tr , james.tyacke@brunel.ac.uk

Türkiye is one of the countries with the highest potential for geothermal energy systems. Conventional geothermal power generation in Türkiye has already exceeded 1700 MWe. The number of projects and research initiatives in this field is increasing day by day. In recent times, Enhanced and Advanced Geothermal Systems (EGS/AGS) have attracted significant interest from companies and research institutions. These systems require a multidisciplinary approach, involving collaboration among geologists, geophysicists, mechanical engineers, chemical engineers, petroleum engineers, and mining engineers within the same project. To achieve technological advancements in Advanced Geothermal Systems, it is crucial to perform transient thermal analysis of borehole co-axial heat exchangers. This involves accurately determining the heat capacities and conductivities of the surrounding rock. Furthermore, it is necessary to develop realistic models of rock formations and to design industrial materials suitable for these systems. In this study, we analyze the performance of co-axial heat exchangers under different geothermal gradients, pipe materials, and depths, with validations conducted in COMSOL. Following this, a 300 kWe system is optimized for a local area with a geothermal gradient of 70°C, based on basic economic assessments for a well. The numerical analysis has been successfully validated using COMSOL, and the results were utilized in a Python-based optimization framework. This framework allows for the design of an economically feasible co-axial deep well geothermal system.

Thermal Management of a Distributed Heat Load Using Bent Aluminum Axially Grooved Heat Pipes

Ahmet Tahir Kalkışım, Zeynel Öztürk

Anadolu Innovation R&D 61300 Trabzon, Türkiye

tahir@anadoluarge.com.tr, zeynelozturk61@yahoo.com

Due to the climate sensitivity of modern energy policies, the use of many popular refrigerants in automotive and refrigeration systems has been restricted. In recent years, alternative refrigerant studies have focused on options with low Ozone Depletion Potential (ODP) and low Global Warming Potential (GWP) values, such as R152a and R1234yf. The primary goal is to ensure that these refrigerants can act as "drop-in" replacements for current systems and are compatible with existing compressor technologies. It is crucial to recognize that the pressure ratio is one of the most important parameters for evaluating alternative refrigerants. Similarly, the Coefficient of Performance (COP) is as significant as the pressure ratio in determining the efficiency of refrigerants. A novel approach involves using Metal-Organic Frameworks (MOFs) to enhance COP through an injection application to the refrigerants. MOFs are highly effective in trapping molecules under low-pressure conditions, allowing for the storage of a greater quantity of refrigerant at a given pressure. This mechanism increases the enthalpy and improves the cooling performance of compressors. The study, conducted using Material Studio software, investigated the effects of MOFs—specifically FAU-type zeolite and MIL-53(Al)—on enthalpy and COP under varying pressure and temperature conditions in a compressor stage. The results demonstrated that MOFs enhance cooling capacity and efficiency depending on the compressor stage conditions.

Numerical Investigation of Novel Derivatives of Owl-Inspired Airfoils for Low Reynolds Number Applications

Ashraf Omar, Yassine El Qamch, Kenza Bouchaala
Sch. Aerospace & Automotive Eng., Int. Uni. Rabat Sale11 100 Al Jadida, Morocco
ashraf_omar@uir.ac.ma, yassine.elqamch@uir.ac.ma, kenza.bouchaala@uir.ac.ma

The current study focuses on the aerodynamic assessment of novel derivatives of owl-inspired airfoils, developed by introducing increased thickness-to-chord (t/c) ratios while retaining their highly cambered and non-symmetrical characteristics. The primary objective is to evaluate the performance of these modified airfoils under low Reynolds number (Re=23,000) conditions, with an emphasis on capturing the complex dynamics of unsteady flow and laminar separation bubbles (LSB). A comprehensive numerical approach has been adopted, starting with steady k−ω SST simulations to identify limitations in predicting separation and unsteady effects. This is followed by unsteady RANS (URANS) simulations, which are expected to provide deeper insights into the oscillatory behavior of lift coefficient (Cl) and its consistency with the dynamics of separation bubbles. The study further evaluates the effectiveness of various turbulence models in capturing these unsteady flow phenomena, offering a systematic comparison of their predictive capabilities. The novelty of this work lies in the development and analysis of thicker, highly cambered airfoil derivatives inspired by owl airfoils but tailored for specific aerodynamic applications. By using advanced numerical techniques, the research aims to bridge the gap between steady-state limitations and the need for robust unsteady analysis, contributing to the broader understanding of low Reynolds number aerodynamics for bio-inspired and engineered airfoil designs.

2D Simulations on a Flat Plate to Study the Effect of Porosity on Skin Friction Drag Reduction

Ashraf Omar, Wafae Lahmili, Kenza Bouchaala

Sch. Aerospace & Automotive Eng., Int. Uni. Rabat Sale11 100 Al Jadida, Morocco

ashraf_omar@uir.ac.ma, wafae.lahmili@uir.ac.ma, kenza.bouchaala@uir.ac.ma

This study investigates the aerodynamic behavior of flat plates and explores potential strategies for drag reduction through detailed two-dimensional computational simulations. The analysis focuses on distinct flat plate configurations, each designed to address specific aspects of aerodynamic performance. To capture the complex flow characteristics, multiple turbulence models are employed, enabling a comprehensive evaluation of their effectiveness in accurately predicting boundary layer behavior, particularly at low Reynolds numbers. The numerical results are systematically validated against available experimental data to ensure reliability and accuracy. In addition to understanding general aerodynamic behavior, this study delves into the impact of porosity on the performance of flat plates by analyzing configurations with pores. A flat plate containing an array of microholes is examined to assess how key porosity parameters—including microhole diameter, aspect ratio, and depth—affect aerodynamic properties. The investigation focuses on their influence on critical metrics such as the skin friction drag coefficient (Cf) and the total drag coefficient (Cd). The introduction of microholes enables passive flow control mechanisms, which have the potential to reduce drag by modifying boundary layer characteristics and redistributing pressure forces along the plate's surface. By systematically varying the porosity parameters and analyzing their effects, this study aims to identify optimal design configurations for achieving effective drag mitigation.

Mixing Analysis in a Stirred Tank Equipped with Innovative Impeller

Klaudia Zwolińska-Glądys, Anna Młynarczykowska, Marek Jaszczur, Marek Borowski

Facul. Civil Eng. & Resource Management, AGH University of Krakow, Krakow, Poland

kzwolinska@agh.edu.pl, mindziu@agh.edu.pl, jaszczur@agh.edu.pl, borowski@agh.edu.pl

The fluid mixing process plays a crucial role in many industrial processes. Numerous studies provide phenomenological or heuristic descriptions of fluid mixing dynamics. Regardless of the approach, the most important aspect is the optimal design of impeller geometry for specific hydrodynamic conditions, which ensures high mixing efficiency and superior product quality. The mixing process's efficiency depends on many parameters, i.e., mixing phases' viscosity, density of liquids, temperature, system configuration, and, foremost, the impeller shape. In this research, the main goal is to investigate numerically and experimentally the impact of an innovative jet-type impeller on fluid flow mixing phenomena in a cubical tank. The analyzed impeller is designed to mix fluids with a wide range of viscosities while ensuring minimization of the final product degradation and low energy consumption. Mixing intensity and efficiency were assessed through fluid motion analysis and power number evaluation. Flow fields were obtained using Computational Fluid Dynamics (CFD) and Particle Image Velocimetry (PIV) measurements while the power consumption was determined using a precise torque meter. The jet-type impeller demonstrates a low power number across a wide Reynolds number range. After exceeding the critical Reynolds number, a turbulent regime occurs, and the power number stabilizes at a significantly lower level compared to conventional designs.

Efficient Eulerian Interfacial Area Transport for Modeling Droplet Coalescence and Breakup in Complex Flow Systems

Bora Kalpakli, Ozan Köken

ROKETSAN A.Ş., 06780 Ankara, Türkiye

This work presents a method based on the Interfacial Area Transport concept to model how droplets merge (coalesce) and split (break up). Originally developed for nuclear reactor applications, it offers several major benefits: more precise droplet size analysis, faster results, and reduced computational requirements. When compared to traditional Lagrangian (discrete phase) and population balance methods commonly used in commercial software, it can deliver more accurate predictions in specific scenarios, while using far less computing power. Our approach relies on a fully Eulerian particle solver and only needs one extra equation to track how droplet sizes change physically. This leads to a continuous distribution of particle sizes and densities across the domain, making the simulations both faster and more robust. In fact, this method can be hundreds of times faster than comparable Lagrangian models, and memory usage is not a limiting factor. This speed advantage is particularly important for transient processes, such as those in combustion chambers or liquid fuel injection, where large numbers of particles—sometimes in the billions—must be tracked quickly. These scenarios often feature sudden combustion changes and in-cylinder pressure fluctuations. In contrast with population balance methods, there is no need to limit droplet size changes to certain discrete ranges; our method captures realistic, continuous shifts in droplet size. While a passive transition to porous structures is already possible for particle buildup and flow blockage, a model for a full liquid-phase transition under droplet accumulation remains under development. A key example is found in solid-fuel rocket motors, where alumina droplets formed during combustion significantly influence performance, stability, and combustion efficiency. These droplets can combine into larger clusters under turbulent conditions, then later break up near the nozzle throat due to flow instabilities. Because they can represent up to 40% of the mass, neglecting their effects would yield incomplete results. By integrating this method into the CMPS software, we can simulate these internal motor phenomena with high accuracy, offering detailed insights into the essential processes that govern motor performance.

Optimization of Closure Model Coefficients with Bubble-Induced Turbulence for Enhanced Two-Phase Flow Predictions

Özgün Güler

IOG Engineering Inc. 06800 Çankaya, Ankara, Türkiye

ozgun@iog.com.tr

Two-phase bubbly flows exhibit complex turbulence characteristics influenced by interfacial forces such as drag and lift. Accurate modeling of these forces is critical for predicting flow behavior in engineering systems. This study extends conventional drag and lift coefficient models by incorporating additional dimensionless parameters, including Eötvös (Eo), Weber (We), and aspect ratio (E), alongside Reynolds number (Re) and void fraction (α). A hybrid closure approach is proposed for Bubble-Induced Turbulence (BIT), enabling a multi-objective optimization of the coefficients to minimize discrepancies with experimental data. An optimization procedure is performed against a comprehensive dataset from Liu, Serizawa, Hosokawa, and Shawkat experiments. The results demonstrate that a single set of coefficients often fails to accurately capture wall peaking and core peaking void fraction profiles simultaneously, highlighting the need for a more flexible modeling approach. By adopting a grouped coefficient determination strategy, the hybrid BIT model achieves improved agreement with experimental measurements across various flow regimes. This study emphasizes the interaction between small-scale turbulence and interfacial forces. The findings advance our understanding of two-phase flow dynamics and provide a robust modeling framework for engineering applications involving bubbly flows.

Performance Analysis of Plate Heat Exchangers under Dynamic Environmental and Freezing Conditions

Marek Borowski, Klaudia Zwolińska-Glądys, Rafał Łuczak, Piotr Życzkowski

Facul. Civil Eng. & Resource Management, AGH University of Krakow, Krakow, Poland

borowski@agh.edu.pl, kzwolinska@agh.edu.pl, rluczak@agh.edu.pl

piotr.zyczkowski@agh.edu.pl

Heat exchangers are a key element of ventilation, allowing for increased energy efficiency of the entire system. Due to the heat losses associated with the ventilation system, which constitute a significant part of the total heat losses of the building, the selection of appropriate heat recovery, operating also at low temperatures, is crucial. The paper evaluates the performance of the plate heat exchanger operating under dynamic environmental conditions, focusing on freezing scenarios and compliance with ECODESIGN standards. The experimental analysis included tests during which key performance parameters were recorded, including pressure differentials, mass and volume flow values, temperature, and relative humidity. The results showed that freezing conditions affect the operational performance of the heat exchanger, especially in configurations without preheating mechanisms. Pressure differences between supply and discharge flows showed periodic fluctuations, while mass and volume flows showed consistent decreases during freeze-thaw tests. Temperature data underscored the key role of heaters in mitigating extreme temperature drops, reducing the risk of condensation, and ensuring consistent performance. Relative humidity measurements highlighted rapid stabilization after initial drops, correlating with system adaptation to environmental changes. This analysis underscores the importance of experimental protocols for evaluating heat exchanger performance under real-world conditions. These findings provide practical information for optimizing HVAC system designs to improve energy efficiency and reliability, especially in climates prone to extreme temperature fluctuations. The results will allow HVAC designs to be optimized for energy efficiency and reliability, especially in climates prone to extreme temperature fluctuations.

Progress in Turbulent Flow, Heat, and Mass Transfer Modeling in Porous and Hybrid Media

Marcelo de Lemos

Department of Aeronautical Engineering, Instituto Tecnológico de Aeronáutica (ITA), Praça Marechal Eduardo Gomes, 50, Vila das Acácias, 12228-900 São José dos Campos, SP, Brazil

delemose@ita.br

This paper explores cutting-edge techniques for the modeling and simulation of turbulent transport within porous media, which are prevalent in various engineered systems and natural environments involving fluid flow through permeable matrices. These models play a pivotal role in optimizing engineering designs and assessing environmental impacts. The discussion centers on recent advancements in turbulence modeling, emphasizing the significance of time and volume averaging, as well as the double-decomposition approach, to address the intricate interactions in heterogeneous systems. The sequence of applying these averaging techniques is analyzed, yielding distinct sets of governing equations for statistical properties. Consideration is given to thermal non-equilibrium conditions between phases, along with macroscopic buoyancy effects in both mean and turbulent regimes. The study also examines hybrid systems combining porous and clear fluid regions, highlighting challenges at their interfaces and numerical strategies for their resolution. Applications span combustion in porous frameworks, moving bed technologies, heat exchange in porous cavities, and double-diffusion phenomena. Particular focus is placed on modeling and simulating thermite reactions, which exemplify chemical processes in heterogeneous media. These simulations reveal dynamic thermal behaviors in hybrid systems, essential for innovative uses such as energy storage and abandonment solutions in the oil and gas sector.

Discrete Green's Function Method for Laplace Equation with Nonlinear Boundary Conditions

Barış Çetin[1], Barbaros Cetin[2], Kevin D. Cole[3]

[1]FNSS Savunma Sistemleri A.Ş., R&D Center 06830 Ankara, Türkiye

[2]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[3]Dept. Mech. & Materials Eng., University of Nebraska–Lincoln, Lincoln, NE 68588 USA

cetin.baris@fnss.com.tr, agunay@bilkent.edu.tr, kcole1@unl.edu

Cathodic protection (CP) is a vital technique to prevent corrosion in submerged or intermittently exposed metallic structures. While numerical methods such as finite element and boundary element methods are commonly used to model CP systems, they often involve high computational costs, especially under nonlinear boundary conditions. This paper introduces a semi-analytical approach based on the Discrete Green's Function Method (DGFM) for solving cathodic protection problems with both linear and nonlinear boundary conditions. The DGFM combines the accuracy of analytical methods with the flexibility of numerical techniques, enabling efficient and precise solutions. A graph theory-based framework is employed to generate the discrete Green's functions from the Laplacian matrix, allowing the electric potential distribution to be calculated through matrix multiplications. The method is validated against analytical solutions for a linear test problem and verified through comparison with COMSOL Multiphysics simulations for a nonlinear CP model involving realistic polarization curves for zinc and steel. Results demonstrate that DGFM significantly reduces computational time while maintaining high accuracy, making it a promising tool for CP design and optimization.

Natural Convection within an Enclosure Fitted with Blocks

Nabila Labsi, Mohamed Amine Oulmane, Fouad Zouiri, Youb Khaled Benkahla

Faculty of Mechanical Engineering, University of Science and Technology Houari Boumediene, BP 32 El Alia, Bab Ezzouar, 16111 Algiers, Algeria

nabilalabsi@yahoo.fr

The present study deals with the numerical analysis of natural laminar convection within a square cavity whose vertical walls are maintained at a constant temperature while the horizontal walls are thermally insulated, with the exception of a centered portion of the lower wall, which constitutes a heat source, whose temperature is greater than that of the vertical walls. Adiabatic blocks are placed inside the enclosure. The resolution of the governing equations is done by means of the finite volume method and the SIMPLER algorithm is adopted to deal with the velocity-pressure coupling. The study focuses on the influence of the presence of these blocks on heat transfer within the enclosure. The results show that the latter is reduced when the cavity is provided with adiabatic blocks and that their position plays a great role in the propagation of heat from the heat source.

Forced Convection of a Viscoplastic Fluid in a Pipe: Pressure Drop

Nabila Labsi, Youb Khaled Benkahla

Faculty of Mechanical Engineering, University of Science and Technology Houari Boumediene, BP 32 El Alia, Bab Ezzouar, 16111 Algiers, Algeria

nabilalabsi@yahoo.fr

This numerical study investigates the flow of a viscoplastic fluid obeying the Herschel-Bulkley rheological model in a pipe maintained at constant wall temperature. The emphasis is on the variation of pressure drop within the pipe, for an isoviscous fluid and a temperature-dependent one, when taking into account viscous dissipation. The results show, among others, that neglecting the thermodependence of the fluid's consistency as well as viscous dissipation leads to an underestimation of pressure drop. Consequently, considerable errors in the sizing of industrial installations can be introduced. In addition, the results reveal that the case of wall heating offers the lowest values of the friction factor (pressure drop) which are greater in the case of isoviscous flow.

Determination of the Local Nusselt Number in the Thermal Entrance Region of Flow between Two Parallel Plates

Aydın Dönmez[1], Yasemen Kuddusi[2], Lütfullah Kuddusi[1]

[1]Faculty of Engineering, Haliç University 34060 Eyüpsultan, İstanbul, Türkiye

[2]School of Engineering, École Polytechnique Fédérale de Lausanne (EPFL), Route Cantonale, CH-1015 Lausanne, Switzerland

aydindonmez@halic.edu.tr, yasemen.kuddusi@epfl.ch, lutfullahkuddusi@halic.edu.tr

Flow between two parallel plates is considered. The flow is hydrodynamically developed. The local Nusselt number would be determined in the thermal entrance region of the flow. The upper and the lower plates have constant but different temperatures. Parabolic velocity profile is assumed for the flow between the plates.

Session: S6-C ICCHMT Türkiye '25 **Paper ID:** 177

Investigation of Characteristics of Different ANN Concepts in Flow Prediction

Ali Cemal Benim, Michael Diederich

Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

alicemal@prof-benim.com, michael.diederich@hs-duesseldorf.de

Digital twins are revolutionizing the process industry by enabling predictive maintenance, optimization, and enhanced system analysis. This study explores the use of flow simulation data to train artificial neural networks (ANNs) for the development of digital twins. The research compares the computational efficiency and predictive accuracy of multiple ANN architectures, providing insights into their suitability for various industrial applications. The analysis includes Feedforward Neural Networks (FNNs), Convolutional Neural Networks (CNNs), Recurrent Neural Networks (RNNs), and Physics-Informed Neural Networks (PINNs). FNNs serve as a baseline, offering straightforward architectures suitable for general- purpose predictions but struggling with complex dependencies in the data. CNNs excel in cap- turing spatial patterns, making them highly effective for grid-based simulation data. RNNs, particularly Long Short-Term Memory (LSTM) networks, demonstrate strengths in temporal dynamics, making them ideal for time-series predictions. PINNs uniquely integrate domain knowledge and physical laws into the learning process, offering robust performance in scenari- os with limited data. This systematic comparison highlights the inherent trade-offs between model complexity, com- putational cost, and prediction accuracy. The findings aim to provide a foundation for selecting ANN architectures optimized for specific industrial challenges, emphasizing their potential in scalable digital twin implementations. By presenting a comprehensive evaluation of these architectures, this work contributes to the broader adoption and refinement of ANN-based digital twins in the process industry.

Thermal-Hydrologic-Mechanic Effects on Heat Transfer Processes in Enhanced/Engineered Geothermal Systems

Yu-Shu Wu, Phil Winterfeld

Department of Mechanical Engineering, Colorado School of Mines, CO 80401, USA

ywu@mines.edu, pwinterf@mines.edu

Enhanced or engineered geothermal systems (EGS), or non-hydrothermal resources, is highly notable among the sustainable energy resources, because of its abundance and cleanness. The EGS concept has received worldwide attention and undergone intensive studies in the last decade in the US and around the world. In comparison, hydrothermal reservoir resource, the 'low-hanging fruit' of geothermal energy, is very limited in amount or availability, while the EGS is extensive and has great potential to supply the world with the needed energy almost permanently. The EGS, in essence, is an engineered subsurface heat mining concept, where water or another suitable heat exchange fluid, is injected into hot formations to extract heat from the hot dry rock (HDR). Specifically, EGS relies on the principle that injected water or another working fluid, penetrates deep into the reservoirs through fractures or high-permeability channels to adsorb large quantities of thermal energy by contact with the host hot rock. Finally, the heated fluid is produced through production wells for electricity generation or other usage. Heat mining from fractured EGS reservoirs is subject to complex interactions within the reservoir rock involving high temperature heat exchange, multiphase flow, rock deformation, and chemical reactions under (THM) processes or thermal-hydrological-mechanical-chemical (THMC) interactions. In this talk, we will present a THM model and reservoir simulator and its application for simulation of hydrothermal geothermal systems and EGS reservoirs. The simulator couples heat flow with geomechanics, describing complex fluid and heat flow behavior in multiphase, multi-component, geothermal reservoirs. In addition, we will present several simulation examples for modeling tracer transport and geomechanics impacts on fluid and heat flow in fractured EGS reservoirs.

Prediction of Heat Transfer Characteristics of Impinging Jets Utilizing ANNs Based on CFD Simulations

Karam Abhary, Onur Bas, Selin Aradag Celebioglu

Faculty of Engineering, TED University 06420 Ankara, Türkiye

karam.abhary@tedu.edu.tr, onur.bas@tedu.edu.tr, selinaradag@gmail.com

This study integrates Computational Fluid Dynamics (CFD) simulations and Artificial Neural Networks (ANNs) to predict heat transfer in impinging jets, addressing the limitations of traditional empirical correlations. Existing correlations for impinging jets are constrained by parameters such as Reynolds number (Re), nozzle geometry, stagnation height-to diameter ratio (H/D), and inlet pipe length to-diameter ratio (L/D), often resulting in errors exceeding 20% under specific conditions. To overcome these challenges, ANNs are employed as a robust alternative, leveraging their capability to model complex, nonlinear systems with higher accuracy. Four ANN models are developed to accommodate different jet configurations: Single Round Nozzles (SRN), Single Slot Nozzles (SSN), Arrays of Round Nozzles (ARN), and Arrays of Slot Nozzles (ASN). Input parameters include non dimensionalized factors such as Re, L/D, H/D, nozzle area ratio, non-dimensionalized temperature, and its dependents (thermal conductivity, density, and viscosity), as well as nozzle arrangement. Outputs predict key quantities like Nusselt number, pressure distribution, and wall stress. A dataset comprising 30 points per parameter is generated using a Latin Hypercube distribution algorithm and validated against experimental data from the literature. CFD simulations are conducted using STAR-CCM+, employing the v2-f turbulence model for low H/D cases and SST k-ω for high H/D cases. Initial results from single-nozzle models demonstrate the feasibility of achieving accurate predictions with ANNs, paving the way for further analysis of nozzle arrays. This research significantly enhances the predictive accuracy of heat transfer and fluid dynamics in impinging jets, addressing gaps in existing literature and contributing to the optimization of engineering designs for industrial applications.

Thermal Management of High Bright Led Display System by Forced Convection and Tailored Cooling System Design with Analytical and Numerical Approaches

Ahmet Salih Adalı[1], Özgür Ertunç[2]

[1]VESTEL A.Ş. 45030 Manisa, Türkiye

[2]Department of Mechanical Engineering, Özyeğin University 34794 İstanbul, Türkiye

ozgur.ertunc@ozyegin.edu.tr

Thermal management of display devices is crucial for the long-term performance and life of the products. In line with customer demands, multipurpose display devices designed for operating in various environments need to comply with high brightness level expectations. However, high brightness requirements mean a considerable amount of power consumption which leads to significant heat dissipation within the device. Effective thermal management is important to ensure the reliability and durability of critical parts like the mainboard, power board, and LEDs which must operate below certain temperature thresholds at 40°C and 90% relative humidity [1]. Analytical and numerical approaches are generally used to optimize the cooling performance by tailoring the cooling system design. Using tools like ANSYS Icepak, the heat generation and dissipation inside the display system could be modeled [2]. A comprehensive review of existing cooling methods used in electronic devices including natural convection, forced convection, synthetic jet cooling, and phase change cooling showed that although it is commonly employed, natural convection is not optimal for managing the high heat dissipation required by high-brightness LED displays. Therefore, our research emphasizes the use of forced convection, which is more effective in such scenarios [2][3][4]. The main objective of this research is to develop and optimize a cooling system that maintains the temperatures of critical components within safe operational limits, thereby enhancing the overall reliability and efficiency of the display system. To achieve this, the study investigates various cooling methodologies while focusing on the combination of conduction and forced convection mechanisms to achieve the desired thermal management. In this study, a simple analytical model was first constructed to understand the basic thermal dynamics of the system. Then detailed numerical simulations were performed to refine the cooling strategies. Empirical correlations and governing equations for forced convection were applied to determine optimum fan positions and speeds. The study investigated various fan configurations and their impact on temperature distribution inside the display device. As a result of the simulations and calculations, it became obvious that intuitive fan placements are insufficient to achieve reliable temperature limits. This underlines the importance of strategic fan placement to ensure effective heat transfer. Key findings from the research show that a tailored cooling system, designed with a combination of conduction and forced convection, can significantly improve thermal management. The study proposes a design methodology that minimizes the number of components in the cooling system while allowing high cooling performance under harsh operating conditions. Comparison of analytical and numerical results, along with prototype testing, validates the feasibility of the proposed cooling solutions. The optimized cooling system maintains the temperatures of critical components within safe thresholds while enhancing the overall efficiency and life of the display system. In summary, our study contributes valuable insights into the thermal management of high-bright LED displays and offers practical solutions for both indoor displays and semi-outdoor display brightness applications. Future work will focus on further refining the cooling design and exploring advanced materials and technologies to enhance thermal performance.

Computational Analysis of Heat and Material Flow During Chemical Recycling of Composites for Wind Turbine Blades

Leon Mishnaevsky, Yi Chen

Department of Mechanical Engineering, Technical Uni. Denmark, Lyngby, Denmark

lemi@dtu.dk

Over the following decades, a large expansion of wind energy generation in Europe is foreseen. The goal is to cover 30% of the European Union's electricity demand with wind energy by 2030. At the same time, the wind turbines installed during the 2000s are approaching the end of their planned lifetime. While many parts of wind turbines can be well recycled, this is different for composite blades, which have been designed to sustain extreme mechanical and environmental loads for decades. Various strategies are available for the end-of-life management of wind turbine blades, including reuse, refurbishment, and recycling. Recycling is a less preferable option compared to waste reduction, recovery, and repair. However, under numerous circumstances, recycling becomes the most viable and necessary step. Among the existing recycling methods (mechanical shredding, crushing, pyrolysis, solvolysis, etc.), chemical recycling, particularly solvolysis, is relatively controllable and yields relatively clean fibers, necessitating lower temperatures than pyrolysis. In this work, a computational model of the depolymerization of composites during solvolysis of wind turbine blades is developed. The model is based on a phenomenological approach, describing the dissolution of an epoxy matrix as a local phase transition influenced by temperature, solvent diffusion, and local microstructures. The model is implemented in the finite element code Abaqus, using the user-defined field and heat flux subroutines. Parametric studies are carried out to study the influence of defects and heterogeneities on the depolymerization of composite materials. The results show that variations of the solvent diffusivity in the vicinity of fiber/matrix lead to nonhomogeneity of depolymerization, and smaller diffusivity may explain matrix residues remaining on the fibers. Fiber volume density and distribution influence the polymer dissolution rate because the resistance to solvent diffusion arises between fibers. Further, voids in the polymer may lead to a local acceleration of the polymer dissolution. This work also looks into the contributions of diffusion and reaction to depolymerization, and the former dominates. The rate and homogeneity of depolymerization, therefore, depend significantly on the manufacturing quality of composites. The developed model can be used to optimize the composite recycling, and to improve the recyclate quality.

Preliminary Macroscopic Non-equilibrium Model for Heat, Air, and Moisture Transfer in Bio-based Building Materials

Michał Wasik, Piotr Łapka

Institute of Heat Engineering, Faculty of Power and Aeronautical Engineering

Warsaw University of Technology, Nowowiejska 21/25, 00-665 Warsaw, Poland

peter.lapka@pw.edu.pl

The paper presented a non-equilibrium model for heat, air, and moisture transfer in bio-based building materials. The macroscopic approach was used to model the porous building material. The three phases, i.e., solid matrix, bound water, and humid air, were considered. Four transport equations were formulated, i.e., bound water, dry air, and vapor continuity equations, as well as an energy equation. The interaction between bound water and vapor was modeled using a non-equilibrium sorption-desorption source term to ensure an accurate representation of the coupling mechanisms. The non-equilibrium approach implies that the sorption-desorption process occurred at a finite rate. The model was implemented in ANSYS Fluent software using advanced customization interfaces such as User-Defined Function (UDF), User-Defined Scalar (UDS), and User-Defined Memory (UDM). A custom experimental setup was designed and developed to validate the proposed model. The setup was placed in the climatic chamber, enabling control of the test conditions. Measurements were conducted on hemp concrete and wood wool cement composite samples. During the experiments, the temperature was maintained at a constant level, while the relative humidity varied. These changes in ambient conditions forced heat and moisture transfer between the sample and the surroundings, which was carefully monitored. The experimental data obtained were then used to validate the model. The simulation results were found to be in good agreement with the experimental data.

Sensitivity Analysis of Micro-scale Based Method for Predicting the Thermal Conductivity Tensor of Heterogeneous Bio-based Building Materials

Piotr Łapka, Szymon Zdziarski

Institute of Heat Engineering, Faculty of Power and Aeronautical Engineering

Warsaw University of Technology, Nowowiejska 21/25, 00-665 Warsaw, Poland

szymon.zdziarski.stud@pw.edu.pl

The paper presents a sensitivity analysis of a method for predicting the thermal conductivity tensor of highly heterogeneous building composites containing bio-additives. The proposed approach is based on solving the heat conduction problem at the microscale while taking into account the actual material morphology. The microstructural data of the composites are obtained using micro-computed tomography (mCT) data. The method is tested on real samples of wood fibers and cement binder composite, commonly used in the production of wood-wool cement boards (WWCB). The material is characterized by wood fibers oriented in specific directions (i.e., along the width and length of the board), which in practical applications are typically perpendicular to the heat transfer direction (i.e., along the thickness of the board). As a result, the material exhibits significant anisotropy in thermal conductivity. The numerical tool developed incorporates a method for processing mCT data, including thresholding, selection of representative elementary volume (REV) size, determination of physical and thermal properties of composite components, and model tuning using experimental data. However, the selection of threshold levels limits to distinguish between different composite constituents is inherently arbitrary and set based on measurement data. In addition, the physical and thermal properties of composite constituents are often unknown or subject to considerable variability. For example, the properties of plant residues may depend on the specific batch, while those of cement may be influenced by production parameters. It is, therefore, essential to assess the extent to which variations in these parameters affect the predicted effective properties of the composite under consideration. Gaining this understanding is critical in assessing the reliability and accuracy of the proposed method.

Session: S7-B ICCHMT Türkiye '25 **Paper ID:** 185

Heat Transfer and Flow Behavior on the DYNOTIS ST-51 Propeller Test Bench for Unmanned Aerial Vehicles

Benjamin Allweyer, Frank Rückert, Sebastian Grün, Gurbet Bulduk, Dirk Hübner

Faculty of Engineering, htw saar – University of Applied Sciences Saarbrücken,
Goebenstraße 40, 66117 Saarbrücken, Germany

benjamin.allweyer@htwsaar.de, sebastian.gruen@htwsaar.de

gbulduk@htwsaar.de, dirk.huebner@htwsaar.de

The construction of unmanned aerial vehicles (UAV), or drones for short, can involve various complex disciplines, such as fluid mechanics, strength of materials, fluid-structure interaction, design, lightweight construction, control and regulation, programming, test planning and project management based on project management using a specific object. It is also difficult to scale-up or scale-down the size of the rotor blades based on physics and not only geometrically. The possible flight duration is also important for UAV. This depends not only on the battery, but also on the efficiency of the propeller's rotor blades. Different temperatures in the surrounding of the drone can also influence its behavior and flight duration. Here we want to take a closer look at the design of the drone propeller. The geometry of the blades of each propeller influences various important parameters such as thrust, efficiency, turbulence generation as well as the flow behavior, which affects also the sound of the drone. Heat transfer to and from the environment also influences these parameters as well as the resistance of the electrical motor. We have designed different rotor blades and used the DYNOTIS ST-51 propeller test rig for measurements of such design parameters and present various results. The rig is shown in Fig. 1, the prototypes of different blades have been produced by additive manufacturing. Heat transfer measurements at the motor were performed. In our work, we compare different 3D-printed rotor geometries for UAVs with different designs and investigate the influence of the ambient air temperature on the propeller properties. Measurements of the temperature balance and the cooling of the electric motor are also presented. The influence on the energy consumption and the thrust of the motor is explained. Generation of swirl and turbulence has been examined. This measurement data will also be helpful for further computational fluid dynamics (CFD) simulations of the generated propeller geometries. An automatic generator for the propellers can be used for geometry generation and 3D printing as well as for future flow simulations.

Experimental Investigation of Twister Type Additively Manufactured Flow Mixer Fins for the Liquid-Cooled Avionics Electronic Units

Mustafa OCAK, Fahreddin Susar, Mümin Türkyılmaz

ASELSAN A.Ş., 06750 Akyurt, Ankara Türkiye

mturkyilmaz@aselsan.com

In liquid-cooled Avionics Electronics Units (AEU), efficient heat transfer is crucial for desired high performance and preventing hot spots on 3U/6U VPX boards. Liquid-cooled heat exchanger fin structures have been well studied in the past, especially longitudinal vortex generator type fins have been pointed out for heat transfer enhancement capabilities in rectangular micro and macro channel flows. Although vortex generators are capable for mixing the flow in the longitudinal direction, some researchers claim that they are not suitable for vertical (from bottom to top) direction mixing. On the other hand, the latest advancements in additive manufacturing have enabled researchers to produce unconventional complex fin structures that can mix the flow both in longitudinal and vertical directions. On this way, a novel twister type fin structures have studied in this study which are produced with additive manufacturing. Twisted fin structures are produced both from plastic and aluminum and not merged with the heat exchanger main body, in order to reveal heat transfer enhancement by mixing, only. Experimental works are performed to illustrate the performance. Results are compared with empty channel and a vortex generator type fin structure. It is observed that additively manufactured; aluminum twister fins are decreased the wall temperatures at least 15.9 % up to 35.1%, plastic twister fins are decreased the wall temperatures at least 17.7 % up to 35.5% compared to empty channel while the vortex generators fins are decreased at least 12.4% up to 33.5 %. Considering those results, twister fins' mixing affect is found as superior than vortex generator fins because they are not merged with the heat exchanger body which disables the conductive heat transfer pathway from heat source to fins. Pressure drop increase of both twister fins and vortex generator fins compared to empty channel are identical.

Computational Study of Laminar Free Convection Heat Transfer Inside a Vertical Convergent Channel Heated Isothermally

Khalil Yassin[1], Ali Ekaid[2], Viktor Terekhov[3]

[1]Power Mechanics Techniques Department, Al-Hawija Technical Institute, Northern Technical University, Hawija, Kirkuk, Iraq

[2]Department of Mechanical Engineering, University of Technology, Al-Sina'a Street, Al-Wehda Neighborhood, Baghdad 10066, Iraq

[3]Kutateladze Institute of Thermophysics, Siberian Branch of the Russian Academy of Sciences (SB RAS), Pr. Ac. Lavrent'eva 1, Novosibirsk 630090, Russia

alialkinany74@gmail.com, terekhov@itp.nsc.ru

In this work, a laminar natural convection heat transfer inside a vertical convergent channel is investigated computationally, to predict the effect of the angle of convergence on the flow behaviour inside the channel and the heat transfer characteristics. The convergent channel is composed of two convergent plates that were heated iso-thermally and symmetrically with different angles of convergence, φ= 1, 2, 3, 4, 5, 10, 15, 20, 25, and 30 degrees respectively. The boundary, initial conditions and the effects at the inlet and exit of the channel were concluded in the study. All these cases are compared with the vertical parallel plate. The computational solution is conducted for Pr = 0.71, and different values of Rayleigh number, Ra = 10^2 ÷10^5, at aspect ratio AR=L/W = 10. A finite volume method is employed to solve the full Navier-Stokes and energy equations; the SIMPLEC algorithm with a collocated grid is utilized to connect velocity and pressure. The local Nusselt numbers are identical for the flow near the two inclined vertical walls, where it varies from high values at the channel entrance below to small values near the channel exit, where the mean air temperature inside the channel tends to be constant due to heat transfer between the airflow and the channel walls. Reynolds number values increase with the increase of values of Rayleigh number, due to buoyancy effects, and near the exit due to an increase of velocity at the contraction area. It is established, that with the increase of a parameter Ra heat transfer is increased, while with the increase of convergence angle heat transfer will be decreased. The article discusses the impact of the convergence on the heat transfer and the amount of gas flow through the convergent vertical channel which is important in engineering applications. Good agreements with the published papers were noticed.

Numerical Analysis of Solar Volumetric Absorbers Using a Two-Energy Equation Model

Fabrício Pena, Marcelo de Lemos

Department of Aeronautical Engineering, Instituto Tecnológico de Aeronáutica (ITA), Praça Marechal Eduardo Gomes, 50, Vila das Acácias, 12228-900 São José dos Campos, SP, Brazil

pena.fabricio@gmail.com, delemose@ita.br

This study investigates the thermal performance of a Solar Volumetric Absorber (SVA) through numerical simulations employing the Thermal Non-Equilibrium Model and the Rosseland approximation. A radiation boundary condition was applied at the absorber's inlet to model heat transfer accurately. The governing equations were discretized using the control volume method within a boundary-fitted non-orthogonal coordinate system, with the SIMPLE algorithm utilized to resolve pressure-velocity coupling. For validation, results were compared against simulations conducted using the open-source software OpenFOAM, showing negligible discrepancies between the in-house code and OpenFOAM. The study explored the influence of key parameters, including inlet velocity, porosity (ϕ), medium permeability (K), and the thermal conductivity ratio, on the temperature distributions of the solid and fluid phases within the absorber. Findings revealed that higher porosity or lower thermal conductivity led to reduced temperatures, while lower porosity or higher thermal conductivity ratios resulted in increased entry lengths. Additionally, higher permeability was associated with elevated solid temperatures at the inlet, longer entry lengths, and a decrease in the final equilibrium temperature. These insights contribute to optimizing the design and operation of solar volumetric absorbers for enhanced thermal efficiency.

Session: S6-E

Paper ID: 192

Numerical Simulation of a Rapid-Abandonment Oil Well

Luiz Monteiro, Marcelo de Lemos

Department of Aeronautical Engineering, Instituto Tecnológico de Aeronáutica (ITA)
Praça Marechal Eduardo Gomes, 50, Vila das Acácias, 12228-900 São José dos Campos, SP, Brazil

energia@ita.br

This work aims to model and simulate an oil well in which modifications are made, during the completion phase, to facilitate abandonment at the time of well decommissioning. The energetic material called thermite is placed in the well, at the sealing location, causing an exothermic reaction that melts the production tubing, casing, cement and rock, permanently sealing the well after cooling of the molten mass. The energy and mass transport equations are solved using a porous medium model, in which solid materials are considered permeable media with extremely low permeability. As the temperature rises, the permeability of the computational node increases, allowing the molten mass to flow, thus simulating the phase change process.

Computational Thermography for Injury Detection and Monitoring in Rugby Players

Bardia Yousefi[1], Christine Vassell[2], Rubén Usamentiaga[3], Andrea Fidanza[4], Lan Ma[2], Giuseppina Iacutone[4], Giandomenico Logroscino[4], Stefano Sfarra[5]

[1]Department of Pathology, SUNY Upstate Medical University, Syracuse, NY 13210, USA

[2]Fischell Dept. Bioeng, University of Maryland, College Park, MD20850, USA

[3]Dept. Computer Eng., University Oviedo, Campus de Viesques, 33204 Gijon, Spain

[4]Dept. Life, Health & Environmental Sciences University L'Aquila, 67100 L'Aquila, Italy

[5]Dept. Industrial and Information Eng. Econ., Uni. L'Aquila, L'Aquila, I-67100, Italy

yousefib@upstate.edu, cvassell@terpmail.umd.edu, rusamentiaga@uniovi.es

andrea.fidanza@univaq.it, lanma@umd.edu, giuseppina.iacutone@graduate.univaq.it

giandomenico.logroscino@univaq.it, stefano.sfarra@univaq.it

Infrared thermography (IRT) has emerged as a valuable tool in sports medicine for non-invasive injury detection and monitoring. In this study, the authors employed a computational thermography framework to analyze thermal variations in 12 rugby players under controlled environmental conditions. Static thermographic images were captured at three time points: before exercise, immediately post-exercise, and after a rest period, with subjects maintaining a fixed position 1.3 meters from the camera. High-dimensional (HD) static thermomic features were extracted, and machine learning models were evaluated for their predictive efficacy. A region of interest (ROI) was defined for each knee, and thermomic attributes were temporally analyzed across acquisition stages. Statistical analysis using the Wilcoxon test revealed significant pixel-level thermal variations between pre-exercise and post-exercise stages, as well as between post-exercise and post-rest stages, indicative of potential injury risk. HD thermomics distillation and classification identified substantial differences in 179, 213, and 89 lower-dimensional (LD) projected attributes for the Before/During, During/After, and Before/After comparisons, respectively. Despite the limited sample size, the unsupervised machine learning model demonstrated a predictive accuracy of 66.6%, with three false positives and one false negative. These findings suggest that computational thermography can serve as a promising tool for early injury detection and athlete monitoring, offering potential applications in sports science and rehabilitation.

Thermal and Hydrodynamic Evaluation of Microchannel Heat Sinks with Inline and Staggered Pin Fins: Enhancing Electronic Cooling

Devendra Kumar Vishwakarma[1], Ali Cemal Benim[2], Suvanjan Bhattacharyya[3]

[1]Manipal Academy of Higher Education, Madhav Nagar

Manipal – 576104, Udupi, Karnataka, India

[2]Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

[3]Dept. Mechanical Engineering, Birla Institute of Technology and Science Pilani, Vidya Vihar Campus, Pilani, Jhunjhunu, Rajasthan 333031, India

p20190451@pilani.bits-pilani.ac.in, alicemal@prof-benim.com

The rising power density of electronic devices has shown the limits of conventional cooling systems, demanding improved thermal management solutions. Microchannel heat sinks (MCHSs) are a viable alternative due to their improved heat dissipation, small design, and efficient fluid flow. This study examines inline and staggered pin fin designs at low-laminar Reynolds numbers (100–1000) to optimize MCHS thermal and hydrodynamic performance. Designing with circular pin fins ensured simplicity of production and cost-effectiveness. A detailed numerical analysis examined how pin fin configurations affect flow dynamics, heat transfer rate, and pressure drop. This work maximizes heat dissipation and minimizes pressure drop to create an effective cooling solution for high-power electronic components. Deionized water and Fe_3O_4-based magnetic nanofluid were used to investigate the cooling performance of MCHSs. Magnetic nanofluids improve heat transfer over traditional coolants by increasing thermal conductivity. To optimize operating conditions, Reynolds number changes on convective heat transfer coefficients and flow parameters were explored.

CFD Analysis of Propane Leak Dispersion and Ventilation Optimization in Refrigeration System

Rahim Jafari[1], Emre Karan[2], Hasan Savi[2], Emircan Özdemir[2]

[1]Mechanical Eng. Department, Türk Hava Kurumu University, 06790 Ankara, Türkiye

[2]Nüve Sanayi Malzemeleri İmalat ve Ticaret A.Ş., 06750 Akyurt, Ankara, Türkiye

rjafari@thk.edu.tr, emrekaran@nuve.com.tr, hasansavi@nuve.com.tr

emircanozdemir@nuve.com.tr

Refrigeration and air conditioning systems are widely utilized across various industries. Historically, chlorofluorocarbons (CFCs) and hydrochlorofluorocarbons (HCFCs) were the predominant refrigerants. However, due to their severe ozone depletion potential (ODP) and contribution to global warming, these substances have been phased out or are in the process of being eliminated under the Montreal Protocol. Although hydrofluorocarbons (HFCs) emerged as replacements for CFCs and HCFCs, their high global warming potential (GWP) has raised environmental concerns. Consequently, natural refrigerants, particularly hydrocarbons, have gained attention as viable alternatives. Among hydrocarbons, propane (R290) is considered a promising substitute for R12 due to its favorable thermodynamic properties, compatibility with mineral oils, cost-effectiveness, and environmental benefits. Additionally, propane is compatible with copper and exhibits excellent lubrication characteristics. Despite these advantages, its high flammability necessitates stringent safety measures, including robust leak detection and ventilation strategies to prevent hazardous accumulation. This study employs computational fluid dynamics (CFD) simulations to analyze potential propane leaks, identifying gas accumulation zones and airflow pathways. By leveraging propane's density, which is higher than air, the study optimizes ventilation channel placements in the system's structural design to effectively manage gas dispersion and mitigate risks. The findings contribute to enhancing the safety and reliability of propane-based refrigeration systems.

On the Polaritonic Figures-of-Merit of Ionic Crystals for Subwavelength Optics

Elif Begum Elcioglu

Dept. Mechanical Engineering, Eskişehir Technical University, 26555 Eskişehir, Türkiye

ebelcioglu@eskisehir.edu.tr

Advances in nanotechnology have paved the way for development of devic-es with exceptional properties, making appropriate material selection crucial for desired functionalities. Materials supporting surface phonon polaritons with resonances in the infrared (IR) offer significant advantages for energy and photonic applications, exhibiting lower optical losses compared to plas-monic materials (e.g., metals). In this work, we studied polaritonic behaviors of prominent ionic crystals (e.g., NaCl and LiF) in comparison to that of SiC, a wide-bandgap semiconductor, as a benchmark. The polaritonic figures-of-merit along with polariton propagation lengths and penetration depths are calculated and interpreted. Results from the single material-vacuum interface analysis reveal that the overall polaritonic figure-of-merit peak value (FOM) for SiC is the highest, followed by NaCl and LiF (with slight variations be-tween those of LiF and NaCl). The phonon-polariton resonances of LiF and NaCl exhibit higher damping compared against to that of SiC. On the other hand, LiF and NaCl has their optical phonon modes located at lower fre-quencies (in the far-IR) compared against to that of SiC (in the mid-IR), leav-ing different application ranges to explore for these materials. The findings highlight that materials must be chosen targeting spectral regions to pave the way for future advances in subwavelength optics and infrared photonics.

Computational Modeling of Thermoelectric Generators (TEGs)

Rajkumar Ramachandralal

Engineering College, Sreekaryam - Kulathoor Rd, P.O, Sreekariyam
Thiruvananthapuram, Kerala 695016, India

rajkumar@cet.ac.in

The phenomema of heat transfer associated with the transition from vapor to liquid state has been of utmost importance to many researchers because of its crucial importance in the design and development of many engineering systems, including power generation, water harvesting, seawater desalination, thermal management of electronic devices, and many more.It is commonly recognized that the wettability and surface shape of condensing surfaces have an impact on the modes of condensation (film wise or drop wise). The objective of the present work is to understand the condensation heat transfer from micro scale particle coated vertical plate where vapor condenses into a liquid on a surface that is covered with tiny particles. This investigations aims to understand the type of condensation and the thermal energy transport from vertical plate coated with aluminum oxide and tungsten micro particles. Condensation experiments are performed on a vertical copper plate (coated and uncoated) of dimension 6mm × 8mm × 3mm mounted on a Teflon box of dimensions 60mm × 80mm ×. 40mm. Water is circulated through the teflon box with the help of Polypropylene Random Copolymer pipes which can withstand high temperature. The test plate assembly is mounted inside a condensation chamber of dimensions 300 mm × 300 mm × 700 mm. Steam at 1atm is admitted into the condensation chamber.. A constant temperature bath is used to adjust the coolant inlet temperature..Both coated and uncoated copper plates have been used in experiments with a variety of coolant flow rates and coolant input temperatures. Condensation heat transfer performance of the coated and uncoated vertical copper plate will be discussed in the full length paper.

Application of Artificial Intelligence Model for Extended Jet Impingement Cooling On Wavy Target Surface

Mehmet Berkant Özel[1], Ufuk Durmaz[1], Muhammed Ali Nur Öz[1], Ali Cemal Benim[2], Ünal Uysal[1], Orhan Yalçınkaya[1], Kadircan Kasab[1]

[1]Mechanical Engineering Department, Sakarya University 54050 Sakarya, Türkiye

[2]Dept. Mech. Process Eng., Hochschule Düsseldorf 40225 Düsseldorf, Germany

mozel@sakarya.edu.tr, muhammedoz@subu.edu.tr, alicemal@prof-benim.com

uysal@sakarya.edu.tr, kkasab@sakarya.edu.tr

This study investigates jet impingement cooling (JIC). The JIC model is widely utilized to enhance heat transfer efficiency in high-temperature applications by directing jets onto the surface. However, the effectiveness of this method can be limited due to the deflection of jets caused by crossflow effects. This research uses artificial intelligence-based methods (AI) to reduce crossflow and increase heat transfer performance for an extended jet impingement cooling system with a wavy target surface. This study examines the effects of jet length and wave radius of the target surface on heat transfer performance and the influence of crossflow. The impact of jet length and wave radius of the target surface on JIC has been examined. The input data for the artificial intelligence model were obtained from the numerical model. Simulations were conducted for three different Reynolds numbers (Re = 15000, 25000, and 35000), three different jet lengths (G = conventional, 10, 20, and 25 mm), and four different wave radius (R=6, 13, 25, and 50 mm). The results from artificial intelligence-based optimization methods were compared with those from a computational fluid dynamics model. The optimization based on the AI model process involved a comparative analysis of four different optimization techniques: Nelder-Mead, Powell, SLSQP, and TNC. Artificial intelligence models have generally converged to numerical results.

HVACs Optimal Scheduling for Renewable Energy Communities Using Integrated Solar-Powered Heat Pump and Thermal Energy Storage

Leone Barbaro[1], Roberto de Lieto Vollaro[1], Andrea Vallati[2], Daniele Vitella[1]

[1]Dept. Industrial, Electonical, Mechanical Eng., University of Rome Tre, Rome, Italy

[2]Dept. Astronautical Elec. Energy Eng., University La Sapienza of Rome, Rome, Italy

roberto.delietovollaro@uniroma3.it, daniele.vitella@uniroma3.it

Renewable Energy Communities (RECs) play a significant role in addressing energy transition challenges by enabling local energy sharing from different renewable sources, maximizing environmental, economic, and social benefits. This study presents an innovative simulation tool, designed to optimize self-consumption within a REC framework. The paper focuses on an innovative REC concept that leverages the flexibility of Heat Pumps (HP) combined with Thermal Energy Storage (TES) systems. The case study examines a newly developed social housing district in Naples, designed as a Renewable Energy Community. TES enhances flexibility by integrating thermal and electrical sectors, allowing energy to be stored as heat in hot water tanks and to be used later to reduce building energy demand in the hours following the accumulation of energy. After estimating the buildings' energy demand, numerical simulations were conducted under different scenarios to identify the optimal configuration in terms of self-consumption. The results confirm that TES significantly enhances self-consumption within REC while delivering substantial energy, economic, and environmental benefits.

Investigating the Impact of Roughness Element Distributions on Shear Flow Dynamics in a Backward-Facing Step Channel

Mohammadamin Maleki, Morteza Ghorbani, Ali Koşar

Faculty of Eng. and Natural Science, Sabanci University, 34956 Tuzla, Istanbul, Türkiye

morteza.ghorbani@sabanciuniv.edu, ali.kosar@sabanciuniv.edu

This study examines how strategically distributed rectangular roughness elements upstream of a backward-facing step influence turbulent shear flow dynamic. By systematically varying the orientation angles and transverse displacements of these elements, we explore their role in modifying upstream turbulent structures and their downstream effects on shear layer development, reattachment behavior, and flow recovery. The configurations tested expand upon traditional roughness geometries, enabling a comprehensive analysis of flow control strategies in separation-dominated environments. To address the computational challenges of resolving anisotropic turbulence at high Reynolds numbers, we employ a mixed subgrid-scale (SGS) modeling approach. This method combines the strengths of static and dynamic SGS models to improve predictions of anisotropic stresses in wall-bounded turbulence, particularly at coarse grid resolutions. The hybridized framework maintains computational stability while significantly enhancing accuracy in capturing near-wall flow features critical advancement for practical engineering simulations. Our findings demonstrate that tailored roughness distributions can alter shear layer instabilities and modulate separation zone characteristics, offering insights into passive flow control mechanisms.

Enhanced Design and Performance Optimization of Membranelles Micro Flow Battery for Self-Powered Lab-on-a-Chip System

İsmail Bütün, Ali Koşar

Faculty of Eng. and Natural Science, Sabanci University, 34956 Tuzla, Istanbul, Türkiye

ali.kosar@sabanciuniv.edu

This study focuses on the design and optimization of a membraneless microflow battery, which utilizes the potential difference between electrodes and ion diffusion through liquid-liquid interactions within a linear microchannel configuration, eliminating the need for a membrane. A critical challenge in such systems is de-termining the extent to which geometric dimensions and flow parameters influ-ence the resultant electrical signal. To address this, we conducted a comprehen-sive evaluation of channel properties and flow conditions using the Taguchi De-sign of Experiment methodology, optimizing the process with 16 strategically de-signed experiments instead of the 64 required for a full factorial approach. Through image processing and analysis of the simulation results, we systemati-cally investigated the impact of geometric configurations and flow rates on the mixing efficiency of the two electrolytes. In the analytical procedure, a vertical profile line was drawn at the channel output, and the intensity values along this line were extracted and substituted into Equation 1 to calculate mixing index val-ues, which were recorded for each parameter specified in Table 1. This systematic and quantitative evaluation enabled us to identify the optimal geometric parame-ters and flow conditions that maximize performance. Building on these findings, we are advancing the development of a graphene-based membrane free microflow battery, designed to be both inflexible and flexible, thereby expanding its potential applications in energy storage systems and Lab-on-a-Chip systems for self-powering applications.

Improved Thermal Management of Electrical Vehicles Using Internal Magnetic Field of the Electric Motors

Berkay Arslan, Hakan Erturk

Mechanical Eng. Department,Boğaziçi University 34342, İstanbul, Türkiye

berkay.arslan@bogazici.edu.tr, hakan.erturk@bogazici.edu.tr

To ensure the durability and reliability of electric motors used in electric vehicles, improved cooling performance is required. This study investigates the application of magnetohydrodynamic (MHD) principles to enhance the cooling efficiency of a 110 kW permanent magnet synchronous motor (PMSM) with six axial cooling channels embedded between its coils. The motor's internal magnetic field is utilized to improve thermal management, and its influence on cooling performance is analyzed through numerical simulations. Magnetic nanofluids composed of Cu-water/ethylene glycol are employed, with particle volume fractions of up to 1% and flow rates ranging from 25 L/min to 35 L/min. Results reveal that maximum enhancement in cooling performance is achieved at a flow rate of 25.8 L/min with a 1% nanoparticle volume fraction compared to water/ethylene glycol. Under these conditions, the convection heat transfer coefficient improves by 64%, total thermal resistance decreases by 21%, and cooling capacity increases by 20%. This enhancement, attributed to the MHD effects generated by the motor's magnetic field, significantly reduces motor temperatures, extending endurance limits by up to 218%. These findings demonstrate the efficacy of MHD-based cooling strategies in improving thermal performance, thus contributing to the reliability and operational lifespan of electric motors in electric vehicles. Meanwhile, the operational conditions of pressure drop and pumping power slightly increase in the general system including heat exchanger and electric motor, yet the corresponding increase in cooling performance outweighs those increase.

Numerical Investigation of Dropwise Condensation on Biphilic Surfaces

Hossein Mohassel, Ali Kosar, Ali Sadaghiani

Faculty of Eng. and Natural Science, Sabanci University, 34956 Tuzla, Istanbul, Türkiye

ali.kosar@sabanciuniv.edu, a.sadaghiani@sabanciuniv.edu

This study investigates the heat transfer dynamics of condensation on biphilic surfaces within a minichannel, focusing on the influence of hydrophobic and hydrophilic pattern designs on droplet behavior and heat transfer performance. Two surface designs were analyzed: one with superhydrophobic substrates and hydrophobic patterns, promoting dropwise condensation (DWC), and another with superhydrophobic substrates and hydrophilic patterns, facilitating filmwise condensation (FWC). A parametric study varying pattern sizes (300, 700, and 900 µm) and steam mass flux (SMF) levels (10, 20, and 30 kg/m²s) was conducted. Numerical simulations, validated against experimental data, revealed that biphilic surfaces significantly improve droplet removal efficiency, leading to enhanced heat transfer rates due to coalescence-induced droplet jumping and surface renewal. The study also explored the coalescence mechanism, droplet velocity distribution, and pattern size impact on condensation performance. Smaller hydrophobic patterns (300 µm) enhanced droplet removal and condensation heat transfer, while larger patterns led to pinned droplets, reducing heat transfer efficiency. Notably, biphilic surfaces with hydrophilic patterns exhibited superior performance over hydrophobic designs due to higher nucleation rates. These findings provide valuable guidelines for optimizing biphilic surface designs to maximize condensation heat transfer and improve energy efficiency in thermal management systems.

Machine-learning-assisted Optimal Airfoil Design at High Mach Numbers

Kerem Dülger[1], Barbaros Cetin[1], Gökberk Kabacaoglu[2]

[1]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[2]Dept. Computer Science, Durham University, Durham, DH1 3LE, United Kingdom

kerem.dulger@bilkent.edu.tr, barbaros.cetin@bilkent.edu.tr

gokberk.kabacaoglu@durham.ac.uk

Traditionally, aerodynamic coefficients are calculated using computational fluid dynamics (CFD) methods. In projects that require numerous geometry and flight condition variations, calculating each case will be time consuming. To address this challenge, data driven prediction models are gained popularity. Neural networks are one of the popular data driven prediction model. Conventional prediction models require airfoil's geometry as coordinate data or pre-processing techniques such as shape parametrization methods. In this study, we propose a novel approach that eliminates the preprocessing by directly using NACA 4-digit and 5-digit series names as an inputs to our neural network model. By combining the airfoil series name and flight conditions such as Mach number and angle of attack, the proposed model predicts lift coefficient (Cl) and moment coefficient (Cm). This approach reduces computational cost by decreasing the input size and eliminating pre-processing steps. The results demonstrate that NACA name-based input encoding can serve as an efficient alternative to traditional geometry-dependent methods for aerodynamic analysis.

Implementation of numerical schemes for the computation of incompressible flows in OpenFOAM

Andrey Epikhin

Ivannikov Institute for System Programming of the Russian Academy of Sciences,Moscow, Russia

andreyepikhin@ispras.ru

In this research, the numerical schemes for the calculation of incompressible flows in the OpenFOAM software are developed and tested. In order to select and apply eddy-resolved approaches, the dissipativity and stability of most of the numerical schemes implemented in the OpenFOAM package are analyzed by solving the problems of degeneracy of homogeneous isotropic turbulence and scalar transport. It is found that some schemes correctly describe the evolution of vortex structures but are not stable, so they are improved to eliminate oscillations and maintain an acceptable level of dissipation. Also, a streamline upwind numerical scheme using equal order finite volume cell size to introduce additional stabilization is realized and tested. Classical computational fluid dynamics cases are considered for validation. Both averaged and pulsation characteristics are evaluated. The numerical schemes have been combined in an OpenFOAM library and are available on GitHub.

Session: S5-B **Paper ID:** 210

Investigation of Contact Melting in Molten Salt Phase Change Units with Proposed Euler-Lagrange Iteration

Ziliang Zhu[1], Yuang Jiang[1], Dongjun Xu[1], Mei Lin[1]

Abdulmajeed Mohamad[2], Qiuwang Wang[1]

[1]Xi'an Jiaotong University, Xi'an, Shaanxi 710049, P.R. China

[2]University of Calgary, 2500 University Dr NW Calgary, AB, Canada

jiangyuang@stu.xjtu.edu.cn, xudongjun@stu.xjtu.edu.cn, janeylinm@mail.xjtu.edu.cn

Molten salt phase change units are essential for improving thermal storage efficiency. However, accurately modeling contact melting phenomena, characterized by solid motion and volumetric expansion, remains challenging. This study presents an Euler-Lagrange iteration algorithm to simulate contact melting processes in molten salt units during cold start. By combining internal Euler iterations for flow and heat transfer with external Lagrange iterations for solid motion, the proposed method achieves strong coupling solutions and reduces numerical oscillations by about 51.42%, with the prediction error of 4.93%. The numerical results reveal that the contact melting of $NaNO_3$ has four distinct stages: dominant conduction, expansive convection, natural convection, and decaying convection. This study also captures the intermittent upward and downward solid motions caused by large volumetric expansion, affecting the melting rate and liquid flow patterns greatly. The geometric analysis indicates that reducing the aspect ratio can markedly enhance melting performance, with lower aspect ratio showing up to the 37.3% shorter complete melting time and thicker liquid film. The findings offer theoretical insights and practical guidance for optimizing design and performance of molten salt phase change units.

Responsive Virtual Wall Liquid Crystal Microfluidics

Emre Bukusoglu, Livio Nicola Carenza, Ayşe Nurcan Özşahin, Gülce İlhan

Cansu Erdil, Cansu Dedeoğlu, İrem Özen

Department of Chemical Eng., Middle East Technical University,06800 Ankara, Türkiye

cansu.dedeoglu@metu.edu.tr, emrebuk@metu.edu.tr

We investigate the structures and structural transitions in a liquid crystal (LC)-aqueous two-phase microfluidic system. We stabilized an LC-aqueous soft interface (virtual walls) using the preferential wetting of the two phases on a heterogeneous interface with distinct hydrophilic and hydrophobic functionalization. The interface was stable enough to maintain a co-current and countercurrent flow configuration of the two phases and allow the adsorption of the species of analytical interest that causes alignment transitions in the LC phases. To date, we have demonstrated the principles to maintain an intact LC-aqueous interface during pressure-driven flow (up to 40 mbar of the pressure difference between the two phases) and the influence of the bulk and interfacial shear on the structures of the nematic LCs that resulted in more than 16 flow-induced configurations that we have shown in experiments and numerical simulations. We have also demonstrated (i) how the microfluidic resistances are affected by the interfacial shear and the local LC director fields, (ii) the optical response of the LC phase resulting from the configuration transitions upon the adsorption of analytical species present in the aqueous phase in two different systems. Overall, we have demonstrated that the LC-aqueous soft interfaced platform presents a promising candidate for future high-throughput sensing platforms.

An Analysis of the Airflow Patterns of an Electrohydrodynamic Fan

Marek Jaszczur[1], Marek Borowski[2], Klaudia Zwolińska-Glądys[2]

[1]Faculty of Energy and Fuels, AGH University of Kraków, Kraków, Poland

[2] Facul. Civil Eng. & Resource Management, AGH University of Krakow, Krakow, Poland

borowski@agh.edu.pl

Pumps and fans are crucial components of various systems, including ventilation, air conditioning, and cooling systems. In a large number of research studies analyses focus on solution optimization using traditional devices. However, modern ventilation systems and the miniaturization of electronic devices require new solutions, which should comply with noise requirements and be very compact while also consuming less energy. Typical fans operating at high rotational speeds cause significant noise, hence the search for new, compact, efficient, and noiseless solutions is obvious. The proposed in this work electrohydrodynamic fan constitutes a potential solution to the above issues. The proposed solution consists of two electrodes: an emitting needle and a grounded ring. In this bladeless system, flow motion is generated as a result of corona discharges and air ionization. The prototype is designed as a two-element setup that allows for changing the basic configuration during the tests, In this research study, an analysis of the airflow patterns induced by an electrohydrodynamic fan has been carried out. The Particle Image Velocimetry technique was used in the measurements to analyze the velocity fields in the tested sections. The tests included measuring flow patterns in selected planes around the prototype device, operating under preselected configurations. The results indicate directional velocity fields in the analyzed devices, showing the prototype's usability and its potential application in electronic cooling or ventilation systems. Based on the analysis it was possible to determine the optimal solution for the proposed design, which enhanced the efficiency of the prototype device.

Numerical Simulation On Liquid Metals Flowing Through Rough Parallel Plates with Uniform Heat Flux Heating

Yao Xiong, Hong Wang, Haohua Song, Xiao Yan, Xun Zhu, Qiang Liao

Chongqing University, Shapingba, Chongqing 400044, China

zhuxun@cqu.edu.cn lqzx@cqu.edu.cn

The Lead-Bismuth Cooled Fast Reactor is one of the key reactor types developed in the fourth-generation nuclear energy system due to its inherent passive safety characteristics, high neutron utilization rate, and high fuel utilization rate. In the reactor, lead-bismuth liquid metal with the characteristic of low Prandtl number is used as coolant due to its high molecular heat conductivity. However, the liquid metal is easy to corrode the flow channel wall during long-term operation of the reactor, resulting in rough wall surface. Therefore, a deeper understanding of the flow and heat transfer characteristics of liquid metal in rough wall flow channels is of great significance for maintaining the safety of reactor operation. In this study, the turbulent flow and heat transfer of liquid metal with Pr number from 0.01 to 0.05 between smooth/rough parallel plates with uniform heat flux heating are studied by using the direct numerical simulation method, and the general fluids with Pr number of 0.2 to 1 are also studied as comparison. The simulation exhibits the statistical data such as time-averaged temperature, turbulent heat flux. The results show that molecular conduction is the dominant heat transfer mechanism in liquid metals. Notably, the region governed by the linear law extends further, while the logarithmic law region is shorter or may even disappeared, compared to general fluids. As the molecular Prandtl number decreases, the turbulent heat flux diminishes. Furthermore, the turbulent Prandtl number is independent of both the wall-normal distance and the molecular Prandtl number for Pr > 0.2, but it is higher for liquid metals than for general fluids, and exhibiting a strong sensitivity to the molecular Prandtl number for liquid metals.

Molecular Dynamics Simulation of Epoxy-Based Polymer Coatings: Microstructure And Water Transport Mechanism at Metal Interfaces

Julian Pichler[1], Simge Tarkuc[2], Markus Valtiner[1], Alper Tunga Celebi[1]

[1]Institute of Applied Physics, TU Wien, Vienna, Austria

[2]Dept. Metal and Surface Tech., Res. Develop. Center, Arçelik 34950 İstanbul, Türkiye

julian.pichler@tuwien.ac.at, simge.tarkuc@beko.com, celebi@iap.tuwien.ac.at

Polymer coatings are widely used approach to protect metals from corrosion by limiting exposure to aggressive environments while providing additional benefits such as chemical resistance, mechanical strength, and adhesion. This study employs molecular dynamics simulations to investigate epoxy-based polymer coatings, both in bulk and metal interface, with a focus on molecular interactions, microstructure, and water transport behavior. The impact of varying functional groups, specifically the length of polyfluorinated carbon chains and the presence of amine groups from different silanes is explored. Simulations reveal that fluorinated chains hinder water diffusion by creating steric barriers, while water tends to bind at hydrophilic sites such as oxygen and nitrogen rich regions of the coating, forming hydrogen bonds. Water molecules diffuse from one hydrophilic site to the next hydrophilic site, due to the attractive interatomic forces. This is so-called as Hopping diffusion. Hydrogen bonding between water molecules and the polymer indicates partial incorporation of water into the coating, altering its structure and slowing diffusion at the interface. By providing atomistic insights into the mechanisms of structural and transport properties, this study contributes to the development and optimization of new polymer coatings materials to mitigate against corrosion.

A Machine Learning Framework for Hemodynamic Analysis of Stenosed Arteries

Kerem Dülger[1], Barbaros Cetin[1], Gökberk Kabacaoglu[2]

[1]Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

[2]Dept. Computer Science, Durham University, Durham, DH1 3LE, United Kingdom

kerem.dulger@bilkent.edu.tr, barbaros.cetin@bilkent.edu.tr

gokberk.kabacaoglu@durham.ac.uk,

Accurate analysis of blood flow in stenosed vessels is critical for understanding disease progression, assessing the severity of arterial narrowing, and supporting clinical decision-making. Traditionally, such hemodynamic assessments, particularly the evaluation of shear stress and velocity values, rely on computational fluid dynamics models. However, performing CFD simulations for a wide range of stenosis geometries is computationally demanding and impractical for large-scale or time-sensitive applications. To address this challenge, we created a novel dataset of stenosed artery geometries, incorporating a diverse set of morphological parameters that have not been studied in previous works. Since no existing database was available, all cases were manually generated and simulated. Using this dataset, we trained a machine learning model that predicts shear stress and velocity values based solely on geometric input parameters. This geometry driven prediction model significantly reduces computational cost while enabling rapid estimation of key hemodynamic metrics. The proposed approach has strong potential for integration into real-time diagnostic tools and patient-specific treatment planning, offering a scalable and efficient alternative to conventional CFD methods.

Cladded Porous Material Integrated Pin Fin Heat Sink Performance Evaluation

Deniz Alp Yılmaz[1], Özgür Bayer[1], İsmail Solmaz[2]

[1]Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

[2]Atatürk University 25240, Yakutiye, Erzurum, Türkiye

denizao93@gmail.com, bayer@metu.edu.tr, ismail.solmaz@atauni.edu.tr

As electronics and informatics technology progress, there is an increasing need for more efficient cooling methods in electronics. Open-cell metal foams provide a unique solution by combining the strength of metals with the benefits of foams, such as being lightweight and offering a high heat transfer surface area. While metallic foams provide a large surface area for heat transfer, filling heatsinks with these materials leads to undesirable high pressure drops, causing a considerable decrease in hydrothermal efficiency. Having cladded porous foams around pin fins is a solution to increase the heat sink hydrothermal performance. On the other hand, having porous foams around the whole perimeter increases the air-solid contact area in the wake region after the separation point, resulting in unnecessary foam usage and extra pressure drop. This study introduces a novel design featuring cladded aluminum foams placed on the frontal half of the pin fin of a heatsink. Both experimental and numerical investigations are conducted to evaluate the hydrothermal performance of this design. Hydrothermal performances of fully and partially foam-cladded heat sinks are compared under different heat loads and cooling air velocities.

Session: S7-B **Paper ID:** 217

Computational Design of a High-Pressure Compressor and its Experimental Validation

Ahmet Berkay Şimşek[1], Barış Erdoğan[2], Necip Berker Uner[1]

[1]Chemical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

[2]Logos Kimya INC. 06374 Ankara, Türkiye

barise@logoskimya.com

Providing high-pressure compressible gas streams is a requirement in various industrial processes related to energy storage, transportation, propulsion, and petrochemistry. For a wide range of flow rates and pressures, oil-free reciprocating piston compressors are commonly employed for this purpose due to their portability, small form factor and inherent safety. These compressors are called booster pumps if the intake gas is already at high pressure. Modeling and designing reciprocating compressors, especially booster pumps, are important since making changes in product specifications is time-consuming and expensive. Designing a high-pressure compressor for a desired set of throughput and delivery pressure depends heavily on suction and discharge gas temperature and pressure, rotating shaft power and mechanics, compression ratio within the piston barrel, check valve properties such as cracking pressure and conductance, materials used in the piston, barrel and seals, and finally, the efficiency of heat transfer. Due to this complexity, the design and manufacturing of high-pressure reciprocating compressor is commonly conducted through trial-and-error, and a very small number of companies possess the experience and funds to iterate and develop new compressors with desired properties. Interestingly, experimental and computational studies are scarce in literature as well, therefore there appears a need to establish a robust and capable modeling framework that can unravel key points for high-pressure compressor design. In this study, a new and comprehensive modeling framework was developed for predicting the effects of operation and design parameters on compressor performance. First, a 2-D computational fluid dynamics (CFD) model was developed to simulate compression, fluid flow and heat transfer inside the piston-barrel assembly with semi-empirical check-valve flow physics taken into account. It was hypothesized that heat exchange with the surroundings plays an important role in compressor performance. To fill this gap, experiments were conducted in a test bed to measure the overall heat transfer coefficient that describes the heat loss from the gas inside the barrel. Finally, for rapid sizing, prototyping and exploration, a macroscopic model was developed to assist and replicate CFD results with significantly less computational resources. The results of the modeling work will be shared along with comparison to data obtained on a prototype booster that can operate with an outlet pressure of 140-700 bar.

Wearable Platforms for Continuous Ultrasonic Imaging and Biomarker Monitoring

Levent Beker

Research Center for Translational Medicine, Koç University 34450 Istanbul, Türkiye

LBEKER@ku.edu.tr

Wearable ultrasonic transducers, driven by high-voltage signals, generate substantial thermal loads, which can adversely affect wearer comfort and device longevity. This thermodynamic challenge necessitates innovative heat-management solutions to facilitate safe, continuous usage. On the other hand, interstitial fluid is a rich environment with challenges in sampling. In this research, we present detailed thermal analyses of wearable ultrasound devices operating at clinically relevant excitation voltages. Our results quantify temperature elevations on the skin and surrounding tissues, revealing significant dependencies on transducer design and excitation parameters. We discuss potential solutions, focusing particularly on novel thermally conductive yet flexible materials and coatings specifically engineered to dissipate heat effectively. Our material-based thermal management strategies demonstrate promising pathways for mitigating thermal issues and enhancing device functionality and user acceptance.

Flowing Liquid Crystal-Aqueous Interfaces that Respond to Lipid Adsorption

Cansu Dedeoğlu, Emre Bukusoglu

Chemical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

cansu.dedeoglu@metu.edu.tr

The mechanical properties of biological membranes are critical for maintaining proper cellular activities, and changes in them contribute to a range of diseases, including Alzheimer's, neurodegenerative disorders, and cardiovascular dysfunctions, among others. Previous studies show that at stagnant nematic liquid crystal (LC) and water interfaces, the presence and lateral organization of the phospholipids induce interface-driven orientational transitions. However, approaches to date lack the precise quantitation of the mechanical properties of the vesicles that determine the kinetics of adsorption. To overcome this, we employed a flowing interface of nematic LC and aqueous phases in microfluidic channels, offering a continuous and rapid method for quantification. We investigated a range of phospholipids that include 1,2-dilauroyl-sn-glycero-3-phosphocholine (DLPC, Tm=-2°C), 1,2-dioleoyl-sn-glycero-3-phosphocholine (DOPC, Tm= -17°C), 1,2-dipalmitoyl-sn-glycero-3-phosphocholine (DPPC, Tm= 41°C), sphingomyelin (Egg SM, Tm= 38°C) that differ as liquid-like (DLPC and DOPC) and solid-like (SM and DPPC) mechanical properties, or as their change in mechanical properties when mixed with guest molecules. By using fluorescence confocal and polarized light microscopy, we demonstrated that flowing systems provide a rapid response for all types of lipids, even including the solid-like SM and DPPC, which differ in their distribution along the interface. We also show that guest-dissolved lipid complexes that exhibit a change in mechanical properties result in different LC-response characteristics. We also employed stagnant systems and LC droplet systems in aqueous phase to create the LC-water interface for further evidence. Our ongoing efforts is to model the flowing interface system to predict the dependency of the LC response to the mechanical properties and fusion kinetics of the lipid vesicles at the flowing LC-aqueous interfaces. Our results to date consist of results that propose a system that advances our understanding of membrane rigidity-associated changes and promises a continuous early diagnosis platform that can sense mechanical alterations in cell membranes.

Identification Of Transient Fluid Temperature Using Thermometer Readings

Jan Taler, A Dawid Taler, Tomasz Sobota

Cracow Unversity of Technology, 31-155 Kraków, Poland

tomasz.sobota@pk.edu.pl

The paper presents a new method for measuring fluid temperature using a thermometer shaped as a full cylinder, with the temperature measured along its axis. The fluid temperature is determined by solving the inverse heat exchange problem for the thermometer. The heat transfer coefficient on the outer surface of the cylinder is determined based on the thermometer's response to a step change in fluid temperature from the initial temperature to a final temperature close to the boiling point at atmospheric pressure. The heat transfer coefficient is calculated using the condition of minimising the sum of squared differences between the measured and computed temperatures along the axis of the thermometer. A general mathematical model of the thermometer in the form of a cylinder, sphere, or plate is developed using the finite volume method. The developed model allows for determining the thermometer temperature with high accuracy for given temporal changes in fluid temperature. After determining the heat transfer coefficient, the fluid temperature is calculated using the mathematical model of the first, second, and third-order thermometer. The accuracy of determining the unsteady fluid temperature using the first, second, and third-order models is estimated by calculating the mean square error between the measured and computed thermometer temperatures.

Microsystems Direct Cooling Using A Bi-Phase Microfluidics Droplet

Abdel Illah El Abed[1], Xiaoyan Ma[2], Rachid Bennacer[1]

[1]Laboratoire Lumière Matière aux Interfaces, ENS Paris Saclay

Université Paris Saclay, France

[2]Hangzhou International Innovation Inst., Beihang University, Hangzhou 311115 China

abdel.el-abed@ens-paris-saclay.fr, maxiaoyan@buaa.edu.cn

The operational temperature critically influences the performance, reliability, and lifespan of a wide range of electrical and electromechanical microsystems. Conventional cooling methods employing oils are constrained by the inherently low thermal conductivity of these liquids. Although water-based fluid loops can provide enhanced thermal management, their application is limited by the non-dielectric nature of water, which poses significant risks when used in direct contact with electronic components. Furthermore, liquid-gas multiphase cooling systems are associated with stability and reliability challenges. Droplet microfluidics presents a promising alternative, enabling the controlled generation and manipulation of highly monodisperse microdroplets, each functioning as an independent microreactor. This technology facilitates high-throughput parallel operations with integrated analytical capabilities, achieving processing rates on the order of kilohertz. Over the past decade, microfluidic systems have found extensive applications, particularly in the biomedical sector, with growing interest across diverse scientific fields including biology, physics, chemistry, and materials science. In this study, a novel cooling strategy based on droplet microfluidics is proposed for the thermal management of microsystems. The approach involves a two-phase flow, wherein discrete water droplets are suspended within a continuous oil phase under microfluidic conditions. A comprehensive theoretical analysis is conducted to evaluate the thermophysical properties of the two-phase fluid and their impact on the direct-contact cooling performance. Key parameters such as microchannel geometry, droplet size, inter-droplet spacing, contact surface area, heating profiles, and material properties are systematically investigated using both analytical modeling and numerical simulations. The overarching objective is to design and realize an experimental platform capable of precise droplet generation and accurate control of the oil-to-water loading ratio, while quantitatively assessing enhancements in heat transfer performance and the associated pressure drop. The results reveal notable and unexpected improvements, which are attributed to the complex flow structures and internal streaming patterns induced within the microdroplets.

Session: S6-A **Paper ID:** 222

Simulation-Based Analysis of Oil Flow Behavior in Hydrostatic Bearings for Vertical Francis Turbines

Kamer Şevval Demir, Kutay Çelebioğlu

Hydro Energy Research Center, TOBB Uni. Economics and Technology, Ankara, Türkiye

kamersevvaldemir@gmail.com, kutaycelebioglu@gmail.com

Hydrostatic bearings play a critical role in supporting both radial and axial loads in vertical-type Francis turbines, particularly under high-load and high-precision operational conditions. Ensuring stable operation and minimizing frictional losses in these systems is crucial for the overall performance and reliability of hydroelectric power plants. In this study, the oil flow characteristics within a combined radial-axial hydrostatic spindle bearing are investigated through advanced computational fluid dynamics (CFD) simulations. A comprehensive simulation model was developed, based on the Reynolds-averaged Navier-Stokes equations. The numerical analysis systematically explores the effects of key operational parameters, including supply pressure, oil viscosity, radial clearance, and recess geometry, on critical bearing performance metrics such as load-carrying capacity, pressure distribution, flow uniformity, and minimum oil film thickness. Baseline simulations were performed for the current hydrostatic bearing design. Subsequently, parametric studies were conducted to evaluate the sensitivity of bearing behaviour to design modifications. The results reveal how subtle changes in design and operating conditions can significantly influence the lubrication performance and stability margins. This research provides essential insights for optimizing hydrostatic bearing systems in heavy-duty hydraulic machinery and contributes to improving turbine efficiency and operational safety. Future work will extend this framework by integrating machine learning models to predict bearing performance and guide intelligent design optimization strategies.

Interferometry Guided Investigation of a Thin Film

Yiğit Ata Ağartan[1,2], Barbaros Çetin[3], Zafer Dursunkaya[2]

[1]ASELSAN AŞ. 06830 Gölbaşı, Ankara, Türkiye

[2] Mechanical Eng. Department, Middle East Technical University 06800 Ankara, Türkiye

[3] Mechanical Engineering Department, Bilkent University 06800 Ankara, Türkiye

yaagartan@aselsan.com, barbaros.cetin@bilkent.edu.tr, refaz@metu.edu.tr

Interferometry is a precise technique that utilizes the phase differences between two or more light waves traveling different optical path lengths for measurements. Among its various types, fiber optic interferometers stand out due to their easy setup and low cost. The light beam exiting from and returning to the fiber optic cable tip interferes with the reference beam reflected back from the same cable tip, creating a phase difference with the reference beam along its different path. This process carries information about many parameters such as distance, the refractive index of the medium the beam passes through, the presence of gas or liquid interfaces along the optical path, and the shape of the reflective surface it comes into contact with. In this study, the effects of the tilt of the reflective surface used in the setup, probe height, and the medium through which the light passes along the optical path on the interferometric data have been investigated through simulations and validated with experiments.

Effects of Kinematic Hardening of Mucus Polymers on the Airway Closure Phenomenon

Bartu Fazla[1], Oğuzhan Erken[2], Daulet Izbassarov[3]

Francesco Romanò[4], Metin Muradoğlu[5]

[1]ASELSAN AŞ. 06830 Gölbaşı, Ankara, Türkiye

[2]University of Edinburg, UK

[3]Finnish Meteorological Institute, Finland

[4]Ecole Nationale Supérieure d'Arts et Métiers 75013 Paris, France

[5]Mechanical Engineering Department, Koç University 34450 Istanbul, Türkiye

bartufazla@aselsan.com

Liquid plug formation inside a human airway, a phenomenon known as airway closure, is studied computationally by considering the elastoviscoplastic (EVP) properties of the pulmonary mucus covering the airway walls. Airway is modeled as a rigid tube with a thin layer of EVP liquid lining the wall. A range of liquid thickness and the Laplace number is considered. For the EVP layer, The Saramito-Herschel-Bulkley (Saramito-HB) model is coupled with an Isotropic Kinematic Hardening model (Saramito-HB-IKH) to allow energy dissipation at low strain rates. The rheological model is fitted to the experimental data under the following conditions: cystic fibrosis (CF), asthma, chronic obstructive pulmonary disease (COPD), and healthy. Particularly for the CF case, it is shown that the kinematic hardening significantly affects the closure time, with the effect being more noticeable at low Laplace numbers and initial film thicknesses. In addition to the kinematic hardening implementation, the independent impacts of rheological characteristics on wall stresses are investigated, using their physiological values as a baseline. The closure time and wall stresses are found to be significantly influenced by the elastic modulus, whereas the relaxation of stresses following closure is specifically impacted by the polymeric viscosity. The closure time is also notably influenced by yield stress.

Modeling Localized Heating Induced Size Effects in Semiconductor Devices

Canberk Dündar[1], Fatma Nazlı Dönmezer Akgün[2]

[1]ASELSAN A.Ş. 06750 Akyurt, Ankara Türkiye

[2]Mechanical Eng. Department, Boğaziçi University 34342, İstanbul, Türkiye

cadundar@aselsan.com

Liquid plug formation inside a human airway, a phenomenon known as airway closure, is studied computationally by considering the elastoviscoplastic (EVP) properties of the pulmonary mucus covering the airway walls. Airway is modeled as a rigid tube with a thin layer of EVP liquid lining the wall. A range of liquid thickness and the Laplace number is considered. For the EVP layer, The Saramito-Herschel-Bulkley (Saramito-HB) model is coupled with an Isotropic Kinematic Hardening model (Saramito-HB-IKH) to allow energy dissipation at low strain rates. The rheological model is fitted to the experimental data under the following conditions: cystic fibrosis (CF), asthma, chronic obstructive pulmonary disease (COPD), and healthy. Particularly for the CF case, it is shown that the kinematic hardening significantly affects the closure time, with the effect being more noticeable at low Laplace numbers and initial film thicknesses. In addition to the kinematic hardening implementation, the independent impacts of rheological characteristics on wall stresses are investigated, using their physiological values as a baseline. The closure time and wall stresses are found to be significantly influenced by the elastic modulus, whereas the relaxation of stresses following closure is specifically impacted by the polymeric viscosity. The closure time is also notably influenced by yield stress.