ICCHMT ROME 2019

XII

International Conference on Computational Heat, Mass and Momentum Transfer

3-6 SEPTEMBER 2019

Rome, Italy

Book of Abstracts

Organized by

ROMA TRE UNIVERSITÀ DEGLI STUDI

SAPIENZA UNIVERSITÀ DI ROMA

in collaboration with

TOR VERGATA UNIVERSITÀ DEGLI STUDI DI ROMA

PK

école normale supérieure paris-saclay

UNIVERSITY OF CALGARY

Sponsor

Sapienza University
Via Eudossiana 18 | Rome

Roma Tre University
Via della Vasca Navale 79 | Rome

Chairs:

Andrea Vallati
Sapienza University

Roberto de Lieto Vollaro
Rome Tre University

www.icchmt2019.com

Date: Tuesday, 03/Sep/2019

7:30am - 9:00am	**Registration**
9:00am - 9:10am La Sapienza	**Welcome Greetings of La Sapienza University**
9:10am - 11:10am La Sapienza	**KEY-01: Keynote Session 1** Session Chair: **Prof. Jan Taler** Plenary room
11:10am - 11:30am	**Coffee Break**
11:30am - 1:30pm La Sapienza	**SE-01: Day 1 Session 1** Session Chair: **Prof. Paweł Ocłoń** Plenary room
11:30am - 1:30pm La Sapienza	**SE-02: Day 1 Session 2** Session Chair: **Prof. MingJia Li** Room 7
11:30am - 1:30pm La Sapienza	**SE-03: Day 1 Session 3** Session Chair: **Prof. Mathias Krause** Room 17
1:30pm - 2:20pm	**Lunch**
2:20pm - 4:20pm La Sapienza	**SE-04: Day 1 Session 4** Session Chair: **Dr. Sandra Corasaniti** Plenary room
2:20pm - 4:20pm La Sapienza	**SE-05: Day 1 Session 5** Session Chair: **Prof. Ali Cemal Benim** Room 7
2:20pm - 4:20pm La Sapienza	**SE-06: Day 1 Session 6** Session Chair: **Prof. Emanuele Habib** Room 17
4:20pm - 4:40pm	**Coffee Break**

Date: Wednesday, 04/Sep/2019

7:30am - 9:00am	**Registration**
9:00am - 9:10am Uniroma Tre	**Welcome Greetings of Roma Tre University**
9:10am - 11:10am Uniroma Tre	**KEY-02: Keynote Session 2** Session Chair: **Prof. Abdulmajeed Mohamad** Plenary room
11:10am - 11:30am	**Coffee Break**
11:30am - 1:30pm Uniroma Tre	**SE-07: Day 2 Session 7** Session Chair: **Dr. Marek Jaszczur** Plenary room
11:30am - 1:30pm Uniroma Tre	**SE-08: Day 2 Session 8** Session Chair: **Prof. Rachid Bennacer** Room 20
11:30am - 1:30pm Uniroma Tre	**SE-09: Day 2 Session 9** Session Chair: **Prof. Sang-Ho Suh** Room 21
11:30am - 1:30pm Uniroma Tre	**SE-10: Day 2 Session 10** Session Chair: **Dr. Ivano Petracci** Room 22
1:30pm - 2:20pm	**Lunch**
2:20pm - 4:20pm Uniroma Tre	**SE-11: Day 2 Session 11** Session Chair: **Prof. Lubna Younis** Plenary room
2:20pm - 4:20pm Uniroma Tre	**SE-12: Day 2 Session 12** Session Chair: **Prof. Janusz T. Cieslinski** Room 20
2:20pm - 4:20pm Uniroma Tre	**SE-13: Day 2 Session 13** Session Chair: **Dr. Magdalena Piasecka** Room 21
2:20pm - 4:20pm	

Uniroma Tre	**SE-14: Day 2 Session 14** Session Chair: **Dr. Piotr Łapka** Room 22
4:20pm - 4:40pm	**Coffee Break and Conference Group Photo**
4:40pm - 6:00pm Uniroma Tre	**PS-01: Day 2 Poster Session 1**

Date: Thursday, 05/Sep/2019

8:00am - 9:00am	**Registration**
9:00am - 11:00am Uniroma Tre	**KEY-03: Keynote Session 3** Session Chair: **Prof. Francesco Asdrubali** Plenary room
11:00am - 11:20am	**Coffee Break**
11:20am - 1:20pm Uniroma Tre	**SE-15: Day 3 Session 15** Session Chair: **Prof. Aldo Fanchiotti** Plenary room
11:20am - 1:20pm Uniroma Tre	**SE-16: Day 3 Session 16** Session Chair: **Prof. Artur Maciag** Room 20
11:20am - 1:20pm Uniroma Tre	**SE-17: Day 3 Session 17** Session Chair: **Prof. Rafal Kobylecki** Room 21
11:20am - 1:20pm Uniroma Tre	**SE-18: Day 3 Session 18** Session Chair: **Prof. Lucia Fontana** Room 22
1:30pm - 2:20pm	**Lunch**
2:20pm - 4:20pm Uniroma Tre	**SE-19: Day 3 Session 19** Session Chair: **Prof. Krishnaswamy Nandkumar** Plenary room
2:20pm - 4:20pm Uniroma Tre	**SE-20: Day 3 Session 20** Session Chair: **Prof. Bujalski Wojciech** Room 20
2:20pm - 4:20pm Uniroma Tre	**SE-21: Day 3 Session 21** Session Chair: **Prof. Sławomir Pietrowicz** Room 21
2:20pm - 4:20pm Uniroma Tre	**SE-22: Day 3 Session 22** Session Chair: **Prof. Oronzio Manca** Room 22
4:20pm - 4:40pm	**Coffee Break**
4:40pm - 6:00pm Uniroma Tre	**PS-02: Day 3 Poster Session 2**

Date: Friday, 06/Sep/2019

8:00am - 9:00am	**Registration**
9:00am - 11:00am Uniroma Tre	**SE-23: Day 4 Session 23** Session Chair: **Prof. Pieter Rousseau** Plenary room
9:00am - 11:00am Uniroma Tre	**SE-24: Day 4 Session 24** Session Chair: **Dr. Mirosław Seredyński** Room 20
9:00am - 11:00am Uniroma Tre	**SE-25: Day 4 Session 25** Session Chair: **Prof. Tadeusz Wójcik** Room 21
11:00am - 11:20am	**Coffee Break and Closing Greetings**

Other Abstracts

Presentations

KEY-01: Keynote Session 1

Time: Tuesday, 03/Sep/2019: 9:10am - 11:10am · *Location:* La Sapienza
Session Chair: Jan Taler

Plenary room

9:10am - 10:10am

Fluid flow control and simulation with lattice boltzmann methods: application to process and medical engineering

Mathias Krause

Karlsruhe Institute of Technology (KIT), Germany

An overall strategy for numerical simulations and optimisation of fluid flows is introduced. The integrative approach takes advantage of numerical simulation, high performance computing and newly developed mathematical optimisation techniques, all based on a mesoscopic model description and on Lattice Boltzmann Methods (LBM) as discretisation strategies. The resulting algorithms were implemented in a highly generic way in the framework of the open source library OpenLB (http://www.openlb.net). In the talk, the approaches and realisations are illustrated by means of various fluid flow simulation and optimisation examples. Thereby, particular focus is placed on particle-laden fluid flows for process engineering applications and flows in the human respiratory and cardiovascular system.

10:10am - 11:10am

Smart energy systems and 4th generation district heating

Henrik Lund

Energy Planning at Aalborg University, Denmark

Future sustainable national energy solutions may benefit substantially from including district heating and cooling as a measure of energy efficiency. However, district heating technologies will have to adjust to the needs of future smart energy systems in order to fulfil its role also known as 4th Generation District Heating (4GDH). Unlike the previous three generations, the development of 4GDH involves balancing the energy supply with energy conservation and thus meeting the challenge of supplying increasingly more energy efficient buildings with space heating and domestic hot water (DHW), while reducing losses in district heating (DH) grids. Furthermore, 4GDH involves strategic and innovative planning and the integration of DH into the operation of smart energy systems. Following a review of recent 4GDH research, this presentation quantifies the costs and benefits of 4GDH in future sustainable energy systems. Costs involve an upgrade of heating systems and of the operation of the distribution grids, while benefits are lower grid losses, a better utilization of low temperature heat sources and improved efficiency in the production compared to previous district heating systems. It is quantified how benefits exceed costs by a safe margin with the benefits of systems integration being the most important.

SE-01: Day 1 Session 1

Time: Tuesday, 03/Sep/2019: 11:30am - 1:30pm · *Location:* La Sapienza
Session Chair: Paweł Ocłoń

Plenary room

11:30am - 11:45am

Thermal characteristics and mathematical model of thin aluminum vapor chambers

Shuang Han[1,2], Lixin Yang[1,2]

[1]Beijing Jiaotong University; [2]Beijing Key Laboratory of Flow and Heat Transfer of Phase Changing in Micro and Small Scale

With the rapidly increasing of power densities of electronic components, thermal management problem of electronic components is increasingly crucial recently. Aluminum vapor chamber has the advantages of lightweight and high heat transfer performance. Aluminum vapor chamber is an effective way of thermal management.

This paper concerns on the thermal characteristics and mathematical model of thin aluminum vapor chambers. 2 mm thin aluminum vapor chambers with different charging ratio are manufactured. A thermal test system is established to analyze the thermal characteristics such as temperature difference and equivalent thermal conductivity at different heating power. The experimental results are analyzed to obtain the effect of charging ratio and heating power on the vapor chamber. At the same time, a mathematical model for the vapor chamber is established. The analytical results are consistent with the experimental results. The mathematical model provides a theoretical basis for the design of the aluminum vapor chamber.

11:45am - 12:00pm

Effect of solid heat conduction on thermoacoustic oscillations of supercritical n-decane flowing inside a miniature tube

Bo Ruan, Jiaqi Li, Xiaowei Gao

Dalian University of Technology, P. R. China

Themoacoustic oscillation is undesired in regenerative cooling systems of hypersonic vehicles because it will cause flow instability or even structure failure. It needs to be attenuated or be eliminated, however, the underlying mechanism of themoacoustic oscillation formation remains unclear yet.

In the present work, numerical model for simulating transient conjugate heat transfer of hydrocarbon fuels at supercritical pressures has been developed. Extended Corresponding-State Method is adopted to calculate supercritical fluid properties. The effect of solid heat conduction on thermoacoustic oscillations of n-decane is investigated. Results indicate that the solid heat capacity can buffer the fast heat transport process from the solid zone to the fluid zone, thus exert strong impacts on fluid flow and pressure oscillations. The peak-to-peak amplitude of thermoacoustic oscillating increases significantly with the decrease of solid wall thickness.

12:00pm - 12:15pm

Thermal stress analysis of an annular fin of hyperbolic profile using admonian double decomposition method

Balaram Kundu, Dulal Krishna Mandal

Jadavpur University, India

An analytical study on steady-state condition of an annular fin of hyperbolic profile with temperature dependent thermal conductivity has been carried out for the analysis for determination of temperature distribution in annular fins. Once the temperature distribution is known, the thermal stresses of the fins are calibrated using the theory of principal stresses in an axis symmetric radial system and the thermal stress distribution is determined by direct integration of the closed form expression of the temperature distribution. Admonian Double Decomposition Method has been used to solve the governing differential equation of heat transfer through the fin. Our primary aim is to find how the dimensionless temperature distribution and the thermal stresses change with the variation of thermal conductivity of the fin material. It has been observed that the nature of the radial thermal stress distribution is dependent on the special coordinate in the fin.

12:15pm - 12:30pm

Transient response of a plate-fin-and-tube heat exchanger considering different heat transfer coefficients in individual tube rows

Dawid Taler, Jan Taler, Katarzyna Wrona

Cracow Uniwersity of Technology, Poland

The relationships for the air-side Nusselt number on each tube row are needed to design a plate-fin and tube heat exchanger (PFTHE) with the optimum number of tube rows. The relationships for the air-side Nusselt number on each tube row are necessary. The paper presents a new method of modeling the transient operation of PFTHE, considering that the air-side Nusselt numbers of individual tube rows are calculated from different empirical relationships. A method of determining the heat transfer coefficient on individual tube rows was developed, using the results of the CFD modeling of the heat exchanger. A transient numerical model of a double-row, two-pass PFTHE was developed, considering different heat transfer coefficients in the first and second row of tubes. The influence of the adoption of different heat transfer coefficients on each tube row on the heat flow rate exchanged between fluids was shown.

12:30pm - 12:45pm

Heat Transfer and Fluid Flow analysis of porous media heat exchangers using Aluminium Oxide-water nanofluids

Anjan R Nair[3], Rajkumar M. R[2], Dijin J. S[1]

[1]College of Engineering Trivandrum, India; [2]College of Engineering Trivandrum, India; [3]College of Engineering Trivandrum, India

It has been well documented in many literature that porous media and nano-fluid can augment heat transfer in many engineering system. However the combined usage of these two media has not been given much attention in literature. The objective of the present work is to experimentally investigate porous media heat exchanger using Al2O3-water nano-fluids. The heat exchanger effectiveness is determined using transient testing method wherein, only one fluid stream flows steadily through the test core, then a transient perturbation in the inlet fluid temperature is induced and the outlet fluid temperature variation is measured continuously. The measured data is evaluated using the maximum slope method to obtain the heat transfer characteristics.

12:45pm - 1:00pm

Numerical investigation of the flow behavior of a novel fluidization based particle thermal energy storage

Verena Sulzgruber, David Wünsch, Heimo Walter, Markus Haider

TU Wien, Austria

At the TU Wien, Institute for Energy Systems and Thermodynamics a novel sensible thermal energy storage (TES), referred to as Fluidization Based Particle TES (FP-TES), has been developed. The FP-TES is working with bulk material as storage medium, which is transported from one hopper through a counter-current heat exchanger into another hopper by advanced fluidization technology without any mechanical transportation devices. To prove this concept of particle transport extensive numerical investigations with the CPFD (computational particle fluid dynamics) software Barracuda have been performed. Within the numerical investigations an optimized geometry for a cold test rig was developed and its behaviour as well as particle mass flow and pressure drops were predicted. With the cold test rig, working with 800 kg quartz sand, experimental investigations were performed and the results were compared to the prediction of the numerical investigations.

1:00pm - 1:15pm

Transport phenomenon and optimization of simultaneously developing flow and heat transfer in twisted sinusoidal wavy microchannel under pulsating inlet flow condition

Suvanjan Bhattacharyya[1], Sampad Gobinda Das[2], Himadri Chattopadhyay[3], M.A. Moghimi[4], Ali Cemal Benim[5]

[1]University of Pretoria, South Africa; [2]Jadavpur University, India; [3]Jadavpur University, India; [4]University of Pretoria, South Africa; [5]Duesseldorf University of Applied Sciences,Germany.

The transport phenomena in microchannels are significant in thermal equipment design. The current study investigates numerically the simultaneously developing unsteady laminar flow and heat transfer inside a twisted sinusoidal wavy microchannel. At the inlet sinusoidal varying velocity component is applied. Varying pulsating amplitude and frequency represented by the Strouhal number was study for Reynolds number ranging from 1 -100. Water is used as a working fluid. ANSYS Fluent 19.2 CFD package was used. The governing equations are solved with a finite volume based numerical method. Comparison with steady flow it was found that imposed sinusoidal velocity at the inlet can provide improved heat transfer performance at different amplitudes and frequencies while keeping the pressure drop within acceptable limits. To find the optimum working amplitude and frequency the problem went through a multi-objective optimization process for the proposed geometry to maximize its performance and Nusselt number while minimizing its friction factor.

SE-02: Day 1 Session 2

Time: Tuesday, 03/Sep/2019: 11:30am - 1:30pm · *Location:* La Sapienza
Session Chair: MingJia Li

Room 7

11:30am - 11:45am

Numerical and experimental investigations of Fractional Flow Reserve (FFR) in a stenosed coronary artery.

Supratim Saha, T Purushotham, Arul Prakash K
Indian Institute of Technology Madras, India

The coronary artery is numerically investigated based on CFD techniques for measuring the severity of stenosis. In mild cases, medication is often preferred whereas for severe cases surgical intervention is required but most cases fall in between.Thus it poses a problem for clinicians in choosing an appropriate action. The Fractional Flow Reserve (FFR) is a number which helps to predict the functional significance of stenosis in this scenario. In this study , various cases of percent blockage ranging between 40 to 70 are considered using different models for predicting FFR in the stenosed coronary artery. The lesion length is also varied between 1 cm and 5 cm based on patient-specific data .The experimental investigation of FFR in the coronary stenosis using silicon model is also carried out in this study and compared with numerical results.The effect of occlusion percentage and lesion length on the FFR value are quantified .

11:45am - 12:00pm

Numerical and experimental study on convective heat transfer characteristics in foams materials

Hongyan Lu[1,2], Lixin Yang[1,2], Zhiyong Wu[3]
[1]Beijing Jiaotong University; [2]Beijing Key Laboratory of Flow and Heat Transfer of Phase Changing in Micro and Small Scale; [3]The Key Laboratory of Solar Thermal Energy and Photovoltaic System

This paper reports the experimental measurement the temperature of foam by the single-blow transient testing technique. The specimens present different thickness, ranging from 30 to 105 mm, with varying number of pores per inch: 30,45 and 60 PPI and different porosities, ranging from 0.75 to 0.85. the test is carried out at a flow velocity ranging from 0.5 to 1.5 m/s. The fluid temperature distribution is obtained by processing the experimental data. Subsequently, a simplified model are used to analyze the heat characteristics.

By comparing the simulation with experiment, the regularity of heat transfer coefficient and the Local Thermal Non-Equilibrium phenomenon in the foam material are analyzed and discussed, a correlation of the convective heat transfer coefficient is obtained, which is related to the porosity, PPI and thickness of the foam. The conclusion can be used to guide the optimization of the design of foam material-mediated heat exchange equipment.

12:00pm - 12:15pm

Modelling and simulation of wool steaming for the efficient management of decatizing processes

Fabiana Federica Ferro, Mirco Rampazzo, Alessandro Beghi
Università degli Studi di Padova, Italy

Finishing is one of the fundamental steps of textile production and still, nowadays it largely depends on empirical knowledge. Aim of finishing processes is to impart the required functional properties to the fabric and, in particular, decatizing is the process that lends the fabrics dimensional stability, enhances the luster and improves the so-called ‘fabric hand’, corresponding to the sense of touching a textile. In this paper, we consider wool fabrics and, by exploiting the available process physical knowledge, we derive a model that can predict certain fabric characteristics, such as its temperature and moisture content, correlated with the fabric dimensional stability. We also design a simulation environment according to the model and we use it to easily generate synthetic data, obtaining information about the steaming process under different conditions. By analyzing the data, we can obtain knowledge about how to maximize the fabric decatizing process efficiency.

12:15pm - 12:30pm

Natural convection combined with surface radiation in a rotating cavity with an element of variable volumetric heat generation

Stepan Mikhailenko[1], Mikhail Sheremet[1], Ioan Pop[2]
[1]Tomsk State University, Russian Federation; [2]Babes-Bolyai University, Romania

The aim of the present investigation is a numerical analysis of natural convection and surface radiation within a square rotating chamber having a local element of internal variable volumetric heat generation.

The domain of interest is a square rotating cavity with thermally insulated horizontal walls and active vertical ones. A local element of variable volumetric heat generation is placed on the adiabatic wall. The wall surfaces and heater surface are assumed to be grey diffusive emitters and reflectors of radiation. The fluid flow and heat transfer are described using two-dimensional Oberbeck–Boussinesq equations in the case of rotating chamber under the impacts of surface thermal radiation and heat-generating element. The effects of the Taylor number, surface emissivity, and volumetric heat flux oscillation frequency on the heat transfer performance have been studied.

This work was supported by the Russian Science Foundation (Project No. 17-79-20141).

12:30pm - 12:45pm

The multi-domain model of solute transport in binary alloy

Mirosław Seredyński, **Jerzy Banaszek**

Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Institute of Heat Engineering, Nowowiejska Str. 21/25, 00-665 Warsaw, Poland

A three-domain model of solute transport is proposed in the framework of the front tracking based micro-macroscale binary alloy solidification model, earlier developed by the authors. This multi-domain approach is used only for the solute transport equations, whereas the phase mixture model is maintained for the momentum and energy transfer within the two-phase region. Solute conservation equations are averaged across solid and liquid phases, and the solute transport at the phase interface is included. Additionally, the microstructure evolution is modelled to capture the development of various complex grain structures and more accurately describe the solute transport between the phases. The proposed extended model is then used in the example problem of Al-Cu alloy solidification in a 2D mould, and the obtained fields of solute concentration and macro-segregation pictures are compared with those predicted by the fully single-domain model. Thus, the role of non-equilibrium solute transport is identified and discussed.

12:45pm - 1:00pm

Direct numerical simulations of natural convection flows in a cubical enclosure with differentially heated opposite walls

Alexandre Fabregat Tomas, **Jordi Pallares Curto**

Rovira i Virgili University, Spain

Understanding the transport and deposition of particles and aerosols in enclosed cavities is key to determine what are the preferential deposition locations and at which rate particles accumulate. This information may be crucial in designing better buildings, public spaces and display cases as well as in optimizing heating/ventilation systems. Direct Numerical Simulations have been used to investigate the dispersion of particles in a prototypical configuration consisting in a cubical cavity where the bottom and one side are kept at a constant temperature higher than that of the top and opposite wall. The remaining two lateral sides are thermally insulated. Fully-resolved simulations provided flow statistics at several Rayleigh numbers ranging from steady laminar to fully turbulent regimes (1e5 ≤ Ra ≤ 5.4e8). The resulting database can be used to investigate the dispersion of particles of different sizes and determine the dominating forces that control the deposition process on each surface.

1:00pm - 1:15pm

Numerical investigations of film-cooling flow structures in a fan-shaped hole

Ali Zamiri, **Jin Taek Chung**

Korea University, Korea, Republic of (South Korea)

Advanced gas turbines have been extensively used in aero engines industry. In order to achieve higher power and higher thermal efficiency in gas turbines, the higher turbine inlet temperature is required. Therefore, effective blade cooling techniques are required to reduce the surface temperature of the turbine components. One of these techniques is film cooling method where the cooler air is injected through the film holes.

The aim of this study is to conduct a comprehensive assessment of the LES (large eddy simulation) turbulent model to predict the film-cooling process of a cross-flow through a fan-shaped hole using ANSYS CFX V18.0. The numerical approach is validated by the experimental data for different blowing ratios. The transient pressure disturbances along the fan-shaped hole and mainstream channel were evaluated. It was shown that the presented LES method could accurately predict the heat transfer characteristics and capture the small turbulence structures.

1:15pm - 1:30pm

Numerical investigation on influence of sea-current velocity profile on the propagation characteristics of hazardous noxious substances spilled from the ship

Min Kyu Ko[1], **Chan Ho Jeong**[2], **Moonjin Lee**[3], **Seong Hyuk Lee**[4]

[1]Chung-Ang University, Korea, Republic of (South Korea); [2]Chung-Ang University, Korea, Republic of (South Korea); [3]Korea Research Institute of Ships & Ocean Engineering (South Korea); [4]Chung-Ang University, Korea, Republic of (South Korea)

The purpose of this study is to numerically analyze the influence of sea-current velocity on near-field propagation of hazardous noxious substances (HNS) spilled from the ship. The commercial code ANSYS FLUENT (V. 17.2) was used to analyze the behavior of the spilled HNS. This study investigated the effect of sea-current velocity profile including surface current velocity as well as deep current velocity considering different surface layer depths. In particular, based on the CFD results we suggested the engineering model for predicting the HNS propagation. With 104 scenarios, a number of simulations were carried out, and three important parameters were newly introduced quantitatively. We used the well-known Kriging model for making engineering model. It is expected that the engineering model would provide useful information on practically managing the risk from HNS spill without CFD simulations.

SE-03: Day 1 Session 3

Time: Tuesday, 03/Sep/2019: 11:30am - 1:30pm · *Location:* La Sapienza
Session Chair: Mathias Krause
Room 17

11:30am - 11:45am

Study on the Influence Law of Sensor Material Characteristics on Thermal Environment Measurement

Lei Zeng[1], **Mingsong Ding**[1], **Bo Qiu**[1], **Wengang Zong**[2], **Yewei Gui**[1]
[1]China Aerodynamics Research and Development Center; [2]Sichuan University

In order to verify the effectiveness of the aerodynamic thermal environment prediction and test methods, some flight tests have been carried out. There always were some differences between test results and predicted results, especially under condition that “embedded” sensors with metallic material are installed on the surface of TPS with composite material.

In this paper, unsteady aerothermal and thermal-conduction coupling analysis method has been developed. The flow field and local thermal environment distribution under condition of temperature difference and catalytic characteristics difference between the sensor surface and the TPS of aircraft are calculated.

The results show that the surface heatflux obtained by the coupling calculation method agrees well with the measurement data, and the mechanism of the thermal environment distribution distortion is obtained. This study has deepened the understanding of the mechanism of local convective heat transfer and provided support for analyzing the measurement heatflux data and sensor design.

11:45am - 12:00pm

Effective numerical methods for calculating the non-stationary heat processes in multiphase media

Nadezda Nikolaevna Ermolaeva, **Galina Ibragimovna Kurbatova**
St Petersburg University, Russian Federation

For the long offshore gas pipelines the computation of the total heat flux from an ambient in every cross section of the gas pipeline involves solution of different non-stationary heat problems in multiphase media, in particular, the one-dimensional single- and two-phase problem of a cylindrical lateral surface glaciation in seawater and in bottom soil; the two-dimensional single-phase heat conduction problems in the cylindrical coordinate system in the absence of axial symmetry. In the present work a new numerical method with increased accuracy was proposed to solve these problems. The results of comparative analysis of the different numerical techniques effectiveness of solution to the two-dimensional single-phase heat conduction problems in the bottom soil in the absence of axial symmetry are also presented. The analysis was performed on the base obtained analytical solution of the two-dimensional heat conduction problems for steady-state conditions.

12:00pm - 12:15pm

Repeated Richardson extrapolation with verification of the type and forms of applying boundary conditions in CFD

Fabiana de Fatima Giacomini, **Josafa Magno dos Santos**
Technological Federal University of Parana, Brazil

The objective is to verify the effect on the discretization error and its order caused by the type and form to apply the boundary conditions in Computational Fluid Dynamics. It is considered: one-dimensional diffusion and advection equations; four interest variables with numerical approximations of first and second orders; Dirichlet and Neumann types of boundary conditions and the form of apply them with and without ghost volume. The a priori estimate of the order is based on the Taylor series. The a posteriori estimate is based on the discretization error obtained through Repeated Richardson Extrapolation (RRE). The results expected: i) the mixing between numerical approximations do not change the effectiveness of RRE; ii) the accuracy order obtained a posteriori corroborates the formal order obtained a priori; iii) RRE provides subsidies for cases in which it is not possible to estimate a priori or a posteriori the orders of the numerical error.

12:15pm - 12:30pm

Numerical study of fluid flow and heat transfer in a straight microchannel with standing surface acoustic waves

Sining Li[1], **Jianping Cheng**[1], **Hongna Zhang**[1,2], **Weihua Cai**[1], **Fengchen Li**[2]
[1]Harbin Institute of Technology, China, People's Republic of; [2]Sun Yat-sen University, Zhuhai, China,People's Republic of

In the past decade there has been a markedly increasing interest in applying acoustofluidics as a tool in MEMS systems. In this numerical study, standing surface acoustic waves (SSAW) are considered as a potential disruptive flow technology for enhancing heat transfer in microchannel. Using COMSOL, 2D numerical simulations are performed. In detail, this paper employ perturbation theory and use the solution of the first-order equations to calculate acoustic streaming induced in the first step. In the second step the Conjugate Heat Transfer interface is used to calculate time-averaged second-order flow and temperature in straight microchannel. In this paper, we elucidate the underlying mechanisms of phenomena peculiar to excitation of fluids in microchannels including the appearance of vortices and concomitant mixing and further to enhance heat transfer. The results show that acoustic streaming can disrupt the bulk fluid flow to create rotating vortices in the microchannel and enhancing heat transfer.

12:30pm - 12:45pm

Correlation between geometry and temperature distribution of chicken house using Computaional Fluid Dynamics (CFD)

Chia-Ti Tseng[1], Jia-Kun Chen[1], Tzu-I Tseng[2]

[1]National Taiwan University, Taiwan; [2]National Center for High-Performance Computing

The airflow and temperature in chicken cages directly affect the health of chickens. The aim of the study is to design a flow makes the egg-laying hens work in a comfortable workplace using computational fluid dynamics (CFD).

According to the field sampling, we found that the temperature difference between the front and back section of the chicken house was about 2℃. The relative humidity increased gradually when the cool air flowed through the chicken house. The temperature had the 2℃ difference between inside and outside cage. The chickens in the cages were affected by the temperature drop, and the egg quality and the production were also affected. We would model the space by the thinking of the porous media.

We built the model of porous media model for 40000 chickens in the cages. Base on the model, the flow and temperature would test for the problem finding and improvement suggestion.

12:45pm - 1:00pm

Numerical estimation of thermal load in a three blade vertically agitated mixer

Anshul Singh Tomar, Harish K G, K Arul Prakash

Indian Institute of Technology Madras, India

The objective of this study is to understand the flow physics and resulting heat transfer behind mixing of highly viscous fluid in a vertical three blade mixer which consist of central agitator rotating in counter clockwise direction and other two periphery blades rotating in clockwise direction. Mixing of highly viscous solid/liquid ingredients is a complex phenomenon which has many applications like solid propellant mixing of rocket fuel in space industries. During this mixing process there is enormous amount of heat generation due to viscous dissipation which may result in pre combustion or fire hazards. A detailed CFD study is carried out to quantify the temperature rise due to shearing action of solid propellant. An Open source toolbox named OpenFOAM is used to carry out simulations and a co-relation is proposed to calculate temperature rise as a function of time and viscosity for a given range of angular velocity.

1:00pm - 1:15pm

Numerical analysis of multiphase flow through axial vortex tube cyclone separator

Gopalakrishnan B, Arul Prakash K

IIT Madras, India

Axial cyclone separators are used as primary filtration components in many internal combustion engine applications. A numerical study of gas-solid flow through an axial cyclone separator is carried out by Eulerian-Lagrangian CFD approach, which uses discrete phase modelling (DPM) of particles and solution of incompressible turbulent Navier Stokes equations with RNG $k-\varepsilon$ closure model. The effective filtration of inlet air from dust particles is important for prolonged engine life cycle. In axial separators, a helical swirl generator is used for imparting swirling motion to the particle laden flow and particles are filtered by centrifugal separation and then scavenged using a vortex tube. In the present work, the modelling done in OpenFOAM successfully captures the features of the swirling flow, along with particle-gas and particle-wall interactions for a varying size distribution at the inlet. Studies were aimed at analysing the effect of particles and improving the performance of the filter.

1:15pm - 1:30pm

Evaluation of Human Thermal Comfort Model in Residential Artificial Environment

Hongbo Xu, Mingsheng Tang

Technical Institute of Physics and Chemistry, CAS, China, People's Republic of

In recent years, with the development of network technology and the improvement of hardware computing ability, intelligent wearing equipment has developed rapidly. Individual users' physical parameters like local skin temperature can be easily measured by intelligent bracelet, which can provide long-term observation data for individual users' physiological indicators and subjective thermal response. According to Gagge's two-node human body model, this paper established a program for calculating the core temperature, skin temperature and skin wetness, and measures the wrist skin temperature through intelligent bracelet, which can be used as the basis for real-time adjustment of the user's metabolic rate and other parameters in the calculation model. The core temperature, skin temperature, skin humidity and local skin temperature of human wrist measured by smart bracelet were calculated by the model as indicators of user's thermal sensation, so as to adjust the setting of indoor thermal and humid environment parameters.

SE-04: Day 1 Session 4

Time: Tuesday, 03/Sep/2019: 2:20pm - 4:20pm · *Location:* La Sapienza
Session Chair: Sandra Corasaniti

Plenary room

2:20pm - 2:35pm

Experiment and Numerical investigation on Flow and Heat Transfer Characteristics in a Horizontal Long small channel of plate OTSG

Xiaofei Yuan[1], Lixin Yang[2]

[1]Beijing Jiaotong University; [2]Beijing Key Laboratory of Flow and Heat Transfer of Phase Changing in Micro and Small Scale

The plate OTSG are employed to improve the compactness of nuclear power system.

In this paper, The semi-circular channel in aluminum experimental section has an effective heating length of 700 mm and a channel diameter of 3 mm. The working fluid was deionized water. The experiments were conducted with the conditions of inlet pressure in the range of 0.1~4 MPa, mass flux in the range of 100-600 kg/m2-s, and the outlet vapor quality in the range of 0.1 to 1. At a certain mass flow rate, the local heat transfer coefficient increased with the increase of the heat flux and operating pressure.

The drift-flux models were adopted to test the applicability of existing drift flux correlations for current experimental phenomena prediction. In the whole, the simulation results of the drift flow models showed that the trend of heat transfer coefficients along the path were consistent with the experimental results.

2:35pm - 2:50pm

Numerical investigation on flow and heat transfer characteristics in the 17×17 full-scale fuel assembly

Zihao Tian[1,2], Lixin Yang[1,2]

[1]Institute of Thermal Engineering, School of Mechanical, Electronic and Control Engineering, Beijing Jiaotong University, Beijing, China; [2]Beijing Key Laboratory of Flow and Heat Transfer of Phase Changing in Micro and Small Scale, Beijing, China

In previous study, a large number of computational fluid dynamics (CFD) simulations of fuel assembly thermal-hydraulic problems were presented that contained less fuel rods such as 3×3 and 5×5 due to the limit of the computer capacity. However, a typical AFA-3G fuel assembly consists 17×17 rods. This study concerned on the appropriate CFD method for a full-scale 17×17 fuel assembly. The spacer grids with mixing vanes, springs and dimples are considered. The polyhedral and extruded mesh was generated by Star-CCM+ software and the total mesh number is about 200 million. The lateral velocity and temperature filed distribution in sub-channels was investigated. The pressure drop of each spacer grid was compared to the experiment results. As a result, an appropriate method of full-scale rod bundle simulations was obtained. The CFD analysis of thermal-hydraulic problems in reactor coolant system can be conducted widely by using real size fuel assembly models.

2:50pm - 3:05pm

Performance analysis and tube inlet orifice length evaluation of a once-through steam generator

Hun Sik Han, Han-Ok Kang, Juhyeon Yoon, Young In Kim, Youngmin Bae, Sang Ji Kim

Korea Atomic Energy Research Institute, Korea, Republic of (South Korea)

A numerical study is conducted on the secondary side screw-type tube inlet orifice design of a once-through steam generator. An orifice length criterion for flow stabilization is derived by introducing the hydraulic resistance ratio of the orifice and the subcooled region to the two-phase and superheated regions. Various tube plugging conditions and power levels are considered, and the secondary coolant flow rate is adjusted to maintain a constant thermal power. The steam generator performance is analyzed according to the tube plugging condition in terms of the degree of superheat and secondary side pressure drop. The secondary coolant flow rate curve for the constant thermal power operation is obtained, and the required minimum orifice length to suppress the flow oscillation below the allowable level is evaluated. The lowest power level results in the highest minimum orifice length, and non-plugging condition provides a limiting case for the orifice length criterion.

3:05pm - 3:20pm

Computational prediction of two-phase flow in passive residual heat removal system of an integral reactor

Joo Hyung Moon, June Woo Kee, Seungyeob Ryu

Korea Atomic Energy Research Institute, Korea, Republic of (South Korea)

When an accident occurs in a nuclear power plant, residual heat of the reactor core shall be removed to keep the reactor in a safe shutdown condition. The passive residual heat removal system (PRHRS) is one of the passive safety systems, which removes the residual heat using natural convection heat transfer. In the modelling of a closed loop of natural circulation, an imaginary hydraulic head is employed to separate the effects of both the saturated vapor and saturated liquid from that of the two-phase flow. By balancing the imaginary hydraulic head difference between the Steam Generator and PRHRS Heat Exchanger, the region of saturated liquid in the steam line, and that of saturated vapor in the feedwater line can be determined. In the present study, the transient behavior of the temperature of the reactor coolant system and the cooling performance of the PRHRS are numerically examined.

3:20pm - 3:35pm

A Numerical Investigation of Radiation Feedback in Different Regimes of Opposed Flow Flame Spread

Kenneth Dong, Subrata Bhattacharjee

San Diego State University, United States of America

Radiation has been found to play an important role in opposed-flow flame spread, especially in the low-velocity microgravity environment. To explore the various aspects of flame radiation, an existing comprehensive 2D computational model including gas and surface radiation as well as radiation feedback to the solid is utilized. The comprehensive radiation model is simplified into a number of sub-models: no radiation, gas-only, surface-only, uncoupled (gas and surface without feedback). The sub-models are evaluated over the kinetic, thermal, and radiative regimes and the resulting spread rates, flame and vaporization temperatures, and flame structures are compared to the comprehensive fully coupled model. The computational results reveal that gas-to-surface feedback largely cancels surface radiation losses, leading to reasonably accurate results when only gas radiation is considered. The loss cancelling effect is strongest at low opposed-flow velocities and becomes weaker as the opposing flow increases into the kinetic regime.

3:35pm - 3:50pm

Modeling of flame-generated turbulence and counter-gradient diffusion in stagnating turbulent premixed flames

Ahmed Neche

University of Blida1, Algeria

Turbulent premixed flames impinging onto a wall are modeled and simulated with a single step irreversible reaction. The attention is given to an algebraic model closure of the turbulent fluxes of the fuel mass fraction in lean premixed methane-air flames.

The turbulent transport is analyzed that for a sufficiently high turbulence level, the turbulent transport is of the gradient transport type for the reacting scalar c and when the turbulence level remains low, the thermal expansion due to heat release dominates the process of turbulent scalar transport and the turbulent transport is of a counter gradient turbulent transport.

A numerical simulation is carried out in order to validate the model. The computational results are validated against the experimental measurements done by Cho et al. (1988).

3:50pm - 4:05pm

Modeling of shaped charge jet formation: comparative studies of the numerical approach.

Bartosz Fikus, Paweł Płatek, Jacek Janiszewski

Faculty of Mechatronics and Aerospace, Military University of Technology, Warsaw, Poland

The explosively formed projectiles are classified as one of the most effective kinds of ammunition which allows perforation of high mechanical strength structures. The main aim of this paper is to present the results of theoretical investigations related to the jet formation phenomenon with various numerical approaches considered. Comparative analyses were conducted with application of the arbitrary selected geometry and materials forming the shaped charge. Taking into account an important character of deformation under dynamic loading conditions, the Eulerian and meshless Lagrangian (SPH) methods were proposed to describe the media motion of models with the axial and planar symmetry. Computer simulations were conducted based on two commercial packages – ANSYS AUTODYN and LSTC LS-DYNA. Moreover, the influence of crucial numerical parameters describing the simulation process was estimated. Based on the obtained results, the comparative analyses were conducted. Owing to the literature data, validation of the considered problem was possible.

4:05pm - 4:20pm

A new simulation model for a grate firing system in openfoam

Frank Ulrich Rückert[1], Daniel Lehser Pfeffermann[1], Danjana Theis[1], Ju Pyo Kim[2], Andre Schargen[2], Ingo Zorbach[2], Jens Sohnemann[2]

[1]University of Applied Sciences Saarbrücken, Germany; [2]Steinmüller Babcock Environment GmbH, Germany

The incineration of household-waste or biomass represents, in comparison to other renewable energy sources, a cheap and technically feasible short-term option to substitute fossil fuel and to reduce the emission of carbon dioxide. Additionally German government has set the goal to increase the thermal use of household-wastes in order to reduce the high volume of export for plastic waste. It can be expected that additional plastic waste have to be treated. Modern grate-firing systems, both in retrofit or new plants, are expected to expand significantly in Germany and other European countries in the upcoming years.

For design of modern grate firing-systems, the simulation with CFD (computational fluid dynamics) is necessary. Because of this, a new Euler/Euler-model for movement of the fuel bed on the grate was developed and implemented into the open-source code OpenFOAM. Description of heat-transfer coupled with chemical reactions and drying is possible.

SE-05: Day 1 Session 5

Time: Tuesday, 03/Sep/2019: 2:20pm - 4:20pm · *Location:* La Sapienza
Session Chair: Ali Cemal Benim
Room 7

2:20pm - 2:35pm

A realistic vapour phase heat transfer model for the weathering of LNG stored in large tanks

Felipe Huerta, Velisa Vesovic
Department of Earth Science and Engineering, Imperial College London, London, SW7 2AZ, UK

A new non-equilibrium model relevant to LNG weathering in large storage tanks under constant pressure has been developed. It treats the heat influx from the surroundings into the vapour and liquid phases separately. The results of this work indicate that the advective upward flow dominates the energy transfer within the vapour, while the natural convection, in the body of the vapour, can be neglected. The vapour temperature increases as a function of the height, in agreement with recent experimental results, while the vapour to liquid heat transfer is small and contributes less than 0.3% to boil-off gas rates. The initial liquid filling has a pronounced effect on all the relevant variables, leading to a decrease in vapour temperature and an increase in boil-off rates. The model captures all dominant physics and is capable of simulating a year of weathering in less than 30 seconds on a single core.

2:35pm - 2:50pm

Experimental investigation on effect of helical grooves on condensation heat transfer performance of vertically oriented copper tubes

Jyothish Abraham[1], Venugopal G[2], Rajkumar M R[1]
[1]College of Engineering Trivandrum, India; [2]Government Engineering College Thrissur

Condensation is a heat transfer process involving phase change and is commonly used in industry for applications like power generation, thermal management, desalination and air conditioning. Most of the research that has been done to improve condensation heat transfer is by bringing changes in surface morphology at nanometer or micrometer level. A much more durable and low cost alternative is to introduce millimeter level features on the condensation surface. The present work attempts to reduce the condensate film thickness and hence improving heat transfer, while orienting the condenser tubes in vertical position by providing helical grooves on copper tube surface. The experiment is conducted for various degrees of subcoling, flow rates and helical groove pitches. The experiment aims at developing a correlation between condensation heat transfer coefficient and pitch of helical groove and also finding out an optimum pitch which gives the best condensation heat transfer coefficient.

2:50pm - 3:05pm

Effect of gas spring on the stable operation criteria of free-piston stirling engine

Chin-Hsiang Cheng, Shang-Ting Huang
Department of Aeronautics and Astronautics, National Chegn Kung University, Taiwan

The free-piston Stirling engine(FPSE) is an external combustion engine which the main moving parts, a piston and a displacer are connected by a series of mechanical springs and gas springs. In this study, the effect of gas springs in the displacer chamber and the bounce space chamber on the stable operation criteria of FPSE integrated with linear alternator has been analyzed by a model consists of thermodynamic model, dynamic model and electric circuit model. It is found that the variation of operation parameters, such as temperature ratio, charging pressure and volume of gas spring chamber, change the operation state of FPSE between non-startable, over stroke and stable operation state. By changing the combination of the load of alternator and the operation parameters, the stable operation regions have been clarified under certain practical operation situation of FPSE.

3:05pm - 3:20pm

Effect of the engine speed and the loading on the heat transfer within exhaust valves

Mahfoudh Cerdoun[1], Farsaoui Bouziane[1], Smail Khalfallah[1], Rafik Ankri[1], Carlo Carcasci[2]
[1]Ecole Militaire Polytechnique, Algeria; [2]University of Florence,Italy

The aim of the present paper is to investigate numerically the heat transfer within exhaust valves by considering the actual boundary conditions at different load and speed. For this purpose, the valve is subdivided into seven adequate subdivisions to better assess the effect of each engine parts, therefore, an average value of the transient heat transfer coefficient (HTC) and the adiabatic wall temperature (AWT) for each subdivisions are evaluated during one cycle. The simulations are done at diverse engine regime, and therefore, the trend of the real boundary condition in term of HTC and AWT are given versus engine speed at different load. The findings show that the HTC increases linearly with engine speed however, the AWT decrease slightly at partial load and increase in the case of full engine load. The obtained model is used to highlight the temperature map, which help to avoid damage to the exhaust valve.

3:20pm - 3:35pm

Three dimensional CFD modelling of thermal-lag engine

Chin-Hsiang Cheng, Duc-Thuan Phung
National Cheng Kung University, Taiwan

Thermal-lag engines share some striking advantages in common with traditional Stirling engines such as quiet operation, heat-source versatility, and low risk of explosion. More than that, the thermal-lag engines contain only one moving part. Consequently, they have simpler structure; and require few seal rings and less lubrication than traditional Stirling engines. There arise demands for understanding the behavior of this kind of engines. This paper studies the thermodynamic performance of a thermal-lag

engine. In order to facilitate the thermodynamic analysis of the thermal-lag engine, a three-dimensional CFD model is developed and ANSYS Fluent is used to solve the governing equations for flow and thermal fields. The position of the piston is defined by a given velocity function and imported into ANSYS Fluent via a UDF file. The parametric study is performed by adjusting values of some parameters such as the rotation speed, heating and cooling temperature about the baseline case.

3:35pm - 3:50pm

Performance evaluation of a beta-type Stirling engine based on CFD simulation

Chin-Hsiang Cheng, Duc-Thuan Phung
National Cheng Kung University, Taiwan

In this paper, the thermodynamic performance of a beta-type Stirling engine is investigated using ANSYS Fluent. In the CFD model, the mass, momentum, and energy equations are invoked to obtain the flow and thermal behaviors of the working gas. The existence of a two-phase medium in the regenerator requires special treatments; that is, the Darcy-Forchheimer model is used to describe the momentum transport meanwhile the local thermal equilibrium model is solved for the temperature field. The flow in the engine is assumed to be fully turbulent and the k-ε realizable turbulent model is used. The positions of displacer and piston are determined by integrating their velocity via UDF files. A baseline case is chosen based on given geometrical and operating parameters. The cyclic variations of temperature, pressure, and heat transfer rates of the baseline case are evaluated. Then, the engine performance is elucidated by the parametric study.

3:50pm - 4:05pm

Thermal transport and melting characteristics of carbon based phase change nanocomposites

Harish Sivasankaran, Yasuyuki Takata
Kyushu University, Japan

The utilization of latent heat thermal energy storage system (LHTES) based on phase change materials (PCMs) has obtained a substantial attention among various types of thermal energy storage systems. Due to lower thermal conductivity of phase change material, practical application of such system is limited. Such limitations can be overcome by seeding nanomaterials of high thermal conductivity. In the present study, we numerically investigate the phase change behaviour of nanocomposites in vertical shell-tube thermal energy storage systems. Various nano-carbon allotropes such as nano diamond (spherical), single walled carbon nanotubes (one dimensional) and graphene nanosheets (two-dimensional) are considered as the thermal conductivity enhancers. Two dimensional numerical simulation of the melting phenomena was carried out with the enthalpy-porosity approximation. The role of interfacial thermal transport between the carbon based nanostructure and the surrounding PCM is taken into consideration using effective medium formulations in the numerical calculations.

4:05pm - 4:20pm

Heat transfer characteristics of circular, elliptical, mixed tube and elliptical tube with splitter plate cross flow heat exchanger

Aneesh K Mohanan, Prasad B V S S S, Vengadesan S, Arul Prakash K
Indian Institute Of Technology Madras, India

A numerical investigation of heat transfer and fluid flow characteristics were performed for a number of heat exchangers with circular, elliptical, mixed tube bundle comprising of first row of circular tubes followed by elliptical tubes and elliptical tubes with splitter plate. The tubes have staggered arrangement and are subjected to cross flow of air with the Reynolds number ranging from 5500 to 14500. Details of heat transfer, friction factor, goodness factor and the effect of longitudinal and transverse pitch-to- diameter ratio are evaluated. The provision of the splitter plate arrangement on elliptical tubes improves the Nusselt number for the fluid flow albeit with a marginal increase in friction factor. The overall thermal performance of elliptical tube heat exchanger in terms of goodness factor is better as compared to the other cross-sections.

SE-06: Day 1 Session 6

Time: Tuesday, 03/Sep/2019: 2:20pm - 4:20pm · *Location:* La Sapienza
Session Chair: Emanuele Habib

Room 17

2:20pm – 2:35pm

Investigation of multi-stage micro turbine with labyrinth seal using computational fluid dynamics

Yunseok Ha, Jungwan Kim, Yongbok Lee

Korea Institute of Science and Technology, Korea, Republic of (South Korea)

Several studies have shown that a multi-stage micro turbine is efficient in low-grade heat source, but there is a lack of research on the performance of multi-stage micro turbine according to seal configuration. This paper conducts a computational fluid dynamic (CFD) analysis quantifying the effects of the labyrinth seal. The amount of leakage flow rate and turbine efficiency were predicted through the analytical model. It also considered an environment in which the labyrinth seal is predominant while changing the properties of working fluid, pressure difference and rotational speed of the rotor. Predictions show the amount of leakage flow rate decreased when the labyrinth seal was applied to the blade containing a shroud. However, in the absence of the shroud on the blade, the labyrinth seal disrupted the main flow and rather reduced the turbine efficiency.

2:35pm – 2:50pm

Analysis of the Proton-Exchange Membrane Fuel Cell in transient operation

Andrzej Tadeusz Wilk, Daniel Paweł Węcel

Silesian University of Technology, Poland

Currently, fuel cells are increasingly used in industrial installations, means of transport and household applications as a source of electricity and heat. The paper presents the results of experimental tests of PEMFC at variable load, which characterizes the cell's operation in real installations. The measurements made show changes in the performance of the fuel cell during step changing or smooth changing of an electric load. Load was carried out as a change in the current or a change in the resistance of the receiver. The analysis covered the times of reaching steady states and the efficiency of the fuel cell system taking into account additional devices. The analysis of the measurement results will allow to determine the possibility of using fuel cells in installations with a rapidly changing load profile and indicate possible solutions to improve the performance of the installation.

2:50pm – 3:05pm

Computational study of active flow control drag reduction for utility vehicle

Jędrzej Mosiężny[1,2], Bartosz Ziegler[1]

[1]Poznań University of Technology, Poland; [2]Redos Trailers Sp. Z o.o

The study presents a computational study of a drag reduction device for a generic truck-trailer utility road vehicle based on an active boundary layer control. The conceptual device is in accordance with upcoming EU regulations regarding attachable aerodynamic devices for heavy utility vehicles.

Design and principles of operation of the conceptual device are presented. The device is intended to increase decrease the trailer's base drag coefficient by manipulation of the separated flow region behind the vehicle base.

Results of a steady state Reynolds averaged analysis and Delayed Detached Eddy Simulation are presented to show the discrepancies of fluid flow patterns between baseline and augmented configuration.

Results for drag reduction for baseline truck-trailer configuration and aerodynamically augmented configuration are presented.

3:05pm – 3:20pm

Analysis of the impact of coal quality on the heat transfer distribution in a high ash pulverized coal boiler using co simulation

Pieter Gerhardus Rousseau, Ryno Laubscher

University of Cape Town, South Africa

The high scattering efficiency and low emissivity of ash particles entrained in the flue gas of boilers burning high ash coal can adversely affect the heat absorption in the furnace and radiant superheater heat exchangers. This, together with the lower calorific value can lead to lower bulk furnace temperatures, lower heat absorption in the furnace and higher tube wall temperatures in the radiative superheaters. This paper presents the results of a numerical study that compares the heat transfer characteristics of a subcritical boiler firing coal with a very high ash content to that of the same boiler burning the original design coal. The analysis is based on a 1D discretized two-phase flow model of the water in the evaporator and radiative superheaters using Flownex SE 8.9. This is coupled with a detailed furnace combustion and heat transfer CFD model using Fluent 19.2 with custom developed code to calculate the variable particle properties.

3:20pm – 3:35pm

Energy-based modelling and simulation of liquid immersion cooling systems

Michele Lionello[1], Mirco Rampazzo[1], Alessandro Beghi[1], Damiano Varagnolo[2], Mattias Vesterlund[3]

[1]University of Padova, Italy; [2]Norwegian University of Science and Technology, Norway; [3]Swedish Institute of Computer Science North, Sweden

Currently, most of existing data centers use chilled air to remove the heat produced by the servers. However, liquids have generally better heat dissipation capabilities than air, thus liquid cooling systems are expected to become a standard choice in future data centers. Designing and managing these cooling units benefit from having control-oriented models that can accurately describe the thermal status of both the coolant and the heat sources.

The aim of this work is to derive a control-oriented model of liquid immersion cooling systems, i.e., systems where servers are directly immersed in a dielectric fluid having good heat transfer properties. More specifically, we derive a general lumped-parameters gray box dynamical model that describes energy and mass transfer that occur between the main components of the system. The proposed model has been validated against experimental data gathered during the operation of a proof-of-concept liquid immersion cooling unit, showing good approximation capabilities.

3:35pm – 3:50pm

One d dynamic thermal model of receiver tube for linear resnel reflectors

Roberto Tascioni[1,2], Luca Cioccolanti[2], Pero Pili[3], Roberto Manca[3], Emanuele Habib[1]

[1]Sapienza University of Rome, Italy; [2]Università Telematica eCampus, Italy; [3]Elianto S.R.L., Italy

One of the most challenging components in concentrated solar power systems is the receiver tube. Indeed it has to assure high optical properties whilst reducing the thermal losses as much as possible. In this study, a dynamic model of the receiver tube used in a small-scale Linear Fresnel Reflectors solar field for residential applications is presented. The solar field is part of a micro Combined Heating and Power plant (CHP) developed within the EU funded project 'Innova MicroSolar' by a consortium of academic and industrial partners. Here, a 1D model of the collector receiver is presented. Simulations developed in MATLAB, are validated with data of existing collectors reported in literature and compared to the provided technical specifications of the receiver installed at the solar field under investigation. Results show that the simplified thermal analysis is suitable for the simulation of the whole system.

3:50pm – 4:05pm

Individual pump flow control method of booster pump system

Kyungwuk Kim[1], Md Rakibuzzaman[1,2], Sang-Ho Suh[1,3]

[1]Department of Mechanical Engineering, Soongsil University, Seoul, Korea; [2]Department of Mechanical Engineering, Soongsil University, Seoul, Korea; [3]Department of Mechanical Engineering, Soongsil University, Seoul, Korea

The booster pump system can control the number of revolutions through an inverter by combining two or more vertical or horizontal centrifugal pumps connected in a series. The objective of this study is to find a performance evaluation method. Further to obtain the performance characteristics, instead of the method of measuring the flow rate by installing the flow meter on the discharge side pipe of the pump, the flow sensor was inserted into the check valve installed on the discharge side. The developed flow sensor measured accurately the performance characteristics of the individual pump in the pump system. Instead of efficiency, after obtaining by the performance test. The power consumption of the booster pump system in combination with daily flow patterns and duty cycles was investigated. The amounts of energy savings by the control method by the field test were 5.4% and 6.2%, respectively.

4:05pm – 4:20pm

Numerical simulation of natural convection heat transfer between circular cylinder located inside wavy enclosure filled with ag-nanofluid superposed saturated-porous nanofluid layers

Hasan shakir Majdi, Ammar Abdulkadhim Fathi, Azher M Abed

AlMustaqbal University, Iraq

Numerical investigation is presented for natural convection heat transfer between inner circular cylinder located inside wavy enclosure filled with two layers. The dimensionless governing equations of heat transfer (mass, energy and momentum of the fluid) are solved numerically using the Galerkin weighted residual method. The considered dimensionless parameters are Rayleigh number (103≤Ra≤106), Darcy number (10-1≤Da≤10-5), radius of inner cylinder ($0.2 \le R \le 0.4$), inner circular cylinder vertical position ($-0.2 \le \delta \le +0.2$), the number of undulation number ($0 \le N \le 3$) and nanoparticle volume fraction ($0 \le \varphi \le 0.1$). The results indicated that increasing the inner cylinder radius reduces the fluid flow strength. Also, when the cylinders moves vertically downward ($\delta = -0.2$) gives the better fluid flow strength and heat transfer characteristics. Finally, at high value of Rayleigh number, it is obtained that when the number of undulation is N = 1 gives better heat transfer enhancement.

KEY-02: Keynote Session 2

Time: Wednesday, 04/Sep/2019: 9:10am – 11:10am · *Location:* Uniroma Tre
Session Chair: Abdulmajeed Mohamad

Plenary room

9:10am - 10:10am

Recent developments of lattice Boltzmann methods for complex transport phenomena

Giacomo Falcucci

Tor Vergata University, Italy

We shall present recent developments of the Lattice Boltzmann method for a variety of transport phenomena in complex fluids, including, among others, the rheology of soft flowing crystals in microfluidic devices and heat transport in nano fluids.

10:10am - 11:10am

A numerical and experimental analysis of multiphysical effects on microalgae growth in photobioreactors

MingJia Li

Xi'an Jiaotong University, China, People's Republic of

In the photosynthesis process of microalgae growth, light energy is converted into biomass energy that sequestrating CO2 in the produced carbohydrate. Microalgae cultivation becomes a promising solution to limit CO2 emission and produce renewable energy because of its high efficiency and outstanding productivity. There is a complex multi-physical process for the microalgae cultivation in photobioreactors (PBRs). The gas-liquid multiphase flow in the cultivation medium carries the microalgae cells to circulate in the PBR with non-uniform light intensity distributions. The concentration of nutrients and CO2, as well as the light history of cells, influence the growth of microalgae simultaneously. In order to analyze the influences of multiphysical effects on the growth of microalgae and carbon sequestration, a comprehensive model including the liquid-gas multiphase flow, light distribution, microalgae cell motion and growth kinetics should be established. In this work, the comprehensive multi-physical model for the simulation of microalgae growth in PBRs is presented. The free-surface lattice Boltzmann model is developed to simulate the bubble flows. The radiative transfer equation with collimated light irradiation is solved by a discrete ordinate model to provide the light intensity distribution. The microalgae cell motion is simulated by a Lagrangian tracking model. The combination of the above models provided the light history of microalgae cells. By coupling with kinetic models, the growth of biomass concentration is finally obtained. Meanwhile, a temporal extrapolation scheme is proposed to bridge the gap between time scales for flow simulation in several seconds and the microalgae growth in hours and days. The analyses about the interaction between mass and light transportation in the PBRs are presented, and their influences on the growth of microalgae will be discussed. Experiments are further conducted to validate the proposed multi-physical models and the optimization of microalgae cultivation. An optimal nutrients supplementary strategy is proposed to optimize the growth rate by maintaining the optimal concentration of nitrogen and phosphorus. Finally, the evaluation system of flow fields in PRBs is established based on the above numerical and experimental results that include the integrating parameters such as turbulent kinetic energy, dead zone area, and gas holdup. Based on the evaluation criteria, the flow field of plate PRBs and column PRBs are optimized. In summary, this study explains the influences of multiphysical effects on microalgae growth in PBRs. It provides guidance to the optimal ambiance of carbon sequestration and the future applications of PRBs.

SE-07: Day 2 Session 7

Time: Wednesday, 04/Sep/2019: 11:30am - 1:30pm · *Location:* Uniroma Tre
Session Chair: Marek Jaszczur

Plenary room

11:30am – 11:45am

Some accuracy related aspects in two-fluid hydrodynamic sub-grid modeling of gas-solid riser flows

Joseph Mouallem[1], Seyed Reza Amini Niaki[2], Norman Chavez Cussy[3], Christian Costa Milioli[3], Fernando Eduardo Milioli[3]

[1]Department of mechanical and mechatronics engineering, University of Waterloo, Canada; [2]Institute of Mathematics and Computer Sciences, University of São Paulo, Brazil; [3]Department of Mechanical Engineering, School of Engineering of São Carlos, University of São Paulo, Brazil

Sub-grid closures for filtered two-fluid models (fTFM) useful in large scale simulations (LSS) of riser flows can be derived from highly resolved simulations (HRS) with microscopic two-fluid modeling (mTFM). Accurate sub-grid closures require accurate mTFM formulations as well as accurate correlation of relevant filtered parameters to suitable independent variables. This article deals with both of those issues. The accuracy of mTFM is touched by assessing the impact of gas sub-grid turbulence over HRS filtered predictions. A gas turbulence alike effect is artificially inserted by means of a stochastic forcing procedure implemented in the physical space over the momentum conservation equation of the gas phase. The correlation issue is touched by introducing a three-filtered variable correlation analysis (three-marker analysis) performed under a variety of different macro-scale conditions typical or risers. While the more elaborated correlation procedure clearly improved accuracy, accounting for gas sub-grid turbulence had no significant impact over predictions.

11:45am – 12:00pm

Lattice Boltzmann simulation for liquid-gas-solid flow based on phase-field model

Qiang He, Weifeng Huang

Tsinghua University, China, People's Republic of

Based on phase-field theory, we develop a lattice Boltzmann model for liquid-gas-solid flow from multiphase and particle dynamics algorithms. A modified momentum exchange (ME) method is developed for the velocity-based LB approach. A curved boundary treatment with second-order accuracy based on velocity interpolation is developed. We propose a predictor-corrector scheme algorithm for specifying the three-phase contact angle on curved boundaries within the framework of structured Cartesian grids. In order to make the algorithm more stable, we combine the implicit particle velocity update scheme and the modified ME method. The proposed method is validated through several single-and multi-component fluid test cases. It was found the surface tension force associated with the interface acting on the solid structures can be captured. We simulate the sinking of a circular cylinder due to gravity, the numerical results agree well with the experimental data.

12:00pm – 12:15pm

Evaluation of droplet drag force model on spray dynamics and thermal ingdom of R134a two-phase flashing spray

Zhifu Zhou[1], Dongqing Zhu[1], Guanyu Lu[1], Weitao Wu[2], Yubai Li[3], Bin Chen[1]

[1]State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an, 710049, China; [2]School of Mechanical Engineering, Nanjing University of Science and Technology, Nanjing 210094, China; [3]Department of Mechanical Engineering, Carnegie Mellon University, Pittsburgh, PA 15213, USA

This paper comparatively evaluates the performance of a selected number of droplet drag force models in predicting droplet evolution for single droplet and droplets in flashing spray. The studies span from a single, isolated R134a droplet that evaporates in a convective environment, to a fully turbulent, flashing spray formed through a accidental release of high pressure R134a liquid. The effect of nozzle diameter on spray dynamics and thermal behavior of R134a two-phase flashing spray is also examined. The result indicates that most of the drag force models have little effect on droplet diameter, velocity and temperature distributions in both single isolated droplet evaporation modeling and fully two-phase flashing spray simulation, except the Khan-Richardson model. The nozzle diameter greatly affects R134a spray morphology and tip penetration, that smaller diameter contributes to a larger radial expansion at the near nozzle region while leading to a much smaller penetration distance.

12:15pm – 12:30pm

Numerical study on the Reynolds number effects in a prismatic tank using natural frequency of the prismatic shapes

Hyunjong Kim[1], Parthasarathy Nanjundan[2], Yeon Won Lee[2]

[1]Korea Atomic Energy Research Institute, Daejon, Republic of (South Korea); [2]Department of Mechanical Design Engineering, Pukyong National University, Busan, Republic of (South Korea)

In this numerical study, a 2D prismatic tank – subjected under horizontal excitation – is used to analyze the sloshing characteristics for a specific range of Reynolds number, from 2.5x104 to 2.0x105. Three models of geometric variable δ1 – namely, δ1=50mm, δ1=150mm, and δ1 =250mm – is used to observe the effects of the lower chamfered shape of the tank, where the Reynolds number for each δ1 ranges from 2.5x104 to 2.0x105 (12 Cases). The Fast Fourier Transforms (FFT) technique is used to analyze the frequency components of the excitation force and the magnitude of the amplitude spectrum. The results show that the sloshing wave fluctuation becomes small when the geometric variable δ1 is larger than 150 mm. Also, the FFT technique shows that the resonance does not occur due to frequencies which are not the integral multiple of the excitation frequency.

12:30pm – 12:45pm

Numerical Study of Sensitivity of Turbulent Jet Scour to porosity and internal friction angle.

Mariana Mendina Gourgues, Gabriel Usera Velasco

Facultad de Ingeniería, Universidad de la República, Uruguay

In this work a computational model was used to simulate local scour at a sand bed impinged by a circular turbulent submerged jet. The CFD base model used is the caffa3d.MBRi. This model implements finite volume method in curved block structured meshes for incompressible, viscous or turbulent flows, where the fluid-particle system is considered as a pseudo-continuous material of a single phase. The presence of particles is taken into account in terms of an effective viscosity.

Different jet conditions were simulated and results were compared with experimental values previously obtained by other authors. Finally, the sensitivity of the model to some parameters (porosity and internal friction angle) was studied. The results achieved for these application examples, show that the model adequately represents the different physical processes involved in these cases, reaching a good quatitative match with the results previously reported in the bibliography.

12:45pm – 1:00pm

Experimental investigation of flow boiling heat transfer in mini-channel heat sink.

Hyeong-geun Kim, Sung-min Kim

Univ. Sungkyunkwan, Korea, Republic of (South Korea)

Thermal management systems using phase change of the fluid are attracting attention, given their ability to achieve very high heat transfer coefficient. Because of the miniaturization of electronic devices pursued by various industry fields, the required heat dissipation rate per unit surface area has increased with increasing power density of the devices. Flow boiling is one of the major thermal solutions for many applications demanding high-flux heat removal, such as military avionics, aerospace, radar, and electric vehicle power electronics. In this study, two-phase flow experiments are performed in the copper heat sink consisted of an array of rectangular mini-channels, using FC-72 as a working fluid. The FC-72 flow visualization, pressure drop, and heat transfer coefficient data will be provided.

1:00pm – 1:15pm

Simulation of single water droplet evaporation in limited space

Bin Liu, Yang Li, Aiqiang Chen, Georges El Achkar

Tianjin University of Commerce, People’s Republic of China

The process of single water droplet in limited space from the room temperature to zero temperature was simulated and the evaporation process of the fog droplet sprayed by the nozzle was analyzed. By considering the latent heat and the convection heat caused by the droplet motion, the effect of the latent heat was replaced by the inner heat source, and the volume of the droplet was calculated by the dynamic grids. As for the inner temperature of the droplet and the room temperature were simulated by the laminar flow heat transfer model. The effect of the factors including the relative humidity, the room temperature, the droplet diameter, the initial temperature of the droplet and the velocity of the droplet were studied. It is found that the critical dimeter of the droplet will be increased with the deceasing of the relative humidly.

1:15pm – 1:30pm

Numerical analysis of a turbocharger compressor.

Kristaq Hazizi[1], Ahad Ramezanpour[1], Aaron Costall[2], Mehrdad Asadi[1]

[1]Anglia Ruskin University, United Kingdom; [2]Imperial College London, United ingdom

The automotive industry is under obligation to meet regulations for emission control that has resulted in further use of turbochargers to enable downsizing and increase engine power density. A set of numerical simulations are conducted along two turbocharger compressor speed lines of 150,000 rpm and 80,000 rpm and k-omega SST turbulence model is used to solve the compressible flow using ANSYS Fluent software. Three points on each speed-line are selected: one point each in regions close to surge and choke and a point in the stable zone. The simulations predict compressor performance in terms of the total-to-total pressure ratio and total-to-total efficiency. Results reveal the predicted pressure ratio error is in the range of 1-6%. In all cases, the predicted efficiency increased when a finer mesh is used.

In conclusion, the finer mesh leads to higher pressure ratio and efficiency values that overpredict the performance, especially for the point close to choke.

SE-08: Day 2 Session 8

Time: Wednesday, 04/Sep/2019: 11:30am – 1:30pm · *Location:* Uniroma Tre
Session Chair: Rachid Bennacer

Room 20

11:30am - 11:45am

Strength analysis on edge sealing glass using microwave

Jae Kyung Kim[1], Euy Sik Jeon[1,2]

[1]Industrial Technology Research Institute, Kongju National University, Korea, Republic of (South Korea); [2]Department of Mechanical Engineering, Graduate School, Kongju National University, Korea, Republic of (South Korea)

Recently, there has been a rapid increase in interest in vacuum insulating glass having excellent heat insulating performance. Edge sealing of glass technology as a core process technology have methods using glass frit and mixed gas of hydrogen. The method of using glass frit has a problem that the strength decreases due to the difference in thermal expansion coefficient between frit and glass. The method of using mixed gas of hydrogen has a limit in vacuum glass fabrication due to the deflection shape.

Microwave has been applied in many industrial fields. The microwave has a advantage that the operation time can be greatly shortened, uniform heating and heating only a specific portion. In this paper, heating and edge sealing of glass were carried out using microwaves. Bending strength test was carried out on sealed glass, and the effect of sealed thickness at the glass edge on strength was analyzed.

11:45am - 12:00pm

Macro-segregation Modeling of Eutectic Solidification using Lattice Boltzmann Method

Himadri Chattopadhyay, Runa Samanta, Chandan Guha

JADAVPUR UNIVERSITY, India

Prediction of the macro-segregation of the metal alloy has been important issue in casting. In recent years, the understanding of eutectic solidification in Al-Si based casting alloys has drawn a great attention due its significant effect on final microstructure, casting defects, and mechanical properties. Evaluation of solidification interface and re-melting dynamics are linked to product quality

The Lattice Boltzmann Method is coupled with the energy and solute conservation equations and a modified LB model is used for solution of governing equations. This work considers a top cooled rectangular cavity considering re-melting phenomenon in mushy region during Al-Si solidification. The top wall of the cavity is maintained at lower temperature (Tc) than the bottom wall temperature (Th)to create a thermal convection in the cavity. The eutectic temperature (Te) has been kept between the higher and lower boundary temperature (Th<Te<Tc) of domain. The side walls are maintained at adiabatic condition. The schematic of the cavity is shown in fig. 1.

The Rayleigh-Benard convective cells interacting with solid-liquid interface at the flow domain has been studied with different Rayleigh number. The temperature distribution, solid fractions and the streamlines patterns are predicted and the results show important characteristics of re-melting problems of Al-Si alloy in rectangular cavity.

Key words: Eutectic solution, solidification and re-melting, macro-segregation, lattice Boltzmann method, simulation.

12:00pm - 12:15pm

Numerical solutions for the Carreau thin film flow and heat transfer over an unsteady stretching sheet

Kohilavani Naganthran, Roslinda Nazar, Ishak Hashim

Universiti Kebangsaan Malaysia, Malaysia

The present paper probed the problem of thin liquid film flow and heat transfer in the Carreau fluid along a permeable stretching sheet. The similarity transformation reduced the partial differential equations into a system of ordinary differential equations which was then solved numerically by a collocation method. The influence of the governing parameters towards the model was discussed and presented graphically. The dual solutions are observable when Pr = 1.

12:15pm - 12:30pm

Modelling drying kinetics of Black Soldier Fly (Hermetia illucens, L.) larvae

Pascoal da Silva[1,3], Nuno Ribeiro[2], Maria Nazare Coelho Pinheiro[1,4], Rui Costa[2]

[1]Instituto Politécnico de Coimbra, Instituto Superior de Engenharia de Coimbra, Rua Pedro Nunes, Quinta da Nora, 3030-199 Coimbra, Portugal; [2]CERNAS, Instituto Politécnico de Coimbra, Escola Superior Agrária, Bencanta, 3045-601, Coimbra, Portugal; [3]Centro de Matemática da Universidade de Coimbra, Apartado 3008, EC Santa Cruz,3001 - 501 Coimbra, Portugal; [4]Centro de Estudos de Fenómenos de Transporte, Faculdade de Engenharia da Universidade do Porto, Rua Dr. Roberto Frias, 4200-465 Porto, Portugal

Hermetia illucens (Black Soldier Fly) larvae (BSFL) have great potential as new protein source for either feed or food use due to the high content of protein and fat. BSFL may be used in different forms including dried with extended shelf life. A mathematical model describing dehydration kinetics is useful in BSFL drying conditions optimization.

A non-stationary 3D diffusion model, given by Fick's Law was used to describe the moisture depletion inside BSFL during drying. The numerical solution of PDE was obtained with COMSOL Multiphysics assuming simple boundary conditions.

Data were obtained in a drying experiment performed with fresh larvae uniformly laid on a metallic grid in a forced air oven at 80 □C. Five randomly larvae were periodically selected and removed for moisture content determination until equilibrium moisture was reached.

Good agreement between predictions and data was observed corroborating that main physical phenomena were considered in the mathematical description.

12:30pm - 12:45pm

Double Diffusive Magnetohydrodynamics Casson Fluid Flow over a Shrinking Sheet with Soret and Dufour Effects

Siti Suzilliana Putri Mohamed Isa[1,2], Shahanaz Parvin[1], Amira Yusoff[3], Norihan Md. Arifin[1,3]

[1]Institute for Mathematical Research, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor Darul Ehsan, MALAYSIA.; [2]Centre of Foundation Studies for Agricultural Science, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor Darul Ehsan, MALAYSIA.; [3]Department of Mathematics, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor Darul Ehsan, MALAYSIA.

The mathematical formulation of double diffusive mixed convection boundary layer flow in magnetohydrodynamics Casson fluid caused by a shrinking sheet is developed. This model is subjected by the presence of Soret and Dufour effects. The variations of shrinking velocity, wall temperature and wall concentration are assumed to have exponential function forms. Non-similarity transformation is applied on the governing basic equations (flow, momentum, energy and concentration equations) before they are solved numerically using bvp4c MATLAB program. The numerical results of velocity, temperature and concentration profiles, together with skin friction coefficient, local Nusselt number and local Sherwood number are presented in the form of tables and graphs. On the other hand, stability analysis is performed to select the stable solution due to the occurrence of dual numerical solutions.

12:45pm - 1:00pm

Sensitivity of numerical predictions to the permeability coefficient in simulations of melting and solidification using the enthalpy-porosity method

Amin Ebrahimi[1], Chris R. Kleijn[2], Ian M. Richardson[1]

[1]Department of Materials Science and Engineering, Delft University of Technology, Mekelweg 2, 2628 CD Delft, The Netherlands; [2]Department of Chemical Engineering, Delft University of Technology, van der Maasweg 9, 2629 HZ Delft, The Netherlands

The high degree of uncertainty and conflicting literature data on the value of the permeability coefficient (also known as the mushy zone constant) is a critical drawback when using the fixed-grid enthalpy-porosity technique for modelling phase-change processes. In the present study, the sensitivity of numerical predictions to the value of this coefficient is scrutinised. Using finite-volume based numerical simulations of isothermal and non-isothermal melting and solidification problems, the causes of increased sensitivity are identified. It is found that depending on the mushy-zone thickness and the velocity field, the solid-liquid interface morphology and the rate of phase-change are sensitive to the permeability coefficient. It is demonstrated that numerical predictions of an isothermal phase-change problem are independent of the permeability coefficient for sufficiently fine meshes. It is also shown that sensitivity to the choice of permeability coefficient can be assessed by means of an appropriately defined Péclet number.

1:00pm - 1:15pm

The numerical study of the effects of LES/RANS turbulent models to estimate flow structure within bubble column reactor (BCR)

Mojtaba Goraki Fard, Youssef Stiriba, Xavier F. Grau

Rovira i Virgili University (URV), Spain

The CFD simulation of a cylindrical BCR (contained air-water system at 24°c) in superficial gas velocities (6 and 8.4 cm/s) to evaluate the effects of LES and RANS turbulent model on flow structure within the BCR is investigated. The reactor diameter, height and the initial level of the water are 0.1, 2, 1.35m respectively. Two fluid approach of Euler-Euler in Open FOAM software package is used to model the flow pattern inside the reactor. The RANS turbulent model (Mixture k-ϵ) as well as the LES turbulent models (Smagoranski, SmagoranskiZahang and NicenoKEqn models) are examined. CFD results such as global and local volume of fraction and the local value of liquid velocities for the bulk zone of the reactor are examined and compared with experiment. It can be deduced that the results of Mixture k-ϵ and NicenoKEqn models have a good agreement with experiments specifically at the center of BCR.

1:15pm - 1:30pm

On numerical investigation of Nusselt distribution profile of heat sink using lateral impingement of air jet

Umair Mohammed Siddique

SVKM's Narsee Monjee Institute of Mangament Studies, India

The looming world of electronic packaging and material processing industries needs a non-uniform cooling of product. Generally, the cooling of heat sink is achieved by impinging air jet over it. As far as the non-uniformity in the cooling rate is concerned, lateral geometric thickness and thermo-physical properties of target surface plays a vital role in its contribution. Study of previous research works avails immense gap in the area of characteristic heat transfer augmentation study. Looking into this, the present work takes an assignment to justify the degree of non-uniformity in the Nusselt distribution curve and its dependency on geometric thickness. After critical thickness of 0.5 mm, Nusselt profile seems to be saturated and constant throughout the radial distance. Also an inverse variation is observed between the magnitude of area averaged Nusselt number and non-dimensional geometric thickness (t/d) which is applicable up till geometric thickness of 0.05.

SE-09: Day 2 Session 9

Time: Wednesday, 04/Sep/2019: 11:30am - 1:30pm · *Location:* Uniroma Tre
Session Chair: Sang-Ho Suh

Room 21

11:30am – 11:45am

The effects of exterior boundary conditions in tumor tissues fluid transport under heating conditions

Assunta Andreozzi[1], **Marcello Iasiello**[1], **Paolo Antonio Netti**[2]

[1]DII-Università di Napoli FEDERICO II, Italy; [2]DICMAPI-Università di Napoli FEDERICO II, Italy

The effects of mechanical conditions at tumor boundaries along with an internal heat source are investigated. Tumor is modeled as a deformable porous sphere, where the two phases are the interstitial fluid and the rest of the tumor with capillaries and tissues, respectively. Transient-state governing equations for mass, momentum and energy are written for both phases, by also considering tumor deformation under the linear elastic material assumption. A situation of Tumor Blood Flow (TBF) rapid decay is considered, while the heat source is modeled as a radial-decay function. Boundary conditions for the energy equation are varied on external surface of the sphere, in order to appreciate the effects of the surrounding on flow and temperature fields inside the tumor. After scaling equations, a finite-element scheme is employed for the numerical solution. Results are shown for different dimensionless parameters, showing in which case external boundary conditions strongly affect tumor interstitial flow.

11:45am – 12:00pm

Thermal effects on fluid flow through curved arteries

Marcello Iasiello[1], **Assunta Andreozzi**[1], **Nicola Bianco**[1], **Kambiz Vafai**[2]

[1]DII-Università di Napoli FEDERICO II, Italy; [2]University of California, Riverside, USA

Atherosclerotic plaque formation and aneurysms are substantially affected by the fluid flow through arteries and its subsequent effect on the shear stress. And so on. In some medical treatments like thermal ablation or endovenous ablation, heat transfer is also involved, and it can affect fluid flow due to the temperature-dependent of viscosity.

In this paper, heat transfer effects on fluid flow through curved arteries is computationally simulated. Coupled mass, momentum and energy equations are solved with the appropriate boundary conditions. Different rheological models are investigated by considering the viscosity variation with temperature. Results are presented for different size arteries and heating conditions, showing for which cases temperature has a remarkable effect on the flow field and in particular on shear stresses.

12:00pm – 12:15pm

A new concept of surgical patch used in vascular surgery

Natalia Lewandowska[1], **Michał Ciałkowski**[1], **Marcin Warot**[2]

[1]Poznan University of Technology, Poland; [2]Medical niversity of Poznan, Poland

During the surgery of atherosclerotic plaque removal, a longitudinal artery incision is made. Frequently used practice is sewing a patch in the incision area. It reduces the risk of narrowing of the artery but also causes its widening. The channel expansion leads to the enhancing separation of the boundary layer and vortexes creation. The selection of the patch size is based on the surgeon's experience. The purpose of the studies is to determine a new geometric concept of a patch. The resulting patch is fully parameterized and can be influenced by its geometric shape, adapting it to the patient's diameter. Simulations of blood flow in the artery were performed, with the patch used during the surgery and a patch developed by the authors. The analysis of the flow field in terms of the blood flow disturbances has shown, that the patch developed by authors give the most satisfactory results.

12:15pm – 12:30pm

The impact of the artery geometry with the sewn surgical patch on the blood flow disorders

Natalia Lewandowska, **Michał Ciałkowski**

Poznan University of Technology, Poland

The research concerns the development of geometric variants of patches sewn into the common carotid artery during surgery of the atherosclerotic plaques removal. Based on analytical methods, the geometry of the patch described by the polynomial function has been developed. The simulations of blood flow in the arteries with the sewn patch were performed. The study included the influence of the patient's diameter and the width of the chosen patch on blood flow disorders. The result of the research is the algorithm of selecting the geometry of the arterial patch to the individual geometrical features of the patient's artery. The studies result will comprise the development of software, which, upon introduction of input data related to arterial geometry, patch length and patient's blood parameters (affecting the fluid density and viscosity), shall generate an accurate contour of the patch of width causing no flow disorders.

12:30pm – 12:45pm

Measurements of the heat transfer rates during therapeutic hypothermia of a newborn's brain cooling

Dominika Bandoła[1], **Ziemowit Ostrowski**[1], **Marek Rojczyk**[1], **Wojciech Walas**[2], **Zenon Halaba**[3], **Andrzej J. Nowak**[1]

[1]Silesian University of Technology, Poland; [2]University Clinical Hospital in Opole, Poland; [3]University of Opole, Poland

This work presents measurements of the heat transfer rates, carried out during therapeutic hypothermia of a newborn's brain cooling. The most important is the heat flux transferred from the brain to the cooling water flowing through the cooling cap of the Olympic Cool-Cap system. Equally important is the heat flux exchanged by thermal radiation between neonate's skin and radiant warmer as well as surrounding walls. This heat rate is measured using home-made glob thermometer. Remaining thermal

measurements are carried out with the help of i-button technology. All measurements are recorded in course of therapy and send through GSM system to the central database which is secured by user account and password. In this way all measurements can be performed without any disturbance to the medical personnel when it is busy with therapy. Then these data are also processed and deeply analysed by neonatologists and/or engineers and complement developed computational models.

12:45pm – 1:00pm

A new simulation method for laser speckle imaging to investigate hemodynamics

Bin Chen, Xu Sang, Dong Li

Xi'an Jiaotong University, China, People's Republic of

Speckle simulation is a powerful protocol to investigate the properties of speckle and evaluate image processing method. However, only static speckle images can be simulated by available methods without considering time-integrated effect of CCD. A time-integrated dynamic speckle simulation method based on coherent imaging was developed. The effect of speckle size on LSCI was investigated through this method. The smaller the speckle size is, the higher the spatial resolution become. The existing speckle contrast imaging methods were compared, and higher statistical accuracy was achieved by stLASCA than those of sLASCA and tLASCA. Complex flow structures of blood flow in a rat dorsal skin window were simulated, and the effects of speckle size and spatial window length on speckle contrast was investigated. In general, the new simulation method for speckle imaging is a powerful tool to monitor blood flow in vivo and lay a solid foundation for the study of hemodynamics.

1:00pm – 1:15pm

Meshing strategy for bifurcation arteries in the context of blood flow simulation accuracy

Natalia Lewandowska, Jędrzej Mosiężny

Poznań University of Technology, Poland

The study presents a mesh dependency study for a carotid artery bifurcation geometry of a real-life specimen. The results of time averaged mass flow thru the arteries, pressure and velocity contours at artery outlets and wall shear stress are compared between a set of structured and unstructured meshes, with varying non-dimensional boundary layer first element thickness (y+) parameter.

A set of four meshes in total is considered: a full-hexagonal structured mesh, an unstructured tetrahedral mesh with prism inflation layer, both created for y+=1 and y+=50.

Apart from numerical results, overall mesh creation work time, overall analysis stability and convergence are compared with the mesh quality results: cell non-orthogonality, cell skew and aspect ratio.

Numerical results are validated against results of real-life CT examination performed in Poznań Medical University

1:15pm – 1:30pm

A temperature based model for measuring the performance of high-rate methane biofilters (HMBFs)

Samadhi Gunasekara, Patrick Hettiaratchi

Center for Environmental Engineering Research and Education (CEERE) and Department of Civil Engineering, University of Calgary, Alberta, Canada

High-rate Methane Biofiltration (HMBF) technology is a promising bioprocess to attenuate methane (CH_4) rich waste gas as an alternative to flaring and venting. Typically, the field performance of an HMBF is evaluated instantaneously through its CH_4 removal capacity, by manually measuring the CH_4 input and output. However, HMBF performance varies over time and is sensitive to atmospheric changes. To determine the CH_4 oxidation over extended periods of time, this research uses temperature as a proxy in remote, field HMBFs. A dynamic, physically-based model was developed to back-calculate CH_4 oxidation by using inlet flow conditions, filter-bed temperature and atmospheric temperature as inputs. Advective, convective, radiative and evaporative heat losses were considered to calculate the heat generated through CH_4 oxidation. The model was calibrated and validated with actual data obtained from an actively aerated field HMBF over 420 days. The model showed high agreement with the actual results obtained from the HMBF.

SE-10: Day 2 Session 10

Time: Wednesday, 04/Sep/2019: 11:30am – 1:30pm · *Location:* Uniroma Tre
Session Chair: Ivano Petracci
Room 22

11:30am - 11:45am

A numerical model for predicting the impact of porosity distribution within a Chemical Biological Radiological and Nuclear respirator canister

Samuel G. A. Wood[1], Nilanjan Chakraborty[1], Martin W. Smith[2], Mark J. Summers[2]
[1]Newcastle University, United Kingdom; [2]Defence Science and Technology Laboratory, United Kingdom

A numerical model has been developed to better understand the impact of particle packing distribution on the performance of an activated carbon bed inside a generic Chemical Biological Radiological and Nuclear (CBRN) canister filter canister, using Reynolds-Averaged Navier-Stokes (RANS) simulations. The model subdivides the bed into discrete sections in which the porosity is randomised using a Gaussian distribution around the expected longitudinally-averaged radial porosity profile. The impact of the length-scale of variations in the packing distribution, and the standard deviation of porosity is measured by considering the pressure drop and residence time distribution within the bed under steady-flow conditions. The range of particle sizes and distributions under which such a model will be appropriate is also considered

11:45am - 12:00pm

Analysis of field synergy in bottom heated lid driven cubical cavity

Rani HP[1], Narayana Vekamulla[1], Rameshwar Y[2]
[1]National Institute of Technology Warangal, India; [2]Osmania University, Hyderabad, India

In the present study, the analysis of three dimensional mixed convective lid driven cavity flow is investigated using numerical simulation. The bottom wall of the cavity is kept at a uniform higher temperature than that of the top moving lid and other walls are assumed to be thermally insulated. Finite volume method is employed to obtain solutions of the non-dimensional governing equations. Numerical results are computed for the control parameters arising in the system, namely, the Reynolds number (Re) and Richardson number (Ri) in the range of $100 \le Re \le 400$ and $0.001 \le Ri \le 10$.The contours of isotherms, streamlines and field synergy are used to visualize the flow and thermal characteristics. When Re is small, with increase of Ri, more synergy is observed in the vicinity of the boundary in comparison with other locations. When Re = 400 the above pattern is reversed as Ri increases.

12:00pm - 12:15pm

Numerical simulation of vortex induced vibrations of a circular cylinder: isothermal and heat transfer cases

Chandrakant Rameshchandra Sonawane[1], Yogesh B More[2], J C Mandal[1], AnandKumar Pandey[1]
[1]Symbiosis International University, India, India; [2]Indian Institute of Technology Bombay, Mumbai

In this paper, the FSI problem - vortex induced vibration of a circular cylinder are studied using a numerical method. An accurate Harten Lax and van Leer with contact for artificial compressibility (HLLC-AC) Riemann solver have been used for flow computation. The Riemann solver is modified to incorporate arbitrarily Lagrangian-Eulerian (ALE) formulation in order to take care of mesh movement in the computation, where radial basis function is used for dynamically moving the mesh. Higher order accuracy over unstructured meshes is achieved using quadratic solution reconstruction based on solution dependent weighted least squares (SDWLS). Two flow scenario: isothermal and heat transfer cases are presented here. The results obtained by the present method is compared with the literature

12:15pm - 12:30pm

Thermal Analyses on the cooling system of HYPROB LOX/LCH4 Demonstrator manufactured by means of electroplating process

Daniele Ricci[1], Francesco Battista[1], Vito Salvatore[1], Manrico Fragiacomo[1], Luca Manni[2]
[1]CIRA - Italian Aerospace Research Center, Italy; [2]CECOM srl - Via Tiburtina, km 18,700, 00012 - Guidonia Montecelio, Italy

The HYPROB Program, aiming at increasing system design and manufacturing capabilities on liquid oxygen-methane rocket engines (LOX/LCH4), foresees the design, manufacturing and test of a 3-tons-thrust ground demonstrator. The baseline concept includes 18 injectors and a regenerative cooling strategy using the engine propellant (liquid methane). The most critical component is the cooling jacket has 96 axial channels, generated by an inner liner (made with a copper alloy), and by a close-out structure (nickel). CFD analyses have been performed on both “methane cooled” and “water cooled” versions of DEMO cooling system to characterize its behavior and support the design activities. In fact, the “water cooled” version is needed to validate the cooling jacket, manufactured by electrodeposition innovative process, before the final firing test campaign. Results of CFD analyses have been taken into account as input for the thermo-structural simulations on the most critical sections to evaluate the thrust chamber assembly lifecycle.

12:30pm – 12:45pm

Effect of top wall confinement on onset of three-dimensionality in sinusoidal wavy channels

Harikrishnan Sankara Varrier, Shaligram Tiwari
IIT Madras, India

In the present study, three-dimensional numerical investigations are carried out to find the influence of top wall confinement on onset of three-dimensionality in sinusoidal wavy channels and its effect on flow and heat transfer characteristics. Different channel confinements considered in the present study are channels having sinusoidal bottom wall with plane, in-phase and out-of-phase top wall. Computations are carried out by using open source CFD package OpenFOAM. Numerical results are validated against those reported in literature. Onset of unsteady flow in the channel has been identified by monitoring velocity at a point in the domain. Flow and heat transfer characteristics have been presented with the help of instantaneous streamlines, velocity contours, Q structures, and isotherms.

12:45pm – 1:00pm

Magnetohydrodynamics (MHD) flow and heat transfer in a dual stratified micropolar fluid over a permeable shrinking/stretching sheet

Najiyah Safwa Khashi'ie[1,2], Norihan Md Arifin[1,3], Roslinda Nazar[4], Ezad Hafidz Hafidzuddin[5], Nadihah Wahi[3], Ioan Pop[6]
[1]Institute for Mathematical Research, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia; [2]Fakulti Teknologi Kejuruteraan Mekanikal dan Pembuatan, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia; [3]Department of Mathematics, Faculty of Science, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia; [4]School of Mathematical Sciences, Faculty of Science and Technology, Universiti Kebangsaan Malaysia, 43600 UKM Bangi, Selangor, Malaysia; [5]Centre of Foundation Studies for Agricultural Science, Universiti Putra Malaysia, 43400 UPM Serdang, Selangor, Malaysia; [6]Department of Mathematics, Babeş-Bolyai University, R-400084 Cluj-Napoca, Romania

The present study accentuates the MHD flow and heat transfer characteristics over a permeable stretching/shrinking sheet immersed in a dual stratified micropolar fluid. Thermal and solutal buoyancy forces are also included to incorporate with the stratification effect. The governing partial differential equations are converted into a set of nonlinear ordinary differential equations using suitable transformations. Using Matlab bvp4c, numerical results obtained for some limiting cases are in favorable agreement with the earlier published results. The velocity, angular velocity, temperature and concentration profiles within the boundary layer for the appropriate values of the magnetic parameter and wall mass suction parameter are visualized graphically. Dual solutions are found within a certain range of shrinking and suction parameters. The execution of stability analysis affirms that the first solution is physically realizable and stable.

1:00pm – 1:15pm

Compressible Flow Simulation of High Pressure Cold Gas Inflator to Analyze Filling Behavior of Different Gases and Control Valve Requirements for Automotive Airbags

Naveen Raghavendra Shirur, hristian Birkner
Technische Hochschule Ingolstadt, Germany

Present automotive airbags are inflated with nitrogen gas which is a major gaseous combustion product of sodium azide or guanidine nitrate solid propellants. When airbag is inflated with hot nitrogen gas, pressure of airbag rises quickly in 30 to 40 milliseconds to restrain occupant. What happens when same nitrogen gas is stored under high pressure and expanded through a control valve to fill airbag? This question is partly addressed in present work through 3D CFD simulations of geometrically equivalent control valve with high pressure and low pressure containers. Main objective of this research is compressible flow analysis of atmospheric air, helium and nitrogen stored at high pressure of 1000 bar and their filling or expansion behavior in a closed tank which is replica of closed airbag. Along with filling behavior, thermodynamic properties are analyzed, based on which, valve requirements and airbag material properties are discussed.

1:15pm – 1:30pm

A parametric study on cavitation flow through steam trap valves under high pressure difference

Chang Qiu[1], Zhi-xin Gao[1], Zhi-jiang Jin[1], Jin-yuan Qian[1,2,3], Bengt Sunden[3]
[1]Institute of Process Equipment, College of Energy Engineering, Zhejiang University, Hangzhou, 310027, PR China; [2]State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University, Hangzhou 310027, PR China; [3]Department of Energy Sciences, Lund University, P.O. Box 118, SE-22100 Lund, Sweden

Steam trap valves are mainly used in thermal power systems to pour out condensate water and keep steam inside. While flowing through steam trap valves, the condensate water is easy to reach cavitation, which may cause serious damage to the piping system. In this paper, in order to suppress cavitation inside steam trap valves, effects of valve body geometrical parameters including orifice diameter, installation angle and the thickness of the cage are investigated by using a cavitation model. The pressure, velocity and steam volume fraction distribution inside valves are analyzed and compared in different geometrical parameters. The total vapor volumes are also predicted and compared. The results show that geometrical parameters have a significant influence on the cavitation distribution and vapor volume. The optimal combination of these three parameters to suppress cavitation are found and verified. The work is of significance for the optimization design of steam trap valves.

SE-11: Day 2 Session 11

Time: Wednesday, 04/Sep/2019: 2:20pm – 4:20pm · *Location:* Uniroma Tre
Session Chair: Lubna Younis

Plenary room

2:20pm - 2:35pm

The interaction effects between droplets condensing from moist air using a distributed point sink method

Shaofei Zheng, Ferdinand Eimann, Christian Philipp, Tobias Fieback, Ulrich Gross
Institute of Thermal Engineering, TU Bergakademie Freiberg, Germany

For dropwise condensation of moist air, the vapor diffusion from the surrounding to the droplet surface will be tremendously influenced by the blocking effect of the neighboring droplets. The influenced spatial distribution of vapor totally determines a different condensation rate comparing with that for the droplet growing by an isolated way. In this work, a distributed point sink method (DPSM) belong to the method of Green's method is used to capture the interaction effect (namely the blocking effect) between droplets. With that, each condensation point is equivalent to a series of mass point sinks which are evenly distributed on an image concentric spherical surface. For validation, DPSM is firstly used to handle some simple droplet arrays. And then considering a representative droplet array during dropwise condensation, the results indicate that the interaction effect between droplets is critical in predicting the droplet condensation behavior in the presence of non-condensable gas.

2:35pm - 2:50pm

The wind test on heat loss from three coil cavity receiver for a parabolic dish collector

Ramola Sinha[2], Nitin. P. Gulhane[1], Paweł Ocłoń[3], Jan Taler[3], Rahimi Gorji Mohammad[4]
[1]VJTI, Matunga-Mumbai, India; [2]K.J.Somaiya College of Engineering, Mumbai, India; [3]Institute of Thermal Power Engineering Cracow University of Technology,Poland; [4]Ghent University, Belgium

A modified three coil solar cavity receiver of inner wall area approximately three times of single coil receiver, is experimentally investigated to study the effect of fluid inlet temperature (Tfi=50ºC to75 ºC) and cavity inclination angle (θ = 0º to 90º) on the heat loss from receiver under wind condition for head on wind and side on wind velocity at 3 m/s. Overall it was found that the heat loss from the receiver under wind condition is up to 25% higher (1.25 times) at 0º inclination, than without wind at mean fluid temperature at 70ºC and minimum 19.64 % (1.2 times) at 90º inclination. In horizontal position of the receiver, the heat loss by head on wind is 18% (1.23 times at 0º inclination) higher than side on wind. For receiver facing vertically downward, for head-on wind heat loss is approximately the same as that for side-on wind.

2:50pm - 3:05pm

Experimental assessment of the lumped lithium-ion battery model

Tanılay Özdemir, Ali Amini, Özgür Ekici, Murat Köksal
Hacettepe University, Turkey

In this study, an axisymmetric computational model of a cylindrical Li-Ion battery is used to investigate the effects of the concentration, activation, and ohmic overpotential terms on the thermal and electrical behavior of the battery at various discharging processes. A typical cylindrical cell consists of multiple spiral layers. However, the model employs the lumped battery interface approach in COMSOL multiphysics in which the multiple layers are approximated as a uniform material with effective properties. Among other parameters, the OCV-SoC curve is determined experimentally and used as input to the model. The model is then used to analyze the effects of the correlations of various parameters on the transient electrical and thermal responses of the battery and validated by comparing predicted results with experimental data. The model is subsequently utilized to examine the effects of adjusted parameters on the heat generation by considering the ohmic, entropic, mixing and charge transfer effects.

3:05pm - 3:20pm

Effect of inserted sphere size on heat transfer characteristics of FCC structured pebble bed in a HTGR

LeiSheng Chen[1], JaeYoung Lei[2]
[1]Shaanxi University of Science & Technology, China, People's Republic of; [2]School of Mechanical&Control Engineering, Handong Global University, Pohang, South Korea

Surface hot spots appearing in a high temperature gas-cooled reactor (HTGR) pebble-bed core has been considered as the most possible reason leading to a severe accident like fission products releasing, therefore, investigation on their positions and thus seeking ways to reduce the possibility of their appearance have attracted much attention. In our previous studies, heat transfer characteristics of a face-centered-cubic (FCC) structured pebble-bed have been discussed and a correlation on heat transfer coefficient with Reynolds number was presented. In this study, whether placing a small sphere in the gap area could enhance the convective heat transfer was investigated. It is concluded after analysis that (1) inserted sphere lowered the local surface temperature of adjacent pebbles;(2) maximum fluid velocity and average heat transfer coefficient increased with sphere diameter, particularly, comparing with no small sphere case, 12.95% enhancement was achieved. Such findings may help develop a safer HTGR pebble-bed core

3:20pm – 3:35pm

New method for determining the optimum fluid temperature when heating pressure thick-walled components with openings

Dawid Taler, Piotr Dzierwa, Jan Taler
Cracow Uniwersity of Technology, Poland

A new approximate method of optimum heating thick-walled cylindrical pressure elements weakened by holes was developed. Optimum variations in fluid temperature during heating were determined from the condition that the total circumferential stress at the edge of the opening, resulting from the thermal load and pressure is equal to the permissible stress. The permissible stress is determined from the Wöhler fatigue diagram for a given number of start-ups and shutdowns of a power unit from the cold state.

This method makes it possible to determine with high accuracy the temperature of the fluid for times close to zero, i.e., at the beginning of the heating process. In the second stage of heating, the optimum fluid temperature was determined with the assumption of a quasi-steady temperature field in the pressure element.

3:35pm – 3:50pm

Comparison of the character of flow between the horizontal and vertical configuration of heating boiler draughts in low power heating boilers

Wojciech ust, Bartosz Ciupek, Rafał Urbaniak
Chair of Thermal Engineering, Poznan University of Technology, Poland

Paper presents a numerical analysis of a heat transfer process during exhaust gas flow through two boiler draughts connected in the reversing chamber. Each boiler draught is composed of four parallel pipes. Heat transfer process occurs between the exhaust gas and cooling water, which surrounds exhaust gas ducts. During research, a comparison between the horizontal and vertical configuration of boiler draughts is analyzed. The aim of the proposed research is defining a character of a flow and a heat transfer process depending on the horizontal and vertical position of boiler draughts. The article shows the main differences in the exhaust gas flow through the boiler construction, when heat exchangers are composed of pipes. Fluid flow and heat transfer are analyzed basing on the amount of exhaust gas flow through the domain.

3:50pm – 4:05pm

Numerical study of a heat transfer process in a heating boiler for solid fuel equipped with afterburning chamber

Wojciech ust, Bartosz Ciupek, Rafał Urbaniak
Chair of Thermal Engineering, Poznan University of Technology, Poland

Analysis of a heat transfer process for a construction of solid fuel heating boiler equipped with additional afterburning chamber is presented. Analyzed construction of heating device is intended for house heating and preparation of hot utility water. Heat exchanger in analyzed boiler is composed of vertical tubes divided into three boiler draughts. Afterburning chamber connect main combusting chamber of heating boiler with second and third boiler draught. Prepared research incorporates model of heat transfer process for analyzed heating device. The aim of this analysis is to identify character of heat transfer and pressure drop of exhaust gases flow through the heating boiler. Study is realized for different heating power of heating boiler and amount of exhaust gases, which are flowing to the afterburning chamber directly from main combustion chamber.

4:05pm – 4:20pm

Research on dynamic processes of molten carbonate fuel cells

Arkadiusz Szczęśniak, Jarosław Milewski, Wojciech Bujalski
Warsaw University of Technology, Poland

The research presents results of research on transient states of molten carbonate fuel cells. The examination is based on the validated transient oriented model of a reference 1 kW class stack. The two main types of stack transient states were examined: (1) regular transient states, i.e. load following mode, (2) emergency conditions – i.e. malfunction of oxidizer and fuel supply system, sudden loss of electrical load and malfunction of individual fuel cells in the stack. The results show variations (profiles) of outlet gas temperatures, temperature gradient across the cell(s), gas composition etc. as a stack response. Obtained results enhance existing knowledge and may be used as source material for further research and development of MCFC generation units as a distributed power sources as well as development of control strategies for MCFC stacks.

SE-12: Day 2 Session 12

Time: Wednesday, 04/Sep/2019: 2:20pm – 4:20pm · *Location:* Uniroma Tre
Session Chair: Janusz T. Cieslinski

Room 20

2:20pm - 2:35pm

A numerical study on performance variation of transonic centrifugal impeller using a combination of flow cut and axial lift

Kun Sung Park, In Hyuk Jung, Sung Jin You, Seung Yeob Lee, Jin Taek Chung
Department of Mechanical Engineering, Korea University, Republic of Korea

In this study, the combination of Flow Cut and Axial Lift was proposed to change the performance of a centrifugal impeller. The Flow Cut is a method to reduce the flow rate by reducing the height of impeller shroud. The Axial Lift is a method to increase the total pressure by increasing the shroud height at impeller exit. A NASA CC3 transonic impeller was used as the base impeller and three-dimensional Reynolds-Averaged Navier-Stokes were solved to obtain the performance curve. When the Flow Cut was applied to the base impeller, the total pressure at the target flow rate was lower than the total pressure at the design point. The Axial Lift was applied to increase the reduced total pressure at target point to the design point level. As a result, the performance of the base impeller was changed to the target point using combination of Flow Cut and Axial Lift.

2:35pm - 2:50pm

Effect of axial & radial clearance on the performance of a positive displacement hydraulic turbine

Arihant Sonawat[1], Hyeon-Mo Yang[2], Young-Seok Choi[3], Kyung Min Kim[4], Jin-Hyuk Kim[5]
[1]University of Science & Technology, Korea, Republic of (South Korea); [2]Korea Institute of Industrial Technology, Korea, Republic of (South Korea); [3]Korea Institute of Industrial Technology, Korea, Republic of (South Korea); [4]Korea District Heating Corporation, Korea, Republic of (South Korea); [5]Korea Institute of Industrial Technology, Korea, Republic of (South Korea)

Renewable energy will play an important role in achieving the energy stability and reducing the dependence on fossil fuels. Hydropower is amongst cheapest and reliable renewable energy sources and potential of micro and Pico hydro power is not yet fully explored. They can prove to be decisive in generating clean energy from conventional and un-conventional resources and making the systems energy efficient for sustainable development. The present work illustrates the Pico hydro power generation potential of a new class of turbine known as Positive Displacement Turbine (PDT). Also, the effect of clearance on the overall performance of the PDT was analyzed using CFD. From the analysis, it was inferred that leakage flow from clearances significantly influenced the performance of PDT. A reduction in clearance from 0.15mm to 0.025mm caused an increase of 9.27 and 10.4% in hydraulic and volumetric efficiency respectively.

2:50pm - 3:05pm

An experimental and numerical analysis of the fluid flow in a mechanically agitated vessel

Marek Jaszczur, Anna Młynarczykowska, Luana Demurtas
AGH University of Science and Technology, Poland

The mixing process is a widespread phenomenon, which plays an essential role among number of industrial processes i.e. gas dispersion in liquids, chemical reactions, formation of suspensions and slow-sedimenting mixtures, prevention of sediment aggregation. The effectiveness of mixing depends on the state of aggregation of mixed phases, temperature, viscosity and density of liquids, mutual solubility of mixed fluids, type of stirrer, a what is the most critical - shape of the mixer. In the present research, the objective is to analyse the process of the fluid flow in the reactor with a mechanically agitated innovative vessel. Velocity field values were determined using computer simulation and experimental particle image velocimetry method. The basis for the assessment of the intensity degree and efficiency of mixing was the analysis of velocity vectors distribution. Experimental and numerical analysis was carried out for various stirred process parameters in order to determine optimal conditions for the mixing process.

3:05pm - 3:20pm

Flow in a cylindrical cavity with a periodically rotating lid

Joshua David Selvaraj, Jordi Pallares Curto, Anton Vernet Pena, Francese Xavier Grau Vidal
Universitat Rovirai Virgili, Spain

In this work, we study the flow behaviour of the fluid in a cylindrical cavity with a rotating lid having the sinusoidal angular velocity at Reynolds number Re = 2800. The aspect ratio (height/radius) of the cylindrical cavity is 2. The sinusoidal rotation has a 20% variation on the lid's time-averaged angular velocity. Particle Image Velocimetry (PIV) and numerical simulations (finite volume) are used to analyse the flow behaviour. We found that the sinusoidal rotation controls the flow periodicity, causing the flow to have higher intensity of the velocity fluctuations (up to 30%) and increases the number of different wave-numbers of the fluctuations of the angular velocity component along the azimuthal direction in comparison with the constant rotation of the lid.

3:20pm - 3:35pm

Field synergy principle for natural convective rotating fluid flow past a vertical cylinder

H.P Rani[1], Koragoni Naresh[1], Y. Rameshwar[2]
[1]National Institute of Technology, Warangal, India; [2]University college of Engineering, Osmania University, Hyderabad, India

In this paper, the thermal and induced magnetic field effects on the natural convective rotating fluid flow past the vertical cylinder are presented. The numerical solution of the field variables, such as velocity, induced magnetic field, and temperature profiles are obtained by solving the non-dimensional governing non-linear unsteady equations. The effects of the Prandtl, Taylor, and

Chandrasekar numbers on the induced magnetic field, skin-friction and Nusselt number are presented graphically. The concept of field synergy (coordination) principle is discussed to understand the enhancement of convective heat transfer. It is observed that synergy increases with the rate of heat transfer.

3:35pm - 3:50pm

Numerical simulation of optical clearing effect on the light propagation of skin tissue

Bin Chen, Jun Ma, Yue Zhang, Dong Li, Xu Sang

Xi'an Jiaotong University, China, People's Republic of

Based on the selective photothermolysis, laser therapy has become the most effective strategy to thermally damage the proliferative vessels in port wine stain. However, the light propagation is limited due to the high scattering characteristics of skin, and optical clearing agent (e.g. glycerol) can significantly increase the optical transparency of skin tissues. In this work, the effect of glycerol on energy deposition was calculated by Monte Carlo method to investigate light propagation and distribution in skin tissue. The result showed that the energy deposition gradually decreases in epidermis and increases in blood vessels as increasing glycerol content. When 50% water in skin is replaced by glycerol, the energy deposition in the center of the vessel can be increased by 83%. It can improve the photon energy deposition and thermal damage of the blood vessels, which provides the theoretical guidance in the clinical treatment.

3:50pm - 4:05pm

Numerical model of biological tissue heating using the models of bio-heat transfer with delays

Ewa Majchrzak[1], Bohdan Mochnacki[2]

[1]Silesian University of Technology, Poland; [2]University of Occupational Safety Management, Katowice, Poland

The mathematical description of thermal processes in the biological tissue domain is usually based on the well known Pennes equation. However, taking into account the specific inner structure of the tissue, it seems that the better approximation of the real heat transfer processes in this domain can be obtained using the hyperbolic Cattaneo-Vernotte equation (CVE). According to experiments described in literature the relaxation time appearing in the equation discussed is of the order of a few to several seconds. In the papers devoted to Cattaneo-Vernotte equation, the first order equation is considered. In this paper the second order CVE is proposed. In the first part of the paper the comparison between numerical FDM solutions basing on the first and the second order CVE is presented. The second part is devoted to the modeling of the multi-layered skin tissue heating, wherein between the layers the ideal contact conditions are assumed.

4:05pm - 4:20pm

Potential Problems on Surfaces: with or without Boundary, Subjected to Discrete Distributed Sources

Ofer Eyal, Ayelet Goldstein

Ort Braude College, Israel

we derive analytical solutions for potential problems on different types of surfaces, such as cylinders, cones and spheres.

These solutions complement the solution obtained for a sphere. Besides examining modes of truncation for a sphere, we also provide a complete coverage of all modes of truncation for cylinders and cones.

Yet since every physical set-up takes place in the R^3 space, some adjustments were required. We focused on three types of geometric surfaces: cylinder, cone and sphere. As being effectively two-dimensional problems, the use of analytic functions and conformal maps has been used to obtain analytical solutions for all these geometries, all are mapped into the complex plane. Special attention is paid to examine particular cases of truncated ends by taking into account boundary conditions and flux considerations. We used reflections on cylinder inversions on cone and explore a spherical version of inversion on a sphere in order to satisfy the boundary conditions.

SE-13: Day 2 Session 13

Time: Wednesday, 04/Sep/2019: 2:20pm - 4:20pm · *Location:* Uniroma Tre
Session Chair: Magdalena Piasecka

Room 21

2:20pm - 2:35pm

Mixed Convective Heat Transfer From A Porous Cylinder In Steady Flow

Shimin Yu[1,2], **Peng Yu**[1]

[1]Department of Mechanics and Aerospace Engineering, Southern University of Science and Technology, Shen Zhen, China; [2]Harbin Institute of Technology, Harbin, China

A numerical study is performed to comprehend the influence of buoyancy on the steady thermal flow past and through a porous cylinder. The finite volume method based on the collocated body-fitted and multi-block grids is applied. Considering the Boussinesq approximation, Navier-Stokes equations are used to govern the clear fluid region, and the Darcy-Brinkman-Forchheimer extended model is applied for the porous region. Flow behaviour and heat transfer are investigated in terms of streamlines, isotherms, and the Nusselt number. Wake structure and thermal characteristics are discussed and compared between forced convection and mixed convection by considering effects of the Reynolds number, Darcy number, and Richardson number. Thermal buoyancy is found to significantly influence the flow pattern and heat transfer rate. Results also show that the role of buoyancy for the suppression of the recirculating wake and the enhancement of heat transfer strongly depends on the interaction between buoyancy and the free stream.

2:35pm - 2:50pm

Numerical investigation of heat loss through cascaded cavity receiver at high temperatures up to 500 C

Kushal S Wasankar[1], **Shreyas C Yadav**[1], **Ramola Sinha**[1,2], **Nitin P Gulhane**[1]

[1]Veermata Jijabai Technological Institute, Mumbai, India; [2]K J Somaiya College of Engineering, Mumbai, India

In solar thermal systems, especially for high concentration applications, natural convection and radiation contributes a significant fraction of energy loss. Its characteristics hence need to be understood to improve system efficiency. In this paper a numerical study carried out to investigate the heat loss through a cascaded cavity receiver of a solar dish collector. The effect of increase in area ratio on heat loss is studied. The cascaded cavity receiver model is electrically heated with constant heat flux. A simulation model for combined natural and surface radiation is developed. The influence of orientation of the receiver, and the geometry on total heat loss from the receiver is investigated. The cavity inclination is varied from 0 to 90 in steps of 30. The Computational Fluid Dynamics package “ANSYS 19.2 Fluent” has been used to numerically investigate the influence of cavity geometry and inclination on the convective loss through the aperture.

2:50pm - 3:05pm

Computational study of a laminar natural convection inside adiabatic channel with ribs

Khalil Farhan Yassin[1], **Ali Lafta Ekaid**[2], **Viktor Ivanovich Terekhov**[3]

[1]Al-Hawija Technical Institute, Northern Technical University, Iraq; [2]Mechanical Engineering Dept., University of Technology, Baghdad, Iraq; [3]Kutateladze Institute of Thermophysics, SB RAS, Novosibirsk, Russia

In this work, a numerical study of laminar free convection and heat transfer in a vertical plane channel with & without two thin adiabatic ribs on its walls are presented. The inlet and outlet of the channel are opened, and the temperature was maintained as the same at its surfaces. The channel height varies A = L/w = 10 to 500, and the ribs were located in the middle of the channel towards each other. Rib length l/w = 0.4, and Rayleigh number values Ra = 100 ÷ 100000 were varied in calculations. The numerical solution is conducted for Prandtl number, Pr=0.7. The effect of these parameters on the flow structure, temperature field, local and integral heat transfer, and gas flow caused by gravitational forces was analyzed in detail. Numerical analysis was based on the solution of the full Navier - Stokes, and energy equations in two-dimensions.

Good agreements with the published papers were noticed.

3:05pm - 3:20pm

Numerical simulation of unsteady channel flow with a moving indentation

Chandrakant Rameshchandra Sonawane[1], **Yogesh B More**[1], **J C Mandal**[2], **AnandKumar Pandey**[1]

[1]Symbiosis International University, India, India; [2]Indian Institute of Technology Bombay, Mumbai

The FSI problem - unsteady channel flow with a moving indentation problem, which represents flow features of oscillating stenosis of a blood vessel, is numerically simulated. The flow inside the channel with moving boundary results in transient and complex flow phenomena mainly due to the interaction between the moving boundary and the flowing fluid. In this paper, an accurate Harten Lax and van Leer with contact for artificial compressibility Riemann solver have been used for flow computation. The Riemann solver is modified to incorporate arbitrarily Lagrangian-Eulerian (ALE) formulation in order to take care of mesh movement in the computation, where radial basis function is used for dynamically moving the mesh. Higher order accuracy over unstructured meshes is achieved using quadratic solution reconstruction based on solution dependent weighted least squares (SDWLS). The present numerical scheme is validated here and the numerical results are found to agree with experimental results reported in literature.

3:20pm – 3:35pm

A temporally piecewise SBFEM adaptive method to solve hyperbolic heat problems with radiative boundary conditions

Bicheng Liu, Yiqian He, Haitian Yang, Chunjiang Ran

Dalian University of Technology, China, People's Republic of

Integrating advantages of SBFEM (scaled boundary finite element method) and a temporally piecewise adaptive method, a novel numerical algorithm is presented to solve hyperbolic heat conduction problems with radiative boundary conditions. An adaptive piecewise recursive computation is addressed by expanding all variables in a discretized time interval, resulting in no iteration requirement for nonlinear problems, and a stable computing accuracy for different step sizes. Numerical examples are provided to verify the proposed approach with the consideration of affects of step sizes and element sizes, and a good accordance with reference solutions can be observed.

3:35pm – 3:50pm

Relaminarization of a hot air impingement on a flat plate

Mongkol Kaewbumrung[1], Chalermpol Plengsa-ard[2]

[1]Department of Mechanical Engineering, Rajamangala University of Technology Suvanrnabhumi, Huntra Campus Phra Nakhon Sri Ayutthaya, Thailand; [2]Department of Mechanical Engineering Faculty of Engineering, Kasetsart University, Bangkok Thailand

The mechanisms of relaminarization from air impingement on a flat plate are investigated. The simulation is presented by using the commercial CFD code FLUENT and the Reynolds Averaged Navier Stokes (RANS) approach is employed in order to predict the flow fields. In particular, the hot gas jet impingement on the flat plate with a constant surface heat fluxes is chosen to be the test case. The jet Reynolds number is equal to 23,000 and a fixed jet-to-plate spacing of H/D = 2.0. Three two-equation turbulence models are tested and compared the results. There are the standard k-omega, k-omega shear stress transport (SST), baseline k-omega (BSL) turbulence models. The instantaneous computed Nusselt number agree fairly well with experimental data. The highest values of simulated Nusselt number are located near the stagnation point and decrease monotonically in the wall jet region. Also, the simulations can capture the relaminarization mechanisms within the boundary layer near walls.

3:50pm – 4:05pm

Utilization of hydraulic energy in a fish farm

Sang-Ho Suh[1], Md Rakibuzzaman[1], Young Tae Ryu[2], Kyung Yup Kim[3]

[1]Department of Mechanical Engineering, Soongsil University, Seoul, Korea; [2]1113, Jugok-ri, Ujeong-eup, Hwaseong-si, Gyeonggi-Do, Korea; [3]Department of Mechanical Engineering, Korea Polytechnic University, Gyeonggi-Do, Korea

In this study, discharge water of fish farm has used to obtain renewable energy using a marine micro-scale hydropower plant. Because it is clean, renewable and abundant energy sources. In addition, micro hydropower has the advantage of installation onshore fish farms, wastewater treatment plants where suitable water is obtained throughout the year. But in the case of micro-scale hydropower systems; lack of technical skills, economic feasibility, the national granting policy expansion, initial investment, the expansion of related industries, and the turbine efficiency are still challenging to improve the operational management of the micro-scale hydro turbine. However, if the design technique is improved and secure economic sustainability, the cost of management could be reliable in the fish farms. The main objective of this study is to investigate the possibility of using renewable energy in the 100kW class tubular type turbines in a fish farm with the permanent magnet synchronous generator (PMSG).

4:05pm – 4:20pm

Studying the impact of an inhomogeneous participating medium in a coupled convection-radiation system using finite element based methods

Jorge Avalos-Patino, Stephen Neethling, Steven Dargaville, Matthew Piggott

Imperial College London, United Kingdom

Combined convection-radiation is a common phenomenon in many engineering problems. A differentially-heated rectangular enclosure is a widely-used benchmark for testing numerical techniques developed for solving the coupled momentum and energy equations related to combined convection-radiation. Previous studies have tended to describe the phenomenon in cases of where the radiative properties inside the cavity are homogeneous. However, the effects of inhomogeneous participating medium properties are arguably under-studied. In this work the effect of an inhomogeneous participating medium on combined convection-radiation inside a rectangular enclosure is considered. The momentum and energy equations are solved numerically using finite element based iscretization methods. The radiative transfer equation is solved numerically using both spherical harmonic and discrete ordinate expansions, with their relative performance compared. The results highlight the impact of a homogeneous vs inhomogeneous participating medium on the velocity and temperature fields.

SE-14: Day 2 Session 14

Time: Wednesday, 04/Sep/2019: 2:20pm – 4:20pm · *Location:* Uniroma Tre
Session Chair: Piotr Łapka

Room 22

2:20pm - 2:35pm

A systematic performance comparison of finite-volume and spectral-hp methods for LES of a representative combustor geometry

Vishal Saini, Hao Xia, Gary J. Page

Loughborough University, United Kingdom

High-order accurate methods are beneficial for well-resolved and geometrically simple scale-resolving fluid simulations. However, the benefit on coarse unstructured grids for relatively complex, industrially relevant geometries is less clear. We present a systematic evaluation of accuracy versus computational cost for Large-Eddy Simulations (LES) of simple to complex geometry cases using a standard second-order solver and a high-order spectral-hp solver from OpenFoam and Nektar++ frameworks respectively. The Taylor-Green vortex LES using coarse resolutions shows that fourth-polynomial-order simulations run 3 to 10 times cheaper for a given accuracy. Further, the comparison is undertaken on a previously (experimentally) studied combustor representative case. The flow consists of six radial jets impinging on main cross-flow, which closely resembles dilution port flows of a rich-burn gas turbine combustor. Fourth-polynomial-order LES is found to resolve a broader range of flow scales for given computational time. Moreover, for similar accuracy, the high-order simulations potentially cost 2 times cheaper.

2:35pm - 2:50pm

Prediction of raceway shape in zinc smelting furnace under the different air inflating conditions

Robert Straka, Mikołaj Bernasowski, Arkadiusz Klimczyk, Ryszard Stachura, Dmytro Svyetlichnyy

AGH Univerisy of Science and Technology, Poland

Shape of raceway in the Imperial Smelting Process plays an important role in zinc production, especially for its effectiveness. A model considers the process in smelting furnace as two oppositely directed flows. Intensive interactions between the flows take place in the raceway zone, in which the combustion and gasification of the coke is simulated. The shape of the raceway zone is the result of such interactions. The purpose of the paper is simulation of combustion in the raceway zone in view of its shape, analysis of the simulation results and shape prediction for different conditions. Simulations for the blast volumetric flow ranging from 30000 Nm3/h to 35000 Nm3/h with different oxygen enrichment ranging from 400 Nm3/h to 800 Nm3/h were performed. The results showed that for these setting the change in the shape and depth of the cavities are not significant but anyway they are noticeable.

2:50pm - 3:05pm

Numerical investigation of the impact of variable particle radiation properties on the heat transfer in a high ash pulverized coal boiler through co-simulation

Ryno Laubscher[1], Pieter Gerhardus Rousseau[2]

[1]University of Cape Town, South Africa; [2]University of Cape Town, South Africa

Co-simulation of the high-temperature heat and mass transfer processes in coal-fired boilers using CFD together with process-level models of the water/steam circuit is a promising approach to investigate off-design conditions. For the present work, a 1D discretized two-phase model of the water flows in the evaporator and radiative superheaters were developed using Flownex SE 8.9. This was then coupled with a detailed furnace combustion and heat transfer CFD model in Fluent 19.2. The coupled models are used to investigate the impact of variable particle emissivity and scattering efficiency on the evaporator and radiant superheater process conditions. The results are compared with the case where the radiation properties of the particles are assumed to be constant, thereby highlighting the importance of accounting for the variations in the properties throughout the process. The case study boiler is that of a real 620 MWe subcritical power plant firing coal with a high ash content (41 %wt).

3:05pm - 3:20pm

Investigation into the mechanisms of deflagration-to-detonation transitions using direct numerical simulations

Dr Weiming Liu, Dr Jonathan Francis, Dr Akinola Adeniyi, Joseph Owede Adoghe

University of Central Lancashire, Preston, United Kingdom, United Kingdom

Detonation, a supersonic combustion wave plays a critical role in the theory and application of combustion. This work presents numerical investigation into indirect initiation of detonation using direct numerical simulations (DNS). The Adaptive Mesh Refinement in object-oriented C++ (AMROC) tool for parallel computations is applied. The combustion reactions occur in a shock tube and controlled by chemical kinetics. The DNS database produced simulates the deflagration to detonation transition (DDT), and investigates the influence of overpressure and chemical kinetics on flame propagations. The numerical simulations showed the influence of pressure and kinetics on the transition of slow flame, fast flame and DDT during flame propagations. When the reaction rate is fast, DDT is achieved, but when slow, DDT will not occur and no detonation and consequently no strong explosion. The influence of free radical H on flame propagation showed that as the reacting species concentration decreases, the flame speed increases.

3:20pm - 3:35pm

Influence of the expression used to account for the heat conduction flux when modeling the laser-induced incandescence of soot aggregates produced in a turbulent flame of Diesel

Sebastien Menanteau[1], **Romain Lemaire**[2]

[1]ICAM, Lille, France; [2]TFT laboratory, Department of Mechanical Engineering, École de Technologie Supérieure, Montréal, Canada

Laser-induced incandescence (LII) is a powerful diagnostic technique allowing quantifying soot emissions and understanding their formation in flame conditions. It can be advantageously coupled with modeling approaches to infer information on soot physical properties which needs solving the balance equations accouting for laser-excited particle heating and cooling processes. To estimate soot size by time-resolved LII, it is necessary to correctly estimate the thermal accommodation coefficient driving the energy transferred by heat conduction between soot aggregates and their surroundings. Within this scope, we used a comprehensive experimental database obtained in a turbulent flame of Diesel and applied a refined LII model integrating correction factors to thoroughly account for the particle aggregate properties with the view to assess the thermal accommodation coefficient. We then investigated the influence of the aggregate size and implemented two heat conduction equation formulations commonly used in LII models to study their effects on the numerically obtained results.

3:35pm - 3:50pm

Analysis of combustion, heat and fluid flow in a biomass furnace

Björn Pfeiffelmann[1], **Michael Diederich**[1], **Fethi Gül**[1], **Ali Cemal Benim**[1], **Markus Heese**[2], **Andreas Hamberger**[2]

[1]Duesseldorf University of Applied Sciences, Germany; [2]Endress Holzfeuerungsanlagen, Burgbernheim, Germany

Biomass is increasingly playing a role as a renewable energy source. For biomass furnaces that are used for heat generation, the system performance can be increased by co-generation of electric power. We initiated a research project, which aims to use Thermoelectric Generators to this purpose. For being able to design the system efficiently, especially, for the configuration of the TEGs, the temperature distribution in the furnace as well as in the following convective heat exchanger needs to be known with a good precision. To this purpose, the prediction methods must be able to deliver sufficiently accurate and reliable results. For ensuring this capability, a careful validation of the applied computational procedures is necessary. This is the scope of the present work. The combustion, heat and flow processes in a biomass furnace is measured and calculated. The computational methods are validated by comparisons with the measured values.

3:50pm - 4:05pm

Numerical analysis of the combustion of straw and wood in a stoker boiler with vibrating grate

Rafal Kobylecki[1], **Robert Zarzycki**[1], **Mateusz Winski**[2], **Zbigniew Bis**[1]

[1]Czestochowa University of Technology, Poland; [2]Energa Kogeneracja, Poland

The hydrodynamics of a biomass-fired 90 MW stoker boiler with vibrating grate was numerically investigated in order to improve the air distribution and boiler performance, as well as to decrease the fouling of the furnace membrane walls. The numerical calculations were supported by experimental video data from a laboratory model of the boiler.

The results indicated that by the optimization of the grate design and the distribution of secondary air significant improvement of obiler performance and reduction of the fouling rate may be achieved.

4:05pm - 4:20pm

Entropy generation in an unsteady reactive viscous flow in a porous cylindrical pipe with an isothermal wall

Philip Iyiola Farayola

Emmanuel Alayande College of Education, Oyo. Oyo State.Nigeria

Studies of entropy in heat generating systems are of great interest to researchers in engineering. This paper studied the entropy generation in an unsteady reactive viscous flow in a porous cylindrical pipe with an isothermal wall under Arrhenius kinetics. The nonlinear equations of momentum and energy governing the flow system were solved using perturbation technique to get an approximate solution of the resulting dimensionless nonlinear equations. The effects of Prandtl number, permeability and heating parameters on the entropy of the system were studied. It was found that increase in Prandtl number and viscous heating parameter cause increase in entropy. Maximum entropy is observed to be between $r = 0.8$ and $r = 0.9$ away from the centre of the cylindrical pipe and as the permeability parameter increases, the entropy increases until when permeability parameter approaches 4.2887, a point at which entropy blow up.

PS-01: Day 2 Poster Session 1

Time: Wednesday, 04/Sep/2019: 4:40pm - 6:00pm · *Location:* Uniroma Tre

Effects of the thermal boundary conditions on the onset of an oscillatory regime of heat transfer from a horizontal heated cylinder inside a water-filled enclosure

Marta Cianfrini[1], Corcione Massimo[2], Alessandro Quintino[2], Vincenzo Andrea Spena[2]

[1]DIMI - Università degli Studi RomaTre, Rome, Italy; [2]Università di Roma "La Sapienza", Italy

Natural convection from a horizontal heated circular cylinder suspended inside a water-filled square enclosure, is studied numerically. Besides the cooling of the four confining walls of the enclosure, several other boundary conditions are applied by replacing one or more cooled walls with perfectly insulated walls.

Numerical simulations are carried out for different values of the Rayleigh number based on the cylinder diameter, as well as of the width of the enclosure and the distance of the cylinder from the bottom wall of the enclosure, both normalized by the cylinder diameter.

Main scope of the present study is to investigate the basic heat and momentum transfer features, to determine in what measure any different cooling condition imposed at the boundary walls of the enclosure affects the onset of an oscillatory regime of heat transfer, and to analyze the effects of any independent variable on the thermal performance of the cylinder.

Natural convection from an enclosed horizontal heated plate

Massimo Corcione, Luca Cretara, Alessandro Quintino, Vincenzo Andrea Spena

Università di Roma "La Sapienza", Italy

Natural convection from a heated horizontal plate inside a gas- or liquid-filled square enclosure cooled at sides, is studied numerically. The upper and lower sides of the plate are either simultaneously heated or individually heated, the other side being kept perfectly insulated.

Numerical simulations are executed for different values of the Rayleigh number based on the plate length and the Prandtl number, as well as of the width of the enclosure and the distance of the plate from the bottom wall of the enclosure, both normalized by the plate length.

Main scope of the present study is to analyze how much any independent variable affects the thermal performance of the plate, to determine in what measure the simultaneous heating of the other side affects the thermal performance of each side of the plate in comparison with the more standard one-sided heating configuration, and to develop heat transfer correlating equations.

Buoyancy-induced convection from a pair of staggered vertical heated plates in interacting flow fields

Massimo Corcione[1], Ivano Petracci[2], Alessandro Quintino[1], Vincenzo Andrea Spena[1]

[1]Università di Roma "La Sapienza", Italy; [2]Università di Roma "Tor Vergata"

Buoyancy-induced convection from a pair of staggered heated vertical plates suspended in free air, is studied numerically considering both the double-side and the single-side heating conditions.

The system of the conservation equations of the mass, momentum and energy expressed in dimensionless form is solved through a control-volume formulation of the finite-difference method.

Numerical simulations are carried out for different values of the Rayleigh number based on the plate length, as well as of the horizontal separation distance between the plates and their relative position in the direction of the gravity vector, both normalized by the plate length.

Main scope of the present study is to determine in what measure any independent variable affects the thermal performance of each plate, and to define the existence of an optimal plate separation distance for maximum heat transfer in relation with the Rayleigh number and the vertical reciprocal location of the plates.

Buoyancy-driven convection from a vertical heated plate suspended inside a nanofluid-filled cooled enclosure

Massimo Corcione, Emanuele Habib, Alessandro Quintino, Vincenzo Andrea Spena

Università di Roma "La Sapienza", Italy

Buoyancy-driven convection from a heated vertical plate suspended inside a nanofluid-filled square enclosure cooled at the walls, is studied numerically using a two-phase model based on the double-diffusive approach.

The computational code incorporates three empirical correlations for the evaluation of the effective thermal conductivity, the effective dynamic viscosity and the coefficient of thermophoretic diffusion.

Simulations are executed using water-based nanofluids with dispersed alumina, copper oxide or titania nanoparticles, for different values of the diameter and the average volume fraction of the suspended nanoparticles, the plate length and position, the width of the enclosure, the average temperature of the nanofluid, and the temperature difference imposed between the plate and the boundary walls.

Main scope of the present study is to determine in what measure any controlling parameter affects the thermal performance of the plate, and to define the existence of an optimal plate position and an optimal particle loading.

Effect of mutual radiative exchange between the surfaces of a street canyon on the building thermal energy demand

Andrea Vallati, Chiara Colucci

Università di Roma "La Sapienza", Italy

In this paper, a building energy simulation tool is exploited to study the impact of multiple radiative inter-reflections exchanges in an urban environment with the aim of evaluating their influence on the thermal energy demand of buildings. A street canyon model validated in a previous work is used in TRNSYS to investigate the effects of the related urban radiative trapping. Due to multiple shortwave and longwave reflections, the actual radiation exchanged by the buildings facades is different compared to a street canyon building where only shadowing phenomena due to canyon geometry are considered. Buildings energy simulation commercial codes do not take in account inter-reflections inside urban canyons. Simulations in TRNSYS are carried out for both models to investigate to what extent this phenomenon affects buildings energy demand by varying solar absorption. Increases in cooling demand up to 34% and decreases in heating demand up to 5% are found.

Experimental investigation about the adoption of high reflectance materials on the envelope cladding of scaled street canyon buildings for their energy demand reduction

Andrea Vallati[1], Gabriele Battista[2], Roberto de Lieto Vollaro[2], Chiara Colucci[1]

[1]Università di Roma "La Sapienza", Italy; [2]Università di Roma "Roma Tre", Italy

In this paper experimental measurements were performed on an apparatus made at the University of Rome. The study aims to investigate the influence of retro-reflective materials on urban canyons buildings envelope. The experimental apparatus consists of three plasterboard blocks with aspect ratio H/W = 1. Sensors capable of monitoring incident and/or reflected radiation (albedometers) have been placed at the canopy layer. Measurements are carried out for different types of paints applied on the facades of the three blocks (Lambertian and retro-reflective paints), for different values of solar absorption and for two different orientations of the canyons (North-South and East-West). Daily measurements were performed from 10.00 am to 5.00 pm during the summer season (June-September). The results show that, when retroreflective paints are applied, canyon albedo is higher than the standard Lambertian paint case, especially in the central hours of the day (2%).

Heat transfer in fluidized and fixed beds of adsorption chillers

Jaroslaw Krzywanski[1], Karolina Grabowska[1], Marcin Sosnowski[1], Anna Zylka[1], Tomasz Czakiert[2], Karol Sztekler[3], Marta Wesolowska[3], Wojciech Nowak[3]

[1]Jan Dlugosz University in Czestochowa; Faculty of Mathematics and Natural Sciences, Armii Krajowej 13/15, 42-200 Czestochowa, Poland; [2]Czestochowa University of Technology, Institute of Advanced Energy Technologies, Dabrowskiego 73, 42-200 Czestochowa, Poland; [3]AGH University of Science and Technology, Poland

One of the main disadvantages of conventional fixed-bed adsorption chillers is the low coefficient of performance due to high voidage of the sorbent beds.

The common method, which can help to handle this problem, is modifying the beds' design, including the application of fluidization.

An innovative idea, shown in the paper constitutes in the use of the fluidized bed of sorbent, instead of the conventional, fixed-bed, commonly used nowadays in the adsorption chillers. Bed-to-wall heat transfer coefficients for fixed and fluidized beds of adsorbent are determined. Sorbent particles diameters and velocities of fluidizing gas are discussed in the study. The calculations confirmed, that the bed-to-wall heat transfer coefficient in the fluidized bed of adsorbent is much higher than that in a conventional bed.

The proposed idea allows significantly reducing the size of a chiller and overcome the main reason for limited dissemination of the adsorption cooling technology.

Numerical Analysis on Confined Round Impinging Slot Jets with Nanofluids in Aluminum Foams

Bernardo Buonomo[1], Anna di Pasqua[1], Oronzio Manca[1], Ghofrane Sekrani[2], Sebastien Poncet[2]

[1]Università degli Studi della Campania "Luigi Vanvitelli", Italy; [2]Université de Sherbrooke

In this paper a numerical study on mixed convection in confined round jets impinging on a porous media is accomplished. The working fluids are pure water or Al_2O_3/water based nanofluids and a single-phase model approach has been applied to describe their behavior. A two-dimensional configuration is analyzed and different Peclet and Rayleigh numbers are considered. The thermal non-equilibrium energy condition is assumed to accomplish two-dimensional simulations on the system. The examined aluminum foams are characterized by several porosities for distinct PPI values. The alumina volume concentrations range from 0% to 4% and the particle diameter is 30 nm. The distance of the target surface is five times greater than the round jet width. The results show the increase of the convective heat transfer coefficient for increasing values of Peclet number and nanoparticle concentration. The heat transfer coefficient shows different behaviors at varying porosity for different Peclet numbers.

Integration adsorption chillers with combined cycle gas turbine

Karol Sztekler[1], Wojciech Kalawa[1], Łukasz Mika[1], Jaroslaw Krzywanski[2], Karolina Grabowska[2], Marcin Sosnowski[2], Tadeusz Wójcik[1], Wojciech Nowak[1], Adam Bieniek[1], Tomasz Siwek[1]

[1]AGH University of Science and Technology, Poland; [2]Czestochowa University of Technology, Institute of Advanced Energy Technologies, Dabrowskiego 73, 42-200 Czestochowa, Poland

Long-term forecasts indicate that the annual increases in electricity demand by 2030 will be approx. 2 ÷ 3% a year. With such a high rate of development of the world economy, the electricity demand will be increasing. More efficient use of the primary energy contained in fuels translates into tangible earnings for power plants while reductions in the amounts of fuel burned, and of non-renewable resources in particular, certainly have a favourable impact on the natural environment. The main aim of the paper is investigate the contribution of the use of adsorption chillers to improve the production energy efficiency in combined cycle gas turbine. As part of their project, the authors have modelled the combined cycle gas turbine integrated with an in adsorption chillers. Simulation calculations were performed using Sim tech's IPSEPro software.

Comparative study for the prediction of cavitating flow inside a square-edged orifice using different commercial cfd software

Gong-hee Lee[1,2], June-ho Bae[1]

[1]Korea Institute of Nuclear Safety, Korea, Republic of (South Korea); [2]University of Science and Technology, Korea, Republic of (South Korea)

Nuclear power plant operators conduct in-service testing (IST) to verify the safety functions of safety-related pumps and valves and to monitor the degree of vulnerability over time during reactor operation. The system to which the pump and valve to be tested are installed has various sizes of orifices for flow control and decompression. Rapid flow acceleration and accompanying pressure drop may cause cavitation inside the orifice, which may result in orifice degradation and structural damage. Though licensing applications supported by using Computational Fluid Dynamics (CFD) software are increasing for IST-related problems, there is no CFD software which obtains a licensing from the domestic regulatory body until now. In this paper, to assess the prediction performance of different commercial CFD software for the analysis of cavitating flow inside a square-edged orifice, the simulation was conducted with ANSYS CFX and FLUENT. The results predicted were then compared with the measured data.

Inclined elliptical footprint drop impact on a solid surface

Sungchan Yun

Korea National University of Transportation, Republic of Korea

Drop impact on a solid surface has received extensive attention in many industrial fields such as inkjet printing, surface cooling and cleaning [1]. We study the influence of the inclined angle of the elliptical drop on the impact dynamics during the drop impact on nonwetting surfaces. The impact angle plays a vital role in changing impact dynamics and suppressing bounce magnitude. Experiment and numerical studies reveal that the high angles only slightly affect the reduction in the maximum bounce height by inducing more extended liquid column than the small angles.

Non-spherical drop impact on a solid surface for reducing the bounce magnitude

Sungchan Yun

Korea National University of Transportation, Republic of Korea

This study suggests an asymmetrically cropped drop impact on a solid surface, while the conventional drop dynamics is limited to spherical drops. Similar to the typical characteristics of bouncing elliptical drop [1,2], it was observed that drop alignment phenomenon occurs in which drops are elongated due to the main flow in one direction in the case of an non-spherical drop. Asymmetric spreading and retracting behavior was observed after collision. The maximum recoil height reductions of about 50% on the hydrophobic surfaces and the superhydrophobic surfaces were observed compared to the spherical drop.

Trefftz method of solving a problem of thermoelasticity for a layered thin plate

Artur Maciag, Kszysztof Grysa

Kielce University of Technology, Poland

One-dimensional thermoelastic mathematical model for heat conduction problem is constructed for a layered thin plate. The basic equations in dimensionless form are presented in such a form that in the displacement equation the temperature only appears on the right in the heterogeneity describing member; similarly in the heat transfer equation, displacements occur on the right side in the heterogeneity describing section. Approximate general solutions of the homogeneous basic equations are assumed in the form of linear combination of Trefftz functions. Then the inverse operators to the operators appearing in homogeneous equations were defined. With the use of inverse operators, particular solutions of non-homogeneous basic equations were obtained. Minimizing the objective functional leads to an approximate solution of considered problem. The solution was applied to a plate of sandwich structure, which is heated, and traction free in the outer sides.

Numerical study for flow distribution inside a fuel assembly with twist-split type mixing vane using ansys cfx

Gong-hee Lee[1,2]

[1]Korea Institute of Nuclear Safety, Korea, Republic of (South Korea); [2]University of Science and Technology, Korea, Republic of (South Korea)

In a pressurized water reactor (PWR), the appropriate cooling of fuel rod bundle is important for thermal margins and safety. Because a spacer grid with mixing vanes may cause rigorous mixing as well as greatly increased local turbulence levels inside the sub-channel, prediction of sub-channel flows, even in isothermal condition, is very difficult. The advantage of a CFD software for subchannel flow predictions is that it does not rely to the same extent on these empiricisms. In this study, in order to examine the flow distribution inside a fuel assembly with twist-split type mixing vanes as a part of benchmark simulation for a Coordinated Research Project (CRP) on the application of CFD codes to Nuclear Power Plant (NPP) design, simulations were conducted with the commercial CFD software, ANSYS CFX. The predicted results were compared with the measured data from the Omni Flow Experimental Loop (OFEL) test facility.

Heat transfer Characteristics between two horizontal pipelines in a heat tracing system

Chi-Ming Lai[1], G.N. Sou[2], Rong-Horng Chen[3], C.J. Ho[2]

[1]Department of Civil Engineering, National Cheng-Kung University, Taiwan; [2]Department of Mechanical Engineering, National Cheng-Kung University, Taiwan; [3]Department of Mechanical and Energy Engineering, National Chiayi University, Taiwan

In this study, a numerical simulation of natural convection between two horizontal pipes with a heat tracing enclosure is performed using the finite difference method. The heat tracing enclosure consists of differentially heated horizontal cylinders inside an adiabatic circular and air-filled enclosure. The cold cylinder is above the hot cylinder, and both are parallel to the ground. The relevant dimensionless parameters and ranges are as follows: an inclination angle of the enclosure of -90°, a center-to-center spacing between cylinders = 0.7, and Rayleigh numbers of 1—10⁴, 1—10⁵, and 5—10⁵; an inclination angle of the enclosure = -90°, a center-to-center spacing between cylinders = 0.833, and Rayleigh numbers of 1—10⁴ and 1—10⁵. The results show that transient irregular fluctuations of the flow field and heat transfer occur in advance when the Ra number increases or the distance between the cylinders decreases.

Esperimental and numerical investigation of free convection in a rectangular tank

Dorota Sawicka[1,2], Albert Baars[1], Janusz Tadeusz Cieśliński[2], Sławomir Smoleń[1], Florian Hoffmann[1]

[1]City University of Applied Sciences Bremen; [2]Gdańsk University of Technology, Poland

Results of an experimental and numerical study of the liquid level influence on free convective heat transfer and flow topology are presented in this study. The test setup consists of a cubical container and a horizontally positioned stainless steel tube, which is used as a resistance heater. The container is filled with fluid to a level of 210mm, 160mm and 110mm from the heating section. For the Particle Image Velocimetry (PIV) measurements polyamide seeding particles were added to the fluid and illuminated by a laser sheet placed vertically in the centre of the container. Investigated fluids are ethylene glycol, water and a solution of ethylene glycol-water 50%/50% by volume. For CFD calculations geometry data are taken from experimental setup to generate a 2D model. Continuity, momentum equation with Boussinesq approximation and energy equation for constant fluid properties are solved by the finite volume open source code OpenFOAM 4.1.

Application of the Trefftz method for pool boiling heat transfer on open microchannel surfaces

Sylwia Hożejowska, Robert Kaniowski, Robert Pastuszko

Kielce University of Technology, Poland

The results of investigations for pool boiling heat transfer on open microchannel surfaces were discussed in the paper. Nucleate pool boiling from copper microchannel surfaces was examined using ethanol as a working fluid. Parallel microchannels fabricated by machining were about 0.3 mm wide, and 0.2 to 0.5 mm deep and spaced every 0.6 mm.

In the mathematical model the heat transfer process was in steady state. The heat transfer in copper block was assumed axisymmetric while the specimen temperature distribution was not. The temperature of the copper block and the specimen were assumed to satisfy Laplace's equations which adequate boundary conditions. The problem was solved by the Trefftz method with two sets of Trefftz functions. Graphs were used to represent the copper block and the specimen temperature distributions, local values of heat transfer coefficients as a function of the fin length, fin temperature distribution and efficiency.

Transient heat transfer of Al2O3-water nanofluid flow in a microchannel

Wei-Mon Yan

National Taipei University of Technology, Taiwan

Transient cooling characteristics of Al2O3-water nanofluid flow in a microchannel subject to sudden-pulsed heat flux are numerically examined in details. The attention is focused on the effects of microencapsulated phase change material inside top wall of a microchannel on the transient cooling heat transfer of the microchannel. The numerical results are compared with the available works in the literature and found in good agreement. Besides, the predictions show that the presence of the MEPCM layer cannot effectively control the rise of the temperature of the microchannel in the presence of a pulse heat flux. For a typical pulse heat flux, the bulk temperature of the channel can be raised up to 2◦C while the presence of the MEPCM layer would only suppress the temperature rise by 0.2◦C. It is also found that the temperature of the top wall is under the significant influence of the MEPCM layer.

Internal flow field and heat transfer investigation inside a working chamber of the scroll compressor.

Józef Rak, Sławomir Pietrowicz

Wrocław University of Technology, Poland

Scroll compressors are positive displacement machines based on a single shaft motion similarly to multi-vane, rolling piston or Wankel technologies. In recent years a number of studies have been carried on over the topic of scroll compressors modelling. This paper focus on specifying parameters that affect a thermal balance inside a working chamber of the scroll compressor. The insight into the flow field was gained through CFD calculations by using numerical grid deformation. A set of parameters in the model: scroll vanes shape, discharge pressure and rotational speed were varied to test their impact on the flow and therefore heat transfer conditions in the working chamber. The results served to formulate a lumped parameter model of a generic positive displacement machine working chamber. The model takes into account nondimensional parameters, common for the whole family of volumetric machines and corresponding to identified parameters.

Tests of changes in the resistance of heating water flow in the home heating system under the influence of various configurations of heating boiler work

Bartosz Ciupek, Wojciech Judt, Rafał Urbaniak

Chair of Thermal Engineering, Poznan University of Technology, Poland

The article presents the results of experimental research on the change of resistance of heating water flow in the home heating system under the influence of boiler work configuration changes. For the tests was used the research object in the form of a solid fuel heating boiler with automatic fuel feeding. The research facility was installed in an open-type heating network designed to reproduce the layout of the house heating system. During the tests, boiler operation was simulated for a few selected thermal loads most often found in individual heating. For each of the selected options, a series of flow resistance measurements were made using a U-type liquid manometer tube made by researchers. The obtained results of experimental research can be used as an aid in the case of designing home heating networks equipped with a solid fuel boiler.

On the thermal characterization of building walls: an overview based on experimental studies

Luca Evangelisti[1], Claudia Guattari[1], Francesco Asdrubali[1], Roberto de Lieto Vollaro[1], Gabriele Battista[1], Andrea Vallati[2]

[1]Roma TRE University, Department of Engineering, Italy; [2]Sapienza University of Rome, DIAEE, Italy

It is well known that the performance of buildings structural elements highly influences annual energy demands. The thermal characterization of a wall can be achieved through its stratigraphy and the thermo-physical parameters of each material. When existing buildings are investigated, technical specifications may be unknown and heat transfer phenomena between walls and environment can be influenced by air-conditioning systems and local thermo-fluid dynamic conditions. In these cases, on-site experimental surveys become fundamental. On the other hand, Standards related to the building thermal behavior can help engineers or technicians to define some unknown information related to heat transfer coefficients and thermo-physical properties. Nevertheless, can Standards' suggestions be considered reliable in every situation? This paper tries to answer this question, debating some experimental investigations conducted in the last years.

Cauchy type inverse problem in a two-layer area in the blades of gas turbine

Michał Ciałkowski[1], Aleksander Olejnik[2], Andrzej Frąckowiak[1], Natalia Lewandowska[1]

[1]Poznan University of Technology, Poland; [2]Military University of Technology in Warsaw, Poland

In order to reduce the thermal load of the gas turbine blade, its surface is covered with an external ceramic layer of high thermal resistance. The main problem considered in our research is the selection of ceramics with a low value of specific thermal conductivity and its thickness. Obtained values should be chosen in such a way that the permissible metal temperature is not exceeded at the metal-ceramic boundary. It would result in the loss of mechanical properties. Therefore, for a given temperature run at the metal-ceramic boundary, the temperature on the inside of the blade and the initial temperature should determine the temperature function on the external surface of the ceramic. This issue is a Cauchy type issue. The paper considers the case of one-dimensional, non-stationary flow.

The effect of microstructure on self-propelled droplet jumping

Zhiping Yuan, Sihang Gao, Zhifeng Hu, Xiaomin Wu

Department of Energy and Power Engineering, Tsinghua University, China

Most of the studies on droplet jumping mainly focus the droplet jumping on almost flat surfaces or ignore the effect of the microstructure. In this work, a simulation is carried out to investigate the effect of microstructure on coalescence-induced droplet jumping. The microstructure with a similar scale to jumping droplet on superhydrophobic will affect the jumping direction and energy conversion efficiency. When the microstructure is perpendicular to the development direction of the liquid bridge, the microstructure will obviously improve the jumping speed and change the jumping direction of the droplet, but when the microstructure is not in the direction of the development of a liquid bridge, it has little influence on the jumping direction of the liquid drop bridge. The wettability of microstructures also affects their effect on jumping. The larger the contact angle of microstructures, the greater the effect on jumping velocity and direction.

Condensed droplet growth and jumping behavior on a superhydrophobic surface

Sihang Gao[1], Fuqiang Chu[2], Xuan Zhang[1], Xiaomin Wu[1]

[1]Department of Energy and Power Engineering, Tsinghua University, China; [2]School of Aeronautic Science and Engineering, Beihang University, Beijing, China

Droplets on the superhydrophobic surface can fall off the surface spontaneously, which greatly promote dropwise condensation. This study considers a continuous droplet condensation process including droplet growth and droplet jumping. A droplet growth model considered NCG is developed and droplet jumping is simulated using VOF(Volume Of Fluid) model. Al-based superhydrophobic surfaces are prepared using chemical deposition and etching method. The Al-based superhydrophobic surface has a contact angle of 157°±1° and a rolling angle of 2°±1°. An observation experiment is designed to observe droplet jumping on superhydrophobic surface using a high- speed camera system. The result of droplet growth model shows a good match with experimental data in mid-term of droplet growth. For droplet jumping, simulation and experiment results show that droplet jumping of different diameter has a universality in a non-dimensional form. The jumping process can be devided into 3 stages and droplet vibration is observed.

The study of the onset of flow boiling in minichannels – heat transfer and dynamic instabilities results

Beata Maciejewska[1], Magdalena Piasecka[1], Artur Piasecki[2]

[1]Kielce University of Technology, Poland; [2]Echo Investment S.A., Al. Solidarności 36 Kielce, Poland

The paper discusses the results for flow boiling heat transfer in minichannels obtained from time-dependent experiments. The main part of the experimental stand was the test section with two minichannels of 1 mm deep. The heated element for FC-72 flowing along the minichannels was a thin foil. In each minichannel temperature of the outer foil surface was measured using different methods: thermocouples or an infrared camera. The main aim of the investigation was to determine the heat transfer coefficient by means of the FEM with time–dependent Trefftz-type basis functions and to recognize dynamic instabilities during boiling incipience. The onset of boiling was induced by two experimental activities: by increasing the heat flux supplied to the heater and by lowering the pressure in the system. The purpose of the work was to investigate the effects of the test method on the boiling incipience initiation and the occurrence of the accompanying instabilities.

Comparison of the 1D and 2D calculation models used for determination of the heat transfer coefficient during flow boiling heat transfer in a minichannel

Kinga Strąk, Beata Maciejewska, Magdalena Piasecka

Kielce University of Technology, Poland

The paper discusses the results of the flow boiling heat transfer in a vertical minichannel with rectangular cross-section. The heating element for FC-72 flowing in the minichannel is a thin plate. Infrared thermography is used to determine changes in the temperature on its outer side. The aim of the calculation is to determine the heat transfer coefficient using 1D and 2D calculation models. Local values of heat transfer coefficient on the surface between the heated plate and boiling fluid are calculated from the Newton`s and Fourier`s laws. In 2D approach the plate temperature distribution is obtained by solving the inverse heat conduction problem. The governing equation is solved by means of two methods: the non-continues Trefftz method and the Beck method. The results are presented as IR thermographs and heat transfer coefficient as a function of the distance from the minichannel inlet, calculated using 1D and 2D models.

Temperature distribution in a composite rod, taking into account non-local spatial effects

George Kuvyrkin, Inga Savelyeva, Daria Kuvshinnikova

Bauman Moscow State Technical University, Russian Federation

One of the important directions in the field of creating new structural and functional materials is the study of materials with a complex structure and unique strength or temperature properties. Due to the heterogeneity of their structure, such materials are called “structurally sensitive”. The heterogeneity of the structure of such materials is justified by the technological features of their creation. Structurally sensitive materials can be obtained by compacting nanopowders, deposition on a substrate, etc.

The paper considers a mathematical model of heat propagation in a composite rod. At one of the ends of the rod, a protective coating of structurally sensitive material is used. Using the finite element method, numerical solutions are found for a problem with conditions of ideal thermal contact of rod parts. The results of numerical calculation for various materials are given. The influence of material parameters on the heat distribution in the rod is analyzed.

On the optimum experiment and heating times when estimating thermal properties through the plane source method

Giampaolo D'Alessandro, Filippo De Monte

University of L'Aquila, Italy

Numerical simulations of optimal experiments related to a three-layer device (thin heater sandwiched between two identical specimens) useful for thermal property estimation of high conductivity materials are performed. In detail, this three-layer device is reduced to a single finite layer (sample) subject to a sixth kind boundary condition at the heated boundary. This non-classical boundary condition account for both thermal inertia of the heat source and imperfect thermal contact occurring at the sample-heater interface. In particular, the former is considered through the volumetric heat capacity of the heater (modeled as a lumped capacitance body), while the latter by means of a surface contact resistance. A finite heating period is also considered. Then, correlated measurement errors are simulated for several experiments aimed at estimating different combinations of unknowns such as sample thermal properties (thermal conductivity and volumetric heat capacity) and contact resistance.

Numerical solution of axisymmetric inverse heat conduction problem by the Trefftz method

Sylwia Hożejowska, Magdalena Piasecka, Tomasz Musiał

Kielce University of Technology, Poland

In this paper the issue of flow boiling heat transfer in an annular minigap was discussed. Main element of the experimental stand was the test section with a of 1 mm width. created between the metal pipe with a heating surface contacting fluid and the external glass pipe. Thermocouples were used to measure the temperature of the heating surface. Mathematical model assumed that in the test section the fluid flow was laminar and the steady state heat transfer process was axisymmetric. The temperatures of the heating surface and of the fluid were assumed to fulfill energy equations with adequate boundary conditions The problem formulated in this way was solved by the Trefftz method. The Robin condition was used to calculate the heat transfer coefficient at the fluid – test surface interface. Graphs were used to represent the values of local heat transfer coefficients as a function of the minigap length.

Nanofluid flow driven by the thermal and magnetic forces – experimental and numerical studies

Elzbieta Fornalik-Wajs, Aleksandra Roszko, Janusz Donizak
AGH University of Science and Technology, Poland

Since the nineties, when the first nanofluid preparation was reported, these fluids are attracting more and more attention. It is mainly due to their potential in heat transport processes.

Introduction of nanofluids in the strong magnetic field has the same aim – the heat transfer enhancement. Additional goal is connected with a deep understanding of the weakly magnetic nanoparticles behaviour in the fluid influenced by this environment.

Because of the nanofluids opaqueness, it is not possible to use the optical experimental methods for the investigations of transport phenomena. Therefore, the spectral and Fast Fourier Transform (FFT) analyses were conducted to get the information about flow structure occurring in the enclosure, filled with silver nanofluid under operation of the strong magnetic field. At the same time, the numerical studies were performed as the complementary ones and the source of detail data. Very good agreement between the results was found.

Experiment and simulation based on a polymer electrolyte membrane (PEM) air dehumidification system

He Yong Li
South China university of technology, China, People's Republic of

Air humidity is closely related to people's lives and production. Excessive humidity will make people feel uncomfortable and affect the quality of product, especially for electronics and precision manufacturing. Because of the fast humidity control, safe operation, small volume and environmental friendliness, electrolytic dehumidification based on a polymer electrolyte membrane (PEM) is promising in recent years. In this paper, an experimental platform of electrolyte membrane dehumidification was established and the dehumidification performance of the system was tested under various working conditions. In addition, an 2-D steady theoretical model was developed in this paper. The model was numerically solved with the finite difference method, by self-developed programme in C++ language. The process of heat and mass transfer was analyzed during the electrolytic dehumidification. Furthermore, the the parameter analysis of system performance were studied too. This research provided a good guidance for the performance optimization of PEM-based dehumidification systems.

A thermal network approach for a quick and accurate study of a multiple connected switchgears

Bogusław Samul[1], Remigiusz Łukasz Nowak[1], Janusz Duc[1], Aravind P. Manjunatha[2]
[1]ABB, Corporate Research Center, Krakow, Poland; [2]ABB, Automation Products Ladenburg, Germany

In the presented paper a thermal network approach was proposed for a quick and accurate study of the thermal performance of two switchgears connected together and unsymmetrically energized. The proposed approach based on the full CFD simulation of a single stand alone switchgear which was used to provide a flow resistance coefficients needed by a constructed thermal network. As a part of a CFD simulation, also a short busbar design optimization was done. Next, a thermal network was compared with CFD simulation results to confirm the correctness of its setup. Finally the network was used to build a model of two adjacent switchgears which was verified by performed heat run test. Comparison of the results showed that the proposed thermal network approach, having an advantage of short computational time can be successfully applied for a simplified simulations of complex switchgear systems.

Improving the efficiency of airborne dust sampling strategies in a quarry plant by means of Sequential Gaussian Simulations

Guido Alfaro Degan, Gianluca Coltrinari, Dario Lippiello
Department of Engineering, Roma Tre University, Italy,Rome

This paper is focused on the optimization of PM10 sampling strategies in quarrying plants by means geostatistical tools. To this aim, an intensive investigation program consisting in about fifty airborne dust concentration field surveys in a quarry plant in Italy was planned: a systematic strategy consisting in monitoring each node of a regular grid was then completed. The following variographic analysis of this dataset revealed some lack in coverage of the whole spatial structure of the PM10 variable so avoiding to catch entirely the behaviour of the concentration all over the domain. In order to improve this result an infilling procedure is carried out assuming to add some further samples to the original dataset. The location of these few additions is selected by defining those zones presenting the highest variability with the aid of geostatistical sequential Gaussian simulations.

Numerical study on heat transfer performance of a reciprocating room temperature active magnetic regenerator

Georges El Achkar[1], Bin Liu[1], Rachid Bennacer[1,2]
[1]Tianjin Key Laboratory of Refrigeration Technology, Tianjin University of Commerce, Guangrong Rd 409, Beichen District, Tianjin, 300134, P. R. China; [2]LMT/ENS-Cachan/CNRS/Université Paris Saclay, 61 Avenue du Président Wilson, 94235 Cachan, France

The convective heat transfer between the magnetocaloric material (MCM), which is gadolinium (Gd) particles of radius 1.5 mm, and the working fluid (water) in a reciprocating room temperature active magnetic regenerator (AMR) was numerically investigated. A two-dimensional transient flow model was developed using Comsol Multiphysics, in order to determine the water flow distribution in two AMRs of cross and parallel MCM particles distributions for different inlet velocities of 0.06 m/s, 0.08 m/s, 0.1 m/s and 0.12 m/s. Based on the simulations of the first model, a two-dimensional transient coupled flow and heat transfer model was then developed using Comsol Multiphysics, in order to characterize the convective heat transfer in the AMR of cross MCM particles distribution for the same water inlet velocities.

A numerical analysis of a hybrid PV+WT power system

Marek Jaszczur, Hassan Qusay, Patryk Palej, Sławosz Kleszcz

AGH University of Science and Technology, Poland

Hybrid Renewable Energy Systems (HRES) are an interesting technological solution in the field of power engineering. It is a combination of two or more renewable energy sources which produce electricity in cogeneration. Such a system can enhance the reliability of power supply due to its differentiation of energy sources which is a very important aspect of energy safety. In this paper, the authors performed a numerical analysis of an HRES which consists of photovoltaic panels, wind turbines and energy storage configuration implemented to supply a household with the electrical load. The main objective of this research is to analyse the system in terms of maximum power production in order to obtain high efficiency and ecological profit. In this work the optimisation process is applied to reduce the cost of energy (COE), maximising the system's efficiency and maximising CO2 emission avoidance.

Numerical and experimental analysis of the air stream generated by square ceiling diffusers

Marek Jaszczur[1], Paweł Madejski[1], Sławosz Kleszcz[1,2], Patryk Palej[1]

[1]AGH University of Science and Technology, Poland; [2]Frapol Sp. z o.o., Kraków, Poland

Ceiling diffusers are one of the most important elements of ventilation and air-conditioning installations. They have a significant impact on the speed and temperature distribution of air. Particullary the square ceiling diffusers are designed for use in low- and medium-pressure ventilation low noise levels systems. The diffusers allow obtaining 1-4 way air supply.

Knowledge of the air flow shaped by the diffuser is very important to ensure the comfort of people staying in a ventilated room. CFD methods allow to analyze the fluid flow of this type however the consistency of the results obtained by numerical methods with the experimental one is an important issue is way. The work presents a laboratory stand designed to test ceiling diffusers. The square ceiling diffuser was tested. The work compares the numerical simulation results with the results of measurements at the test stand.

Numerical investigation of advanced gas turbine combined cycle coupled with high-temperature nuclear reactor and cogeneration unit

Michał Dudek, Zygmunt Kolenda, Marek Jaszczur

AGH University of Science and Technology, Krakow, Poland

Nuclear power systems share at present about 15% of the power market and can be the backbone of a carbon-free energy systems. From a practical point of view currently, the most advanced and most effective technology for electricity generation is based on a gas turbine combined cycle. This technology can use natural gas, synthesis gas or crude oil processing products as the energy carriers but at the same time, such system emits harmful gases to the environment. In the present paper, a thermodynamic analysis of the power plant with a high-temperature nuclear reactor and advanced configuration of the gas turbine combined cycle was investigated. The results show that it is possible to achieve thermal efficiency higher than 50% what is not only more than could be offered by any modern nuclear power plant but it is also more than could be offered by traditional coal or lignite power plant.

Prediction of burning velocity and quenching distance of hydrogen flames

Ali Cemal Benim, Björn Pfeiffelmann

Duesseldorf University of Applied Sciences, Germany

Combustion of hydrogen plays an important role in clean energy supply. Hydrogen represents an attractive alternative to storing excess energy. On the other hand, the gasification is a good possibility for a significant increase in plant efficiencies. The gasification product contains significant amounts of hydrogen. From an environmental point of view, the combustion of hydrogen is most welcome because it produces no carbon dioxide when burned.

However, the combustion-related realization of hydrogen combustion is a great challenge. Hydrogen has very different material properties, so that even relatively small proportions can greatly alter the combustion properties of mixture. Combustion of gas mixtures with hydrogen requires new combustion chamber concepts. A challenge for premixed technologies is the increased flashback propensity. Within this framework, it is important to calculate flame speed and quenching distance accurately. To this purpose, different models will be applied and their performance will be assessed by comparisons with measurements.

Thermal performance of gas turbine of a mixed refrigeration process based on exergy analysis

Abdallah Haouam[1], Chaima Derbal[2], Hocine Mzad[1]

[1]Mechanical Engineering Department, Badji Mokhtar University - Annaba, Algeria; [2]Industrial Mechanics Laboratory, Mechanical Engineering Department, Badji Mokhtar University - Annaba, Algeria

The purpose of this paper is to analyze the performance of a gas turbine 87 MW installed at a mixed refrigeration loop in LNG plant using the exergy concepts. The exergy balance was used in addition to the energy balance to estimate the irreversibility of each component of the gas turbine. The results show that destruction and exergetic efficiency of the gas turbine depend on the variation of the ambient temperature, compression ratio and air-fuel. The combustion chamber has the greatest exergy destruction. To improve the efficiency of the gas turbine, it should reduce the temperature of the intake air and choose a better air-fuel ratio.

KEY-03: Keynote Session 3

Time: Thursday, 05/Sep/2019: 9:00am - 11:00am · *Location:* Uniroma Tre
Session Chair: Francesco Asdrubali

Plenary room

9:00am – 10:00am

Heat transfer enhancement in confined impinging jets with nanofluids and metal foams

Oronzio Manca

Università degli Studi della Campania "Luigi Vanvitelli", Italy

The request for an enhancement of the heat transfer has caused the development more efficient systems in several applications such as in automotive, aerospace and electronic fields and process industry. The use of confined impinging jets represents a possible way to realize efficient cooling systems. Furthermore, the addition of nanoparticles in the working fluid is considered in order to enhance the thermal ehavior of the base fluid. Slot and round jet configurations have recently obtained an important attention because they are characterized by high cooling effectiveness, controllability and uniformity. These aspects are particularly convenient for the cooling systems of modern electronic devices, requiring increasing heat fluxes decreasing dimensions, or combustors and turbine blades. The features of impinging jets in porous media are becoming popular because the application of porous media, characterized by high conductivity and porosity, on impinged surfaces further improves heat transfer performances. The thermal and fluid-dynamic behavior of impinging jets in porous media are affected by different parameters and their analysis allow to optimize the heat transfer increase. Numerical results on confined impinging jets with nanofluids and in metal foams are presented to point out the effect of different parameters on heat transfer.

10:00am – 11:00am

Advanced computational transport phenomena models for design innovations in process industries

Krishnaswamy Nandkumar

Louisiana State University, United States of America

The manufacturing technologies of the future for converting chemicals, materials, energy etc will be done in efficient, distributed, modular process equipment where multiphase flows are ubiquitous. Our traditional design approach has been to rely on rules of thumb, pilot scale development and testing of process equipment which takes up to 20 years to develop a single technology. The design procedures are often highly empirical, dismissing the high degree of freedom that an engineer has at early stages of design by making ad-hoc design decisions, but pay the price during scale-up of processes through expensive pilot scale experiments. The question that I address in this presentation is "Can Advanced Computational modeling tools come to our rescue in minimizing the need for pilot scale experiments?" On the fundamental side, advanced algorithms for direct numerical simulation (DNS) and Discrete Element Modelling (DEM) of multiphase flows aid in detailed understanding but for limited size. For dispersed rigid particles the Navier-Stokes equations are coupled with the rigid body dynamics in a rigorous fashion to track the particle motion in a fluid. These classes of algorithms show great promise in attempting to shed light on multiphase flows from which we can extract statistically meaningful average behavior for use in the design of large scale engineering equipment. We call our effort as EPIC (Enabling Process Innovation through Computation) that integrates multiphase flow modelling with process diagnostics, intensification studies and optimization and control as applied to the process industries. Case studies of industrial relevance will be presented to illustrate the benefits of such an approach.

SE-15: Day 3 Session 15

Time: Thursday, 05/Sep/2019: 11:20am – 1:20pm · *Location:* Uniroma Tre
Session Chair: Aldo Fanchiotti

Plenary room

11:20am - 11:35am

DNS study of turbulent elliptical pipe flow under spanwise system rotation

Rafael Hurtado Rosas, Zhao-Ping Zhang, Bing-Chen Wang

Univ. of Manitoba, Canada

The effect of Coriolis forces on the turbulent flow within an elliptical pipe subjected to spanwise rotation has been studied using direct numerical simulation (DNS). In response to system rotation, large-scale secondary flows appear in the cross-stream plane as a pair of counter-rotating vortices, which significantly impact the turbulent statistics and structures of the flow. It is observed that laminarisation occurs on the suction side of the flow and propagates towards the pressure side as the intensity of rotation increases. At a moderate system rotation speed, the Coriolis term of the Reynolds stress transport equation dominates the energy transfer from the streamwise component to the vertical and spanwise components, far surpassing the role of the pressure-strain term. In the full conference paper, detailed statistical results of the velocity field will be analyzed in both physical and spectral spaces. The effect of system rotation on coherent structures will also be investigated.

11:35am - 11:50am

Numerical analysis of influence of the nozzle shape on the flash boiling phenomena

Piotr Łapka[1], Mirosław Seredyński[1], Andrzej Grzebielec[1], Adam Szelągowski[1], Mateusz Śmiechowicz[2], Emil Gromadzki[2]

[1]Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Institute of Heat Engineering, Poland; [2]Eneon sp. z o.o., Poland

In the proposed paper the numerical analysis of influence of the nozzle shape on the steady-state flash boiling phenomena is presented. The equilibrium model of heat and mass transfer was applied. The Zwart-Gerber-Belamri cavitation model was used to describe the dynamics of the liquid water - water vapor phase change process, supplied with the polynomial relationship linking saturation pressure and temperature.

The effect of the shape of nozzle, particularly its decompression zone on the mass flow rate of two-phase mixture was numerically investigated. Simulations were carried out for conical and stepwise geometries of the decompression zone of the nozzle as well as several inlet pressures (i.e., from 2 to 6 bar) and undercoolings (i.e., from 2 to 50 K) of the liquid water.

11:50am - 12:05pm

Variable temperature effects on teg performance

Björn Pfeiffelmann[1], Cansu Özman[1], Ali Cemal Benim[1], Franz Joos[2]

[1]Duesseldorf University of Applied Sciences, Germany; [2]Helmut Schmidt University, Hamburg, Germany

Thermoelectric generators (TEG) offer useful means of converting low-temperature heat to electric power, and, are being increasingly used in various applications, especially for waste heat utilization.

In practical applications, it is quite often the case that the acting temperature distribution along the surface of the TEG is not uniform. However, until now, the effect of such inhomogeneities of the temperature distribution is not analyzed, either computationally, or experimentally.

The purpose of the present work is to fill this gap of knowledge.

Experiments will be performed on a TEG by applying non-homogeneous temperature distributions. The measured cases will be simulated by means of a previously developed computational model. By means of experimental and computational studies, the effect of variable temperature on the TEG performance will be studied.

12:05pm - 12:20pm

Hygrothermal behavior of polystyrene concrete under cyclic solicitations

Maroua Maaroufi[1], Kamilia Abahri[2], Fares Bennai[1], Rafik Belarbi[1]

[1]University of La Rochelle, LaSIE UMR CNRS 7356, France; [2]LMT, ENS Cachan, CNRS, Université Paris-Saclay, Cachan, France

The walls of buildings experience heat, air and moisture transfers that influence indoor climate.

An experimental and numerical analysis of the hygrothermal behavior of polystyrene concrete was led. Samples were subjected to cyclic hydric solicitations. Water content and temperature and humidity profiles were assessed.

Numerical simulations were held using a hygrothermal transfer model elaborated in this work that took into account the influence of temperature and moisture content on the hygrothermal properties such as the thermal conductivity, as they are included in the input parameters.

The experimental results showed that the residual water content in the samples increased over time, and that the relative humidity value at the same time of each cycle of hydric solicitations is not the same. When comparing the experimental and numerical results, there were differences in the humidity profiles inside the samples, due to the hysteresis phenomenon not taken into account in the model.

12:20pm – 12:35pm

Unsteady disposable flow tracking using Discrete Phase Model

Monika Alicja Zielińska, **Adam Sitko**

ABB. Sp.z.o.o. Corporate Research Center, Poland

Thermosetting materials are commonly used as insulators in medium and high voltage applications. Production of large volume elements brought several problems related to exothermic polymerization reaction which may lead to quality issues. To improve the quality, numerical simulations are commonly used for production process prediction and optimization.

One of the most important part of the process is mold filling with fresh, uncured resin. Laminar behavior of the flow may led to creation of sharp transitions between layers with different curing degree, which may lead to local mechanical weakening of the material or degradation of its insulative properties.

Novel approach for resin residence time tracking has been proposed, where Volume of Fluid (VoF) and Discrete Phase Method (DPM) has been coupled. It allows to predict age of resin, even if it is a multiage mixture.

This methodology enables also production of functional materials based on different resin compounds or microencapsulation.

12:35pm – 12:50pm

Comparison between 1-D and grey-box models of a SOFC

Ramin Moradi[1], **Andrea Di Carlo**[2], **Federico Testa**[1], **Luca Del Zotto**[3], **Enrico Bocci**[4], **Emanuele Habib**[1]

[1]Sapienza university of Rome, Italy; [2]University of L' Aquila, Via Campo di Pile, L'Aquila, Italy; [3]Universita Telematica e-Campus, Novedrate, CO, Italy; [4]Marconi University, Via Plinio 24, Rome, Italy

Solid Oxide Fuel Cells (SOFCs) have shown unique performance in terms of greater electrical efficiency and thermochemical integrity with the power systems compared to gas turbines and internal combustion engines. Nonetheless, simple and reliable models still must be defined. In this paper, a comparison between a grey-box model and a 1-D model of a SOFC is performed to understand the impact of the heat transfer inside the cell on the internal temperature distribution of the solid electrolyte. Hence, a significant internal temperature peak of the solid electrolyte is observed for a known difference between anode and cathode inlet temperatures. Indeed, it highlights the difference between the 1-D model and the grey-box model regarding the thermal conditioning of the SOFC. Therefore, the results of this study can be used to investigate the reliability of the thermal results of box models in system-level simulations.

12:50pm – 1:05pm

Investigation of forced convection heat transfer from a heater mounted in a cavity wall using various nanofluids

Rajesh Kanna[1], **Sayed Sayeed Ahmad**[1], **P. Venkata Reddy**[2], **Chithirai Pon Selvan**[3], **Jan Taler**[4], **Dawid Taler**[5], **Paweł Ocłoń**[4], **Andrea Vallati**[6], **David Santosh Christopher**[7]

[1]College of Engineering and Computing, Al Ghurair University, United Arab Emirates; [2]Department of Mechanical Engineering, Amity University, Dubai, UAE; [3]School of Science and Engineering, Curtin University Dubai, UAE,; [4]Institute of Thermal Power Engineering, Cracow University of Technology, Kraków, Poland; [5]Faculty of Environmental Engineering, Cracow University of Technology, 31-864 Kraków, Poland; [6]DIAEE Sapienza University of Rome, Italy; [7]Department of Mechanical Engineering, Wolaita Soda University, Ethiopia

Forced convection heat transfer from heater mounted in a cavity wall is investigated to reveal the relation among volume fraction, nano material properties and flow Reynolds number. The base fluid is considered as water. Saeidi, and Khodadadi [2006] investigated the effect of nanofluid during forced convection heat transfer from a cavity for different inlet and exit ports. They found that the position of outlet port affects significantly the local Nusselt number. Later Sheikhzadeh et al. [2012] reported that the mixed convection heat transfer for various nanofluids. . They tested both Brinkman and the Maxwell–Garnett model and found significant difference among them. Kalidasan et al 2014] investigated natural convection inside the open square enclosure with diagonally placed twin square blocks. The present study is focused on forced convection heat transfer from square heater mounted on a cavity wall. The interesting physics will be reported in connection with volume fraction, Reynolds number and nanomaterial properties.

1:05pm – 1:20pm

Hall effect on steady mhd flow and heat transfer of a viscous fluid in a rectangular channel with suction and injection

Venkata Ramana Murthy Josyula[1], **Pavankumar Reddy Muduganti**[2]

[1]NATTIONAL INSTITUTE OF TECHNOLOGY WARANGAL, India; [2]NATTIONAL INSTITUTE OF TECHNOLOGY WARANGAL, India

The flow of an in-compressible viscous fluid under the influence of an applied uniform magnetic field in a rectangular channel with suction at the adjacent two side walls is studied by considering Hall current and Joule heating effects. The rectangular channel is subjected to a uniform suction from top wall and injection from right wall. An external uniform magnetic field is applied perpendicular to the flow. Two sides (left and bottom) of the channel are kept at two constant but different temperature and other two sides (right and top) are maintained at constant heat flux. Viscous and Joule dissipations are considered in the energy equation. The relevant equations of motion are solved numerically to yield the velocity and the temperature distribution. The current density in x and y directions and pressure is also studied.

SE-16: Day 3 Session 16

Time: Thursday, 05/Sep/2019: 11:20am – 1:20pm · *Location:* Uniroma Tre
Session Chair: Artur Maciag

Room 20

11:20am - 11:35am

Computational performance analysis of an airborne rotor-type electricity generator wind turbine

Doğan Güneş, Ergin Kükrer, Tolga Aydoğdu
Istanbul Bilgi University, Turkey

This paper presents an analysis of the possible performance of a proposed airborne rotor type electricity generator wind turbine design. The innovative design proposal by inventor is based on the rotation of the airborne structure with blades attached to the airborne zeppelin and thus it is called an airborne rotor generator. In this paper computational fluid dynamics analysis of a model close to the proposed design is carried out and the results are presented. The proposed design examples are set to produce 10-100KW. The electrical energy generated through two symmetrically placed alternators at both ends of the zeppelin is transferred to the ground-based system through the tethered cords used to also stabilize the system. Thus, an airborne rotor generator is formed.

11:35am - 11:50am

Identification of the air gap thermal resistance in the model of binary alloy solidification including the macrosegregation and the material shrinkage phenomena

Edyta Hetmaniok, Adam Zielonka, Damian Slota
Institute of Mathematics, Silesian University of Technology, Poland

The aim of this elaboration is to investigate the mathematical model of the inverse one-dimensional problem of binary alloy solidification within the casting mould, with the material shrinkage and the macrosegregation phenomena taken into account. The inverse problem consists in reconstruction of the thermal resistance of the air gap created between the cast and the mould in the course of solidification process on the basis of measurements of temperature in the control point located in the middle of the mould. The solidification process is described by using the model of solidification in the temperature interval, the shrinkage of metal is described on the basis of the mass balance equation, whereas for modeling the macrosegregation process we use the Scheil model. For solving the problem we apply the implicit scheme of finite difference method supplemented by the procedure of correcting the field of temperature and the selected swarm intelligence optimization algorithm.

11:50am - 12:05pm

Large eddy simulation of high atwood number rayleigh-taylor mixing

Ilyas Yilmaz
Istanbul Bilgi University, Turkey

Large Eddy Simulation of Rayleigh-Taylor Instability at high Atwood numbers is performed using recently developed, kinetic energy-conserving, non-dissipative, fully-implicit, finite volume algorithm.

The algorithm does not rely on the Boussinesq assumption. It also allows density and viscosity to vary.

No interface capturing mechanishm is requried. Because of its advanced features, unlike the pure incompressible ones, it does not suffer from the loss of physical accuracy at high Atwood numbers.

Many diagnostics including local mole fractions, bubble and spike growth rates, mixing efficiencies, Taylor micro-scales, Reynolds stresses and their anisotropies are computed to analyze the high Atwood number effects. The density ratio dependence for the ratio of spike to bubble heights is also studied.

Results show that high Atwood numbers are characterized by increasing ratio of spike to bubble growth rates, higher speeds of bubble and spike fronts, faster development in instability, similarity in late time mixing values, and mixing asymmetry.

12:05pm - 12:20pm

Numerical Investigation into Heavy Gas Dispersion in the Atmosphere with Obstacles

Abdullah Saad Alakalabi, Dr Weiming Liu
University of Central Lancashire - UCLan, United Kingdom

When LNG/LPG leaks into the atmosphere, it evaporates and mixes with ambient air to form a heavier-than-air vapour cloud. If the cloud is within flammable limits and encounters ignition sources, an explosion would happen, which could cause a large amount of damage for humans and property. Therefore, it is necessary to study and understand dispersion of the heavy gas cloud. To that end, the work is numerically investigating dispersions of the heavy gases leaked from a point source. Gases with various densities and temperatures are leaked into the atmosphere that form heavy gas clouds. As wind exists, clouds move downstream and mix with air. The shapes and concentrations of the heavy gas clouds are significantly influence by the downstream obstacles. The computations are performed using ansys-cfx 19.1. Then, the computational results will compared with the experimental observations and measurements done by Ayrault et al (1994) and Ayrault et al (1997).

12:20pm - 12:35pm

Numerical analysis on the influence of vane angle and droplet size of swirlvane steam separator for nuclear power generation

Junho Jeon[1], Hongwu Zhao[2], Yeonggyu Park[3], Nanjundan Parthasarathy[1], Yeonwon Lee[1]

[1]Department of Mechanical Design Engineering, Pukyong National University; [2]Interdisciplinary Program of Biomedical, Mechanical and Electrical Engineering, Pukyong National University; [3]Research Institute of Industrial Science and Technology, Pukyong National University

The effect of the swirl vane angle is an important part of steam separator and the effect of droplet size was investigated through Computational Fluid Dynamics (CFD) analysis. The steam flow rate to each outlet, the pressure drop from the inlet to the outlet, the removal rate by collision with the wall, and the steam quality in the direction of the outlet orifice were analyzed. Three angle model cases – namely, 20˚, 30˚, and 40˚ - were used to study the effects of particle removal rate on the wall and the quality of the steam to the outlet orifice. Six droplet sizes were used in this study. The Lagrangian particle tracking technique was used to track the water droplets. The results show that the smallest angle case of 20˚ has the highest removal rate, the smallest critical droplet size and the 100% steam quality to the orifice outlet.

12:35pm - 12:50pm

Computational study of curved underbody diffusers

Angel Huminic, Gabriela Huminic

Transilvania University of Brasov, Romania

Because aerodynamics contributes significantly to the performances of the road vehicles, designers pay recently attention to the aero-control packages. For high-speed vehicles, these could consist on various aero elements, as wings, air splitters, flaps, side skirts, underbody diffusers etc. The later ones are recently widely used also for passenger cars due to the advantages they could generate: reducing of both drag and lift, which have a significant impact related to fuel consumption, aeroacoustics, stability and handling of vehicles.

Thus, the paper presents new results concerning the aerodynamics of the Ahmed body fitted with a non-flat underbody diffuser. As in previous investigations performed, the angle and the length of the diffuser are the parameters systematically varied within ranges relevant for a hatchback passenger car. Coefficients of lift and drag are compared with the values obtained for the flat underbody diffuser, and the results reveal significant improvements concerning aerodynamic characteristics of body.

12:50pm - 1:05pm

Numerical study on the gas flow based on top-hat flow field near the jet flow region

Han Zhang, Li Jia

Beijing Jiaotong University, China, People's Republic of

Numerical analysis was made to study the feasibility of measuring gas flow based on top-hat flow field near the region of jet flow at atmospheric pressure from a circular symmetrical subsonic nozzle. The objective of this study was to determine the change of top-hat flow profile from nozzle flow to jet without solid boundary constraint. Results revealed a close coupling relationship between the expansion effect and the shear force, leading to the change in kinetic energy of each fluid particle on the cross section was different. The velocity gradient of the boundary layer will change first and the core flow will be maintained. In contrast to core flow part, the proportion of boundary layer part is gradually expanding with the increase of flow distance. A thickness parameter was defined with reference to the displacement thickness to evluate the influence of the velocity profile change on the integral gas flow.

SE-17: Day 3 Session 17

Time: Thursday, 05/Sep/2019: 11:20am - 1:20pm · *Location:* Uniroma Tre
Session Chair: Rafal Kobylecki

Room 21

11:20am - 11:35am

Validation methods of geometric 3D-CityGML data for urban wind simulations

Younis Saeedrashed, **Ali Cemal Benim**

Düsseldorf University, Germany

Validation of the quality of the geometric data such as 3D city model is quite crucial for simulation tasks. That is because the simulation process strongly correlated to the quality of the geometric data being meshed. The most errors-resolved and simplified geometric data, the much better obtained quality of the mesh. Thus, the most obtained certainty in the simulation process.

In this paper, we present the methods of validation of the 3D city models. The most common inherited geometrical errors. Thus, the robust solutions are presented, which illustrate that the required closed solid (watertight entity) and closed shells are obtained within the geometrical structures of the 3D city model.

The goal of the paper is to get more understanding of geometric validation and more understanding of the quality of generated mesh. Thus, performing more reliable simulation process in the field of urban wind flow simulation.

11:35am - 11:50am

Numerical Simulation of different pollution sources in an Urban Environment

Gonzalo Fernandez Bartaburu, **Nicolas Rezzano, Mauro D'Angelo, Mariana Mendina**

Facultad de Ingeniería, Universidad de la República, Uruguay

Air pollution has become a problem in Urban Environments due to the microclimate generated by its topography and the multiple sources to be found in them. This paper aims to present a simulation tool generated to predict the pollution concentration in those environments, using a finite volume solver capable of modeling different sources included in the domain.

Three different types of emissions were considered when programming the tool: vehicular emissions, residential emissions spread along the domain and particular chimneys corresponding to different industries or services. Each of this emission is modelled in a different way, and has its own emission rates which are determined by in-situ measurement campaigns and bibliography. The model developed was then applied to different areas in Montevideo, Uruguay, using statistically significant climate conditions, reaching a diagnosis of the current situation and allowing predictions of different scenarios.

11:50am - 12:05pm

Prediction of flow and dispersion in cross ventilated buildings

Ali Cemal Benim, **Ali Nahavandi**

Duesseldorf University of Applied Sciences, Germany

Natural ventilation is an important factor in the development of sustainable and healthy indoor environments. In general, it is driven by wind or buoyancy, or by a combination of both. In the present work, wind-driven ventilation is considered.

A detailed computational analysis of flow and dispersion by cross-ventilation in isolated single-zone buildings is presented. Impact of opening positions is analyzed. Turbulence models at different levels of sophistication are applied to predict the time-averaged and rms values of the velocity and concentration fields. The results are compared with wind tunnel experiments. Based on this validation, the comparative predictive capability of the turbulence models is assessed.

12:05pm - 12:20pm

Modelling of random packed fixed-beds reflecting different loading strategies for detailed-CFD simulations

Ginu George, **Steffen Flaischlen, Gregor Wehinger**

Clausthal University of Technology, Germany

The Fixed-Bed reactors find immense application in chemical and process industries. These reactors are composed of randomly packed catalyst particles through which reactants flow, while chemical reaction takes place. The hydrodynamic characteristics of these reactor types greatly depend on the packing arrangement, which in turn is governed by size and shape of the catalyst particles, its parent container and the loading methods. For the fixed bed reactors with low column (D) to particle diameter (d) ratio (D/d<10), the porosity distribution significantly varies along the radial direction with peaks near the confined wall, which induces channelling effect. Therefore, the conventional CFD approaches such as pseudo homogeneous flow model is not sufficient to accurately predict the local flow behaviour. Indeed, most of the industrial relevant reactor configurations are of with low D/d, where the advanced numerical schemes such as particle resolved CFD simulations are adopted to investigate the local transport phenomena and chemical kinetics. In particle resolved CFD simulations, the fixed bed morphology have to be modelled as realistic as possible. In this, work an open source software Blender is used to model packed bed, which is based on Rigid Body Dynamics. The beds were generated with different loading strategies and aspect ratios of cylindrical shaped pellets, which are common in industry. The important bed parameters such as bulk porosity, radial porosity and particle orientation were compared for different cases and are validated with available experimental data and correlations. A parametric study on friction and restitution factors used for simulating rigid body dynamics using Blender was also carried out.

12:20pm - 12:35pm

Numerical study of Karman vortex of electrically conducting fluids around circular and rounded square cylinders under magnetic fields induced by electric current

Jong Hui Lee, Il Seouk Park
kyungpook national university, Korea, Republic of (South Korea)

When fluid flows pass around a cylinder above certain speed, the flow separates periodically from the upper and lower surfaces of the cylinder, which is well known as Karman vortex or vortex shedding phenomenon. Because this is closely linked with the noise or vibration, many studies coupled with environment or structural safety issues have been conducted. In this study, when electrically conducting fluid flows around a circular or rounded square cylinder where high electric current is passing through, the changes in features of vortex shedding have been investigated. For unsteady laminar flows for different Reynolds numbers (50 - 200), the magnetohydrodynamics simulation has been conducted with gradually increasing Stuart number from 0 to 10. With increasing magnetic field intensity, Strouhal number and lift coefficient for cylinders are decreased because of the flow suppression by electromagnetism. The detailed electromagnetism and the corner round effect of square cylinder are discussed in this paper.

12:35pm - 12:50pm

Experimental and analytical evaluation of a gas liquid energy storage (GLES) prototype

Andrea Vallati[1], Riccardo Stasio[2], Gabriele Battista[2], Roberto de Lieto Vollaro[2], Chiara Colucci[1], Paweł Ocłoń[3]
[1]DIAEE Department of Astronautical, Electrical and Energetical Engineering, “Sapienza” University of Rome, Via Eudossiana 18, 00184, Rome, Italy; [2]Department of Mechanical Engineering, University of Roma Tre, Via della Vasca Navale 79, 00146, Rome, Italy; [3]Institute of Thermal Power Engineering, Cracow University of Technology, al. Jana Pawła II 37, PL-31-864, Kraków, Poland

In recent times, there has been a significant increase in intermittent renewable electricity capacity additions to the generation mix. This, coupled with an aging electrical grid that is poorly equipped to handle the ensuing mismatch between generation and use, has created a strong need for flexible, advanced bulk energy storage technologies. In this paper, one such technology recently invented and demonstrated at DIAEE Department of Sapienza University of Rome is introduced and characterized. Similar to compressed-air energy storage, the Gas Liquid Energy Storage (GLES) technology is based on gas compression/expansion, however, liquid-piston compression and expansion are utilized. This paper reports on the experimental performance of the first GLIES prototype, and presents formulation and results from a validated physics-based simulation model

12:50pm - 1:05pm

Numerical modelling of the fluid flow and the heat transfer during food freezing with the hydrofluidisation method

Michal Stebel, Jacek Smolka, Michal Palacz, Wojciech Adamczyk, Edyta Piechnik
Silesian University of Technology, Poland

Food freezing method of hydrofluidisation is based on submerging small food products in appropriate water solution. The fluid jets flowing upwards the food products from the orifices, intensify the heat transfer and allow to obtain the mean values of the heat transfer coefficient exceeding 2,000 W/(m2K). For this study, the CFD model has been proposed for the system with sphere-shaped objects with a diameter of 5-30 mm instead of food products. Geometrical configurations covered also the vast range of spheres position within the tank and orifice diameters of 2-5 mm. The model validation has been performed using particle image velocimetry technique. Furthermore, the freezing process has been studied for few food products, e.g. radish, strawberry. The time required to reach the initial freezing temperature for the radish was approximately 6 minutes according to the model, while the measured time differed by less than 20 seconds for the same case.

SE-18: Day 3 Session 18

Time: Thursday, 05/Sep/2019: 11:20am - 1:20pm · *Location:* Uniroma Tre
Session Chair: Lucia Fontana
Room 22

11:20am - 11:35am

Modeling and simulation of closed low-pressure adsorbers for thermal energy storage

Micha Schaefer[1], André Thess[1,2]
[1]University of Stuttgart, Germany; [2]German Aerospace Center (DLR)

Closed low-pressure adsorption systems can be applied for thermal energy storage. As the internal heat and mass transfer processes determine the thermal performance, thorough knowledge of these processes is required for further storage development. The present study contributes to this by providing detailed models and conducting simulations over a broad range of parameters and configurations. The focus is on zeolite adsorbers of larger scale (length L = 0.1…1m). Special emphasis is laid on the rarefaction effects (dependency of the heat and mass transfer on the Knudsen number). Powder, granules and honeycombs are considered as adsorbent configuration. The results reveal that the heat and mass transfer as well as the adsorption processes are strongly coupled and can only be understood in their interaction. The rarefaction effects are found to be only relevant for small particle or channel diameters of the adsorbent. Further specific questions regarding modeling and application are analyzed and answered.

11:35am - 11:50am

European Building Resilience To The Climate Change

Virgilio Ciancio[1], Ferdinando Salata[1], Iacopo Golasi[1], Serena Falasca[2], Pieter De Wilde[3], Massimo Coppi[1]
[1]University of Rome 'Sapienza', Italy; [2]University of Urbino, Italy; [3]University of Plymouth, UK

The dynamic simulations for the energy performance of buildings are generally performed using weather files, containing average climate data based on the history of the meteorological quantities used. The forecast simulations carried out in this way do not indicate anything about hypothetical future scenarios related to the climatic changes. This work analyses the future scenarios due to overheating by analyzing and comparing three time steps (2020-2050-2080) in three European locations chosen according to the different Köppen classification. A residential building is used as a model for the simulations, which presents the constructive characteristics relating to recent building techniques. The simulations carried out, for the different time periods studied, allowed the comparison of energy consumption for heating and cooling of living areas in the various geographical areas analysed. The resilience of the building envelope; to the climate changes in progress, were studied along with the relative change in energy requirements.

11:50am - 12:05pm

Performance analysis of a water ejector using CFD simulations and mathematical modeling

Victor Jorge de Oliveira Marum[1], Lívia Bueno Reis[1], Felipe Silva Maffei[2], Shahin Ranjbarzadeh[2], Ivan Korkischko[2], Rafael dos Santos Gioria[1], Júlio Romano Meneghini[2]
[1]Polytechnic School of the University of São Paulo (Dept. Minning and Petroleum Engineering); [2]Polytechnic School of the University of São Paulo (Dept. Mechanical Engineering)

A one-dimensional (1D) mathematical analysis coupled with modeling and simulations by computational fluid dynamics (CFD) of a water ejector is presented. Using data from CFD simulations, the friction loss coefficients of the ejector components were obtained by 1D-mathematical model. In addition, a sensitivity analysis was performed in order to find the ejector parameters that most affect its performance. The CFD approach employed finite volume and finite element methods to test the application of the main turbulence models found in literature for incompressible-flow ejectors. Several operation conditions (OP) were tested and an optimization study was conducted to increase the accuracy of the k-ε turbulence model. Results show that the SST model is the most suitable to capture the ejector flow characteristics in all OP, while the k-ε model has shown good agreement with experimental results below the maximum efficiency point and also above the maximum efficiency point for its optimized version.

12:05pm - 12:20pm

Thermal performance analysis of a residential house equipped with Phase Change Material

Gounni Ayoub, Louahlia Hasna
UNIV, UNICAEN, LUSAC, France

This work reports the results of a numerical study to qualitatively and quantitatively investigate the thermal performance of a residential house built in France and equipped with Phase Change Materials. The impacts of Phase Change Materials (PCMs) equipped in a residential house on heating loads and the indoor comfort are analyzed by means of dynamic simulation using TRNSYS software. Thermal performance of the PCMs are assessed by comparison to a reference case identical to the PCM house but without the PCM layer. The results show that the PCMs have a remarkable effect on the indoor air temperature, thermal comfort and thermal load of the house compared to the reference house. The integration of the PCMs to the building, leads to better thermal comfort conditions compared to the conventional case. A reduction of 52.28 kWh/m²/an is reached when integrating the PCM layer.

12:20pm - 12:35pm

Cooling load and noise characterization modeling for photovoltaic driven building integrated thermoelectric cooling devices

Himanshu Dehra

Egis Group, India

Photovoltaic driven thermoelectric cooling devices are investigated for installation in a modular outdoor test-room. Because of Peltier effect in a thermoelectric cooling (TEC), heating and cooling is achieved by applying a voltage difference across the thermoelectric module. Theoretical design modeling of cooling load and noise characterization of building integrated Thermoelectric (TEC) Devices is analyzed. System design of photovoltaic driven TEC devices is investigated with varying fresh outdoor ventilation rates. Building integrated design of TEC devices inside ceiling suspended duct along with TEC devices mounted on wall driven by rooftop and active façade photovoltaic devices is considered in the analysis. In this way, two-stage dehumidification is achieved by two different sets of TEC devices. The investigation is conducted for effect of voltage, air flow rate and height of fin heat transfer surface. Expressions along with results for noise characterization in photovoltaic driven building integrated TEC devices are also provided.

12:35pm - 12:50pm

Influence of broken twisted tape on heat transfer performance in novel triangular axial corrugated tubes: experimental and numerical study

Suvanjan Bhattacharyya[1], Arnab Banerjee[2], Ali Cemal Benim[3], Rachid Bennacer[4]

[1]Clean Energy Research Group, Department of Mechanical and Aeronautical Engineering, University of Pretoria, Hatfield, South Africa; [2]Department of Mechanical Engineering, MCKV Institute of Engineering, Liluah, Howrah – 711 204. West Bengal. India.; [3]Center of Flow Simulation (CFS), Department of Mechanical and Process Engineering, Duesseldorf University of Applied Sciences, D-40476 Duesseldorf. Germany.; [4]LMT-Cachan / ENS Cachan / CNRS / Université Paris Saclay, 61 avenue du Président Wilson, 94230 Cachan, France.

Influence of broken twisted-tape on heat transfer performance in novel triangular axial corrugated tubes, are investigated experimentally and numerically. The investigations are conducted for Reynolds numbers ranging from 10 000 to 80 000 covering turbulent regimes. The computations are performed with the transition SST turbulence model. Air is used as a working fluid. The typical twisted-tape and broken twisted-tape with two different twist ratios are investigated and three gap ratios were employed for comparative study. The angular pitch (β) among the corrugation and the angle of the corrugation (α) is kept constant through the experiments at 120o and 60o respectively, while two different corrugations height ratio are studied. Eventually, correlations for predicting friction factor and Nusselt number are developed and presented. The predictions are observed to show a good agreement with the measurements and published correlations of other authors. For all examined cases the performance factors are greater than unity.

12:50pm - 1:05pm

Multiobjective optimization of underground power cable systems

Paweł Ocłoń[1], Monika Rerak[1], Ravipudi Venkata Rao[2], Piotr Cisek[1], Andrea Vallati[3], Jarosław Król[1]

[1]Cracow University of Technology, Institute of Thermal Power Engineering, Krakow, Poland; [2]Sardar Vallabhbhai National Institute of Technology Surat, Department of Mechanical Engineering, Surat, India; [3]Department of Ingegneria Astronautica, Elettrica ed Energetica, Sapienza University of Rome, Rome, Italy

This paper presents a modified JAYA algorithm for optimizing of the material costs and electric-thermal performance of an Underground Power Cable System . Power cables arranged in flat formation are considered. Three XLPE high voltage cables are situated in thermal backfill layer for ensuring the optimal thermal performance of the cable system. The study discusses the effect of thermal conductivities of soil and cable backfill material on the UPCS total investment costs. The soil thermal conductivity is assumed constant and equal to 0.8 W/(m K). The cable backfill dimensions and cable conductor area are selected as design variables in the optimization problem. The PSO and JAYA algorithms are used to provide a multiobjective optimization in order to design a cable system in such a way to minimize the cable backfill costs and maximize the electric current flowing through the cables.

SE-19: Day 3 Session 19

Time: Thursday, 05/Sep/2019: 2:20pm - 4:20pm · *Location:* Uniroma Tre
Session Chair: Krishnaswamy Nandkumar

Plenary room

2:20pm - 2:35pm

The numerical simulation of compressible jet at low reynolds number using OpenFOAM

Kraposhin Matvey[1,2,4], Andrey Epikhin[2,3], Kirill Vatutin[2]

[1]Keldysh Institute of Applied Mathematics of the RAS; [2]Ivannikov Institute for System Programming of the RAS; [3]Bauman Moscow State Technical University; [4]Department of Aeromechanics and Flight Engineering of MIPT

The paper presents an analysis of various approaches for calculation the gas-dynamic parameters and acoustic perturbations generated by the compressible jet at low Reynolds number (M = 0.9, Re = 3600). The flow parameters of the jet at selected conditions are well studied and can be used for validation of the numerical methods and schemes. The OpenFOAM software package and various approaches (solvers) such as pimpleCentralFoam, dbnsTurbFoam and new developed solver QGDFoam based on QGD-algorithms were considered. The results of time-averaged flow parameters and acoustic properties were compared with experimental data. To determine the acoustic perturbation the Ffowcs Williams and Hawkings analogy implemented in our OpenFOAM library (libAcoustic) has been used.

2:35pm - 2:50pm

Thermal design optimization of the pcb through area reduction

Cristina Mihaela Dragan[1,2], Dorin Lelea[1]

[1]University Politehnica Timisoara Romania; [2]Continental Automotive Romania

It is known that a large area of the PCB (Printed Circuit Board) increases the thermal performance. When PCB area is enlarged, the volume and the mass increase.

The current paper shows the importance of the shape relative to area of this shape and the way in which the area of the PCB can be reduced but still has similar or even better thermal performance.

An RSM (Response Surface Methodology) of the DOE (Design of Experiments) quality tool from the DfSS (Design for Six Sigma) concept was developed using MINITAB statistical software based on ICEPAK thermal simulation results.

It is demonstrated that the thermal performance is similar or even better for an area reduction up to 30%, if the PCB dimension in the airflow direction is the shortest and the dimension perpendicular with the airflow direction is the longest, compared with a squared shape of the same PCB area.

2:50pm - 3:05pm

Multi-objective analysis of an influence of a geothermal water salinity on optimal operating parameters in low-temperature ORC power plant

Marcin Jankowski, Sławomir Wiśniewski, Aleksandra Borsukiewicz

West Pomeranian University of Technology, Poland

The geothermal energy is the most widely applied energy source for ORC power plant and a lot of studies have been conducted to optimize such a system. The common feature of the calculation methodology in the previous papers was the assumption that the brine was considered as a pure water. In the present study, the salinity of the water is taken as a variable to investigate its influence on optimal operating parameters in ORC power plant. The multi-objective approach is applied in order to conduct the comprehensive analysis, taking total heat transfer area and exergy efficiency as the criteria for the model. The optimization problem is solved using non-dominated sorting genetic algorithm-II (NSGA-II). The results show that the salinity has an impact on the thermophysical properties of the brine. Thus, the objective functions are affected as well, resulting in different values of the optimal operating parameters in the ORC system.

3:05pm - 3:20pm

A comprehensive thermal and structural transient analysis of a boiler's steam outlet header by means of a dedicated algorithm and FEM simulation

Marcin Pilarczyk[1], Bohdan Węglowski[2], Lars Nord[1]

[1]Department of Energy and Process Engineering, Norwegian University of Science and Technology, Norway; [2]Institute of Thermal Power Engineering, Cracow University of Technology, Poland

The aim of this work is to perform thermal and structural analyses of a boiler's outlet steam header with a capacity of 650 t/h of live steam. Based on the measured steam pressure and temperatures on the outer surface of the component, transient temperature fields were determined by means of an algorithm which was validated in the author's previous work. The algorithm allows determination of transient stress distributions on the internal and external surface as well as at stress concentration regions. In parallel, a finite element method (FEM) simulation was performed. Comparison of the obtained results against FEM analysis showed fully satisfactory agreement. The analysis showed that the boiler's start-up time could be reduced because the total stresses do not exceed the allowable values during a regular start up for the analyzed case.

3:20pm – 3:35pm

Flow rate maximization for an electromagnetic fan using a frequency modulation chip

Hsienchin Su

Dastrong Corp, China, People's Republic of

An electromagnetic fan (Emfan) is an oscillating fan like a piezoelectric fan but driven by electromagnetic force. Previous studies show that an Emfan operates at largest fan amplitude when it is driven at the resonant frequency. However, its resonant frequency may change due to the dust accumulation. The resonant frequency of the Emfan used in the experiments is 63 Hz. However, the resonant frequency can be reduced from 63 Hz to 60 Hz after sticking a 0.15 g clay on the fan tip. It also results 84 % decay in fan amplitude. In the study, a frequency modulation chip is proposed. The chip can adjust the frequency of the input current automatically according to the real-time resonant frequency of the Emfan and keep the Emfan operating under resonant condition. The experimental results show that the decay in fan amplitude is almost 0 % by using the chip.

3:35pm – 3:50pm

Numerical simulation of the aerodynamic unsteady loads on the tail fin of a maneuverable aircraft

Andrey Epikhin[1,2], Vladimir Kalugin[1]

[1]Bauman Moscow State Technical University; [2]Ivannikov Institute for System Programming of the RAS

The investigation presented focus on the numerical simulation of the vortex propagation and decay processes which cause fin buffeting loads on aircraft at incompressible subsonic speed. The current study has been performed with OpenFOAM software package using our modifications. The simulation was carried out using a hybrid URANS-LES approach. A series of calculations of three-dimensional flow around a maneuverable aircraft at angles of attack from 0 to 30 deg are conducted. The influence of an airbrake wake on the aerodynamic unsteady loads on the tail fin of a aircraft has been investigated. The results show that vortices shedding from the airbrake interact with the tail fin and cause buffeting loads. For the worst case the amplitude of the peak force acting on the tail fin could be 6 times higher than the average value without airbrake deployed.

3:50pm – 4:05pm

Turbine start-up optimization under the power unit cyclic operating conditions

Andrzej Rusin, Grzegorz Nowak, Henryk Łukowicz, Martyna Tomala

Silesian University of Technology, Poland

New operating conditions of energy systems which give priority to energy production to renewable sources, cause the need to change the way of exploitation of coal power units. Their current role is balancing the energy demand in the system. This in turn causes the change of their work mode from the continuous one (as the basic source of energy production) to the cyclic mode. This type of operation involves the requirement of quick start-ups, frequent power changes and low-load operation. The article analyzes the possibilities of accelerating turbine start-ups from various initial thermal states. The optimization of the start-up processes was carried out taking into account the limitations resulting from the degree of wear of the turbine components. The analyzes considered the increase in low-cycle fatigue of the main elements of the turbine, i.e. rotors, casings and valves. The risk of further operation of such type was also assessed.

4:05pm – 4:20pm

Analyses of heat pump system as an energy efficient alternative of resistance heater for a household appliance: an applicability aspect

Erkan Kutuk, Ozgur Bayer

Middle East Technical University, Turkey

Nowadays more energy efficient household appliances become preferable with increasing environmental consciousness. Therefore, improvements in production technologies and new innovative designs make us possible to produce white goods that consuming less resources.

The present study investigates the experimental procedure for the applicability of heat pump system aiming better performance in water heater included household appliances.Experiments with current and new heat pump system integrated water heater have been done with various compressor speed and air flow rates.Energy consumption, noise level and heating duration are recorded for the considered experiment matrix.

In the preliminary experiments, energy consumption of heating water up to 60C is decreased by 40% when compared resistance heater with heat pump system. However, heating process duration is extended by 22%,and noise level is increased by 9%.With different compressor and fan speeds, energy efficiency, noise level and heating time are changed and recorded in order to determine optimum working points.

SE-20: Day 3 Session 20

Time: Thursday, 05/Sep/2019: 2:20pm – 4:20pm · *Location:* Uniroma Tre
Session Chair: Bujalski Wojciech

Room 20

2:20pm - 2:35pm

Swirled Injector modelling for cavitating multiphase Flow

Bartosz Ziegler[1], Jędrzej Mosiężny[2], Natalia Lewandowska[3]

[1]Poznan University of Technology; [2]Poznan University of Technology; [3]Poznan University of Technology

The article presents a 1-D numerical model of open-type swirled injector designed for a hybrid rocket engine nitrous oxide injection. The model uses HEM (Homogenous Equilibrium Model) approach with highly accurate saturated fluid EOS, for near critical conditions. The results are compared with VoF analysis performed in Fluent. Flow of nitrous oxide and carbon dioxide are compared, following a common practice to test nitrous oxide fuelling and injection devices with safe and cost efficient carbon dioxide due to their thermophysical similarity. Model results are validated against available experimental data and conclusions about practical implementation of presented models are made.

2:35pm - 2:50pm

Shrinkage induced flow and free surface evolution during solidification of pure metal

Aniket Dilip Monde[1], Anirban Bhattacharya[2], Prodyut Ranjan Chakraborty[1]

[1]Indian Institute of Technology Jodhpur, India; [2]Indian Institute of Technology Bhubaneswar, India

A numerical model is developed to study Shrinkage induced convection and free surface evolution caused by the density difference between the solid and liquid phases during the solidification of pure aluminum. For the analysis, a 2-D rectangular cavity field with aluminum melt undergoing solidification process is considered. The cavity contains a riser at the top, which is filled with aluminum melt as well. Conservation of mass, momentum, and energy are based on volume averaging technique and are solved using the SIMPLER algorithm. The free surface evolution is captured using the Volume of fluid (VOF) method. The proposed model focuses on predicting shrinkage induced surface defects during the solidification process. Two case studies are performed involving solidification ensued (i) solely from the bottom surface, and, (ii) from all sides of the mold cavity. The model successfully predicts the evolution of open macro-shrinkage cavity for both the case studies.

2:50pm - 3:05pm

Analysis of the equilibrium and non-equilibrium models of moisture transport in a wet brick.

Mirosław Seredyński, Michał Wasik, Piotr Furmański, Piotr Łapka, Łukasz Cieślikiewicz, Karol Pietrak, Michał Kubiś, Tomasz S. Wiśniewski

Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Institute of Heat Engineering, Nowowiejska Str. 21/25, 00-665 Warsaw, Poland

In the paper equilibrium and non-equilibrium models of moisture transport across the wet building material are proposed and compared. The equilibrium condition between gaseous and liquid moisture form in the first model is relaxed in the second one, where evaporation or condensation is driven by difference between the actual moisture partial pressure and its equilibrium value related to the sorption isotherm. In both models water is assumed in the gaseous phase as well as continuous (funicular) and discontinuous (pendular) liquid phase. Moreover, the transport of moisture is tightly coupled with heat transfer, which is treated as the fully equilibrium process in both models.

The results obtained from proposed models are verified. Additionally, predicted by both models temporal variations of temperature and moisture content in several points in the computational domain are compared. The analyses are carried out applying different hygro-thermal parameters of the process and properties of wet building material.

3:05pm - 3:20pm

Analysis of the thermodynamic performance of the liquid-vapor separation condenser based on the condensate growth model in horizontal tubes

Ce Liu, Li Jia, Zhang Han

Beijing Jiaotong University, China, People's Republic of

This paper proposed a theoretical model to predict the heat transfer performance of the liquid-vapor separation condenser. The model was based on the convection boundary condition and the effects of gravity and shear force to express the feature of the condensate growth inside a horizontal smooth circular tube. A group of experiments were conducted to measure the heat transfer coefficients of the liquid-vapor separation condenser, to validate the model. The refrigerant circuit of the liquid-vapor separation condenser was optimized according to the principle of equal mass flux with predicted results. The thermodynamic performance of the air conditioner with liquid-vapor separation condenser unit in the actual operating condition was calculated and analyzed. The effects of the vapor mass flux and the saturated temperature of the refrigerant on the performance were investigated and the enhancement of the liquid-vapor separator on the heat transfer was verified.

3:20pm – 3:35pm

Effect of the flow velocity on bubble boiling characteristics

Anatoliy Levin, **Polina Khan**

Melentiev Energy Systems Institute SB RAS, Russian Federation

The present research considers the initial stage of the bubble boiling with high heat fluxes releasing from the technical surface. New experimental data on the dynamics of the vapor phase in subcooled water flow in the channel under nonstationary heat release conditions are represented. It was revealed that the effect of the bubble characteristics on the flow velocity is non-monotonic. Moreover, in a narrow range of flow rates, the onset of developed nucleate boiling may be intensified by an increase of the initial flow velocity.

3:35pm – 3:50pm

Application of a fast transonic trajectory determination approach in 1-D modelling of steady-state transonic two-phase carbon dioxide flow

Wojciech Angielczyk, **Dariusz Butrymowicz, Kamil Śmierciew**

BIALYSTOK UNIVERSITY OF TECHNOLOGY, Poland

An original tilizeded procedure of determination of the transonic trajectory has been proposed. The procedure is much faster than the commonly used Newton Critical Point approach. The approach was applied in modelling of a carbon dioxide transonic two-phase flow through the converging-diverging nozzle by means of the Homogeneous Equilibrium Model and Delayed Equilibrium Model (DEM). The simulations concern flows that were experimentally and theoretically investigated in the literature. DEM was previously used only in choked water flow simulations. Its application in CO2 flow modelling and the supersonic trajectory part determination is a novel contribution. The adjusted for CO2 version of the closure law was proposed. The investigation revealed that the applied Darcy friction factor determination approach has a significant influence on the results. Moreover, the models are unable of producing physically acceptable solutions until Lockhart-Martinelli approach is tilized. It was shown that the Friedel approach might be considered more proper for CO2 flows.

3:50pm – 4:05pm

Green's function method for solving the separated two-phase flow in inclined tubes with driving force contributions

Ayelet Goldstein, **Ofer Eyal**

ORT-Braude college, Israel

A fully developed laminar flow is expressed by Poisson equation, in analogy with some electrostatic problems. Several flow configurations can be transformed to electric setups, where the easily measured electric potentials and fields, should be mapped into velocities and stresses.

Our approach supplies an intuitive solution that is superimposed by the terms that motivates the flow i.e. Driving Force Densities (DFD) subjected to the boundary conditions. This approach is not the conventional mathematical derivation of the N-S equations. Two types of driving forces are distributed:

(ii) The surface (DFD) distributed on the cross section of the tube,

(ii) The unknown longitudinal (DFD) on the interface.

In addition this approach can solves for the local observation near the triple points that leads to the derivation of the converging function of the local shear stresses.

We prove that our approach shows a perfect match to the "conservative" one.

4:05pm – 4:20pm

Numerical simulations and analyses of flow rate exchange between accumulator and main loop

Qingliang Meng, **Zhenming Zhao, Huandong Zhang**

Beijing Institute of Space Mechanics & Electricity, China, People's Republic of

In order to study the dynamic behaviors of heat and mass transfer between accumulator and mechanically pumped two-phase loop system, a transient numerical model is developed. By comparison between simulation and test results, it is found that the error of numerical model is in the range of ±10%, which verifies the validity and accuracy of the model. Simulation results show that accumulator will exchange fluid with the main loop in response to heat load variations. In this case, the temperature and pressure of two phase fluid in accumulator, and the total system flow resistance will be affected. The rate and amount of mass transfer between accumulator and main loop will increase with the increase of heat power, and also for the variation trend of temperature and pressure of two phase fluid in the accumulator. The model can be used to study the operating state, flow and heat characteristics.

SE-21: Day 3 Session 21

Time: Thursday, 05/Sep/2019: 2:20pm – 4:20pm · *Location:* Uniroma Tre
Session Chair: Sławomir Pietrowicz

Room 21

2:20pm - 2:35pm

Selected cases of heat transfer phenomena on the shock waves in atmospheric air

Andrzej Tadeusz Wilk, Sławomir Dykas
Silesian University of Technology, Poland

The content of water vapor, liquid water or ice in a dispersed form in the atmospheric air is very common and it might affect aerodynamic characteristics, especially in the transonic or supersonic flow regime. In the paper, special attention will be paid to identifying the heat transfer phenomena appearing on the shock waves due to the phase change process, such as condensation, evaporation, and melting. The in-house CFD code will be employed for performing the numerical analysis. The relation between the shockwave strength and heat transfer intensity on it due to the phase transition will be presented and discussed.

2:35pm - 2:50pm

Heterogeneous Computing (CPU-GPU) for pollution dispersion in an Urban Environment

Gonzalo Fernandez Bartaburu, Mariana Mendina, Gabriel Usera
Facultad de Ingeniería, Universidad de la República, Uruguay

Numerical simulations have proven to be a useful tools when studying flow phenomena in Urban Environments, but due to the problem scale, compromise has to be made when it comes to resolution, given the computational resources needed. Therefore, alternatives are being considered, being one of them the use of heterogeneous computing. This paper aims to present a transition done from a CPU based solver to a CPU-GPU based one in order to simulate pollution dispersion in Urban Environments.

In order to accomplish this, different capabilities of the old solver had to be restructured into the new one, including the models to compute the emission sources and to represent the buildings in the domain through Immersed Boundary Conditions. In this work results for pollutants dispersion in an urban environment in Montevideo, Uruguay, are presented, together with an assessment of the performance of the new GPU based solver, showing interesting performance improvements.

2:50pm - 3:05pm

Numerical investigation of pipelines modeling in solar combined heating and power plant

Roberto Tascioni[1,2], Alessia Arteconi[2,3], Luca Del Zotto[2,4], Emanuele Habib[1], Enrico Bocci[4], Ramin Moradi[1], Khamid Mahkamov[5], Carolina Costa[5], Luisa F. Cabeza[6], Alvaro de Gracia[6,7], Piero Pili[8], André C. Mintsa[9], Matteo Pirro[10], Toni Gimbernat[11], Teresa Botargues[12], Elvedin Halimick[13], Luca Cioccolanti[2]
[1]Sapienza University of Rome, Italy; [2]Università Telematica eCampus, Italy; [3]Università Politecnica delle Marche, Italy; [4]Università degli Studi Guglielmo Marconi, Italy; [5]Northumbria University, UK; [6]Universitat de Lleida, Spain; [7]University of Perugia, Italy; [8]Elianto S.R.L., Italy; [9]Enogia S.A.S, France; [10]S.TRA.TE.G.I.E. srl, Italy; [11]SINAGRO ENGINYERIA S.L.P, Spain; [12]USER FEEDBACK PROGRAM SL, Spain; [13]AAVID Thermacore Europe, UK

In this study a quasi steady-state model of a micro solar Combined Heat and Power plant developed under the EU funded project Innova MicroSolar, is presented. The integrated plant consists of a Linear Fresnel Reflectors solar field, 3.8 tons of Latent Heat Thermal Energy Storage system equipped with reversible heat pipes and an Organic Rankine Cycle unit designed for a power production of 2 kWe/18 kWth.

Previous numerical analyses carried out by some of the authors have revealed the high incidence of the thermal losses of the pipelines connecting the different subsystems on the global performance of the plant. Hence, in this paper a more detailed model of the pipelines has been developed to better estimate the expected thermal losses of the real plant for its potential optimization. The comparison of the results to the previous ones proves that the deviations of the thermal losses are around 8%.

3:05pm - 3:20pm

Numerical method for generalized constitutive laws

Robert Kovacs
Budapest University of Technology and Economics, Hungary

As a result of technological development, micro, and nanoscale materials appear in engineering applications. Moreover, the material heterogeneity is also relevant. In such cases, the classical Fourier's law may not be applicable, and many generalizations exist to overcome this situation.

In these generalized models, one cannot interpret the boundary conditions in a classical way. In parallel, a system of partial differential equations has to be solved numerically. This system consists the balance and constitutive equations and can be more general than the Fourier's law, depending on the phenomenon in question.

In this talk, an efficient numerical method is presented, based on a staggered spatial discretization that allows eliminating certain boundary conditions. The corresponding stability conditions are also discussed.

Various examples are shown to demonstrate how the scheme works in practice. Furthermore, a finite element solution is also presented as a comparison, performed in COMSOL.

3:20pm – 3:35pm

Coupled system of boundary value problems by galerkin method with cubic b splines

K.N.S. Kasi Viswanadham

National Institute of Technology, Warangal, India, India

Coupled system of second order linear and nonlinear boundary value problems occur in various fields of Science and Engineering including heat and mass transfer. In the formulation of the problem, any one of 81 possible types of boundary conditions may occur. These 81 possible boundary conditions are written as a combination of four boundary conditions. To solve a coupled system of boundary value problem with these converted boundary conditions, a Galerkin method with cubic B-splines as basis functions has been developed. The basis functions have been redefined into a new set of basis functions which vanish on the boundary. The nonlinear boundary value problems are solved with the help of quazilinearization technique. Several linear and nonlinear boundary value problems are presented to test the efficiency of the proposed method and found that numerical results obtained by the present method are in good agreement with the exact solutions available in the literature.

3:35pm – 3:50pm

Prediction of roughness effects on wind turbine aerodynamics

Ali Cemal Benim, Micheal Diederich

Duesseldorf University of Applied Sciences, Germany

Erosion can occur at the leading edge of wind turbine blades that have been on operation for a long period. The extent of erosion depends on the temperature, humidity, presence of contaminants in the atmosphere, and wind speed. The consequence is power loss due to the disturbed aerodynamics. On the other hand, for low Reynolds number airfoils, a positive influence of roughness can be expected under certain flow conditions, since the roughness can trigger an earlier transition and, thus, suppress laminar separation. The present work is concerned about the predictability of roughness effects.Two types of modelling approaches are applied. Firstly, the roughness effects are analyzed by resolving the surface structures. Secondly, roughness models are applied that assume a sand-grained roughness. A problem in that respect is the calculation of an equivalent sand-grained for the given periodic/technical roughness. By comparisons with the measured data, the predictive capability of different modeling approaches will be assessed.

3:50pm – 4:05pm

A low-storage Runge-Kutta OpenFOAM solver for compressible low-Mach numbers flows: aeroacoustic and thermo-fluid dynamic applications

Valerio D'Alessandro[1], Matteo Falone[1], Luca Giammichele[1], Sergio Montelpare[2]

[1]Università Politecnica delle Marche, Italy; [2]Università degli Studi "G. d'Annunzio" di Chieti-Pescara

A solver for compressible Navier–Stokes equations is presented in this paper. Low-storage Runge-Kutta schemes were adopted for time integration; on the other hand the finite volume approach available within OpenFOAM library has been adopted for space discretization. Kurganov-Noelle-Petrova approach was used for convective terms, while central schemes for diffusive ones. The aforementioned techniques were selected and tested in order to allow the possibility of solving a broad range of physical phenomena with particular emphasis to aeroacoustic and thermo-fluid dynamics problems. It is worth noting that standard OpenFOAM solution techniques produce an unacceptable dissipation for acoustic phenomena computations. Non–reflective boundary treatment was also considered to avoid spurious numerical reflections.

The reliability and the robustness of the solver is proved by computing several benchmarks. Lastly, the impact of the thermal boundary conditions on the sound propagation was analyzed; it was showed that the surface temperature increase can reduce aeroacoustic perturbations.

4:05pm – 4:20pm

Performance improvement of air amplifier using the response surface method

Myoungwoo Lee[1], Yo-Hwan Kim[1], Youn-Jea Kim[2]

[1]Graduate School of Mechanical Engineering, Sungkyunkwan University; [2]School of Mechanical Engineering, Sungkyunkwan University

Air amplifier is an aerodynamic device that uses a small amount of compressed air to draw in the surrounding air to create a large volume of working fluid flow. Compared to an ejector, this device can produce high air mass gain with low noise and low compressed air volume. It has various design parameters that affect performance. In this study, geometrical configurations (clearance, chamber volume, and curved wall radius) were set by using the central composite design (CCD) method. Numerical analysis was performed using shear stress transfer (SST) turbulence model, and the optimum design of the air amplifier was determined by using the response surface method (RSM). The performance of the air amplifier was evaluated by the outlet flow rate. The results of velocity and pressure distributions were graphically depicted with various configurations and operating conditions.

SE-22: Day 3 Session 22

Time: Thursday, 05/Sep/2019: 2:20pm – 4:20pm · *Location:* Uniroma Tre
Session Chair: Oronzio Manca

Room 22

2:20pm - 2:35pm

CFD ANALYSIS OF PHASE CHANGE BEHAVIOR OF PHASE CHANGE MATERIAL ENCAPSULATED IN INTERNALLY FINNED SPHERICAL CAPSULE

Kumaresan G[1], Santosh R[1], Revanth H[2], Raju G[2], Bhattacharyya S[3]

[1]Institute for Energy Studies, CEG, Anna University, Chennai 600025, India; [2]Global Nodes Engineering Solutions (P) Ltd, Chennai 600078, India; [3]Mechanical & Aeronautical Engineering Department, University of Pretoria, Hatfield 0028, South Africa

Phase change material (PCM) based Thermal Energy Storage (TES) system has gained momentum in the past two decades because of its ability to store/release a large amount of latent heat energy during phase change process. Considering its low thermal conductivity, several attempts have been made earlier to enhance the heat transfer rate of encapsulated PCM's with various container shapes. However, no attempt has been reported to evaluate the heat transfer augmentation in internally finned spherical capsules. In the present study, CFD analysis is carried out to explore the effect of fin orientation on heat transfer enhancement of paraffin PCM filled in spherical capsules. Capsules with no fin, orthogonal fin and circumferential fin configurations were analysed respectively. The CFD results showed that the orthogonally finned spherical capsule resulted in appreciable reduction in total time taken for complete melting/ solidification process than the circumferential fin and no fin configurations respectively.

2:35pm - 2:50pm

Simple convective and radiative heat exchanger model for process modelling

Wilhelm Franz Fuls

University of Cape Town, South Africa

Various process modelling tools offer the user with a simple convective heat exchanger component that requires the design-based process conditions as inputs. The models would then calculate an effective UA value, and make use of gas flow mass ratios to scale the UA value for off-design conditions. These models make the assumption that the contribution of gas radiation is insignificant, hence only applies convection scaling laws. This paper presents an improved model which also takes into account the contribution of the gas radiation, as is often encountered in the first few heaters in coal fired boilers. A simple design ratio of radiation vs convection heat is needed from the user, and the model will then determine an equivalent radiation term and scale it according to the gas temperature at off design. The model performance was validated against a highly discretised heater model that solves the fundamental convection and radiation terms.

2:50pm - 3:05pm

Determination of Nusselt Numbers and Euler Numbers in Depending on Reynolds Numbers for the Compact Tube Bundle of Small Diameter Tubes by Experimental and Numerical Methods of Researches

Valery Gorobets[1], Yurii Bohdan[2], Viktor Trokhaniak[1], Ievhen Antypov[1], Mykola Masiuk[1]

[1]National University of Life and Environmental Sciences of Ukraine; [2]Kherson State Maritime Academy

The work is devoted to experimental research on the determination of the number of Nusselt for a compact bundle of smooth tubes of small diameter. The bundle has an inline arrangement of tubes in the absence of gaps between adjacent tubes of longitudinal rows. Such a bundle, in comparison with the traditional inline arrangement, has significant advantages, as increased coefficient of heat transfer and reduced aerodynamic resistance of the tube bundle. The use of such bundles allows to reduce the size and mass of heat exchangers of the shell-and-tubes type. Experimental investigations of investigated bundle were carried out in open circuit section type wind tunnel of subsonic speeds under various hydrodynamic conditions of the flow.

As a result of experimental and numerical investigations, dependences of the Nusselt numbers Nu and Euler numbers Eu on the Reynolds number Re of the investigated tube bundle were obtained.

3:05pm - 3:20pm

Comparison of 2D and 3D modeling methodologies for flat-grooved heat pipes

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

[1]Middle East Technical University, Turkey; [2]ASELSAN A.Ş., Turkey; [3]Bilkent University, Turkey

Although there are 1D and 2D thermal modes available in the literature, rigorous 3D modeling of the heat pipes is a challenging task. A mathematical model should address evaporation, condensation, and free surface flow phenomena, which should be determined as a result of the computation and should require an iterative procedure for the geometry of the solution domain. In this study, a 3D computational methodology which iterates the geometry of the free surface, and computes the in-groove flow and heat transfer is developed with the help of a commercial CFD program (Fluent) along with Python programming language. To assess the computational performance and accuracy of the 3D modeling, a mathematical model based on a 2D modelwhich neglects the details of fluid flow and heat transfer in the axial direction is also developed. Two approaches are compared for flat grooved heat pipes with different length-over-diameter and filling ratios.

3:20pm - 3:35pm

Disjoining Pressure Instigated Slope Break of a Condensing Film in a Fin-Groove Corner

Osman Akdag[1], Yigit Akkus[1], Zafer Dursunkaya[2]

[1]ASELSAN Inc., Turkey; [2]Middle East Technical University, Turkey

There are several numerical and experimental studies in literature investigating the film condensation in heat spreaders; however, existing numerical models have not been experimentally validated so far, even for the simple geometry of grooved heat pipes. All of these models assume a continuous slope and continuous or zero curvature at the fin-groove corner and neglect disjoining pressure. Experimental observations, however, reveal that there is a rapid change of film profile slope at the fin-groove corner, defined as a "slope break" and attributed to the possible effects of disjoining pressure and axial flow on the fin top [Frontiers in Heat Pipes 1, 023001, (2010)]. The aforementioned assumptions of existing models preclude the resolution of slope break at the corner. In the current study, the condensation on the fin top is modeled without using these assumptions and the effect of disjoining pressure on the liquid film shape is investigated.

3:35pm - 3:50pm

Study on characteristics of heat transfer and flow resistance in porous foam metal

Aiqiang Chen, Sizhong Gu, Lin Chai, Ruonan Wang, Georges El Achkar, Bin Liu

Tianjin University of Commerce, People's Republic of China

In this paper, the traditional rectangular fins with the same parameters was used as the comparison and reference for porous foam metal. And the heat dissipation model of structured fin and foam metal were studied by simulation and experiment in order to compare the characteristics of heat exchange between the two structures. The results show that, inlet flow rate has a great influence on the temperature uniformity of the bottom plate of the straight rib radiator, and the porous material can improve the uniformity of the temperature of the bottom plate. The convective heat transfer coefficient of the porous foam heat exchanger is significantly higher than that of the structured fins under the same conditions, showing a good heat transfer superiority. The index of comprehensive evaluation for flow and heat transfer indicates that the porous structure with a porosity of 55% has the best performance.

3:50pm - 4:05pm

Investigation of the Effects of Vortex Generator in Improving the performance of Heat Pipes

Ali Tarokh

lakeheadUniversity, Canada

Heat pipes (HP) play an essential role in the many industrial applications, especially in the renewable energy-related technologies in which improving the performance of heat exchangers is in focus. Although various researches are performed for enhancing the performance of HPs, there are still different unknowns available that should be studied to have a clear vision from the transport phenomena in HPs. In this study, computational fluid dynamics is employed to simulate and analyze the thermal fluid flow inside the HP. A computer script is developed and linked to the ANSYS-Fluent to model the evaporation, condensation, and phase change phenomenon in the HP. The effects of the vortex generator on the improvement of heat transfer inside the HP are investigated, and results are compared with the available data in the literature. Also, the influence of the thermo-physical properties of the working fluid on the rate of heat transfer is studied.

PS-02: Day 3 Poster Session 2

Time: Thursday, 05/Sep/2019: 4:40pm - 6:00pm · *Location:* Uniroma Tre

Experimental measurement and numerical simulation of dust particle deposition on super-hydrophobic surface

Anjian Pan[1], **Li-zhi Zhang**[1,2], **Hao Lu**[1]

[1]School of Chemistry and Chemical,South China University of Technology, People's Republic of China; [2]Key Laboratory of Enhanced Heat Transfer and Energy Conservation of Education Ministry, Guanngzhou, People's Republic of China

Airborne dust deposition on a large number of energy devices would cause serious efficiency and lifetime reduction, such as heat exchanger surfaces. As a kind of self-cleaning material, super-hydrophobic coating may become a new effective way to mitigating the dust deposition issue. However, mechanism of dust deposition on super-hydrophobic surfaces remains unclear. Thus this paper aims to investigate dust deposition behaviors and mechanisms on super-hydrophobic surface by experimental measurement and numerical simulation. Lattice Boltzmann Method-Immersed Boundary Method-Discrete Element Method will be developed to predict dust deposition process including settling, collision, adhesion and rebound behaviors. Moreover, high-speed micro camera will be used to record particle deposition process and validate the numerical results. The mechanisms and interactions between coating surface energy, particle characteristics, particle incident velocity and particle adhesion or rebound behavior will be studied carefully. The results may be useful to guide the development of high performance self-cleaning superhydrophobic coating

Design of internal supports for double-walled LNG road tanker

Filip Lisowski, **Edward Lisowski**

Cracow University of Technology, Faculty of Mechanical Engineering

Transport and storage of liquefied natural gas (LNG) requires the use of special double-walled cryogenic tanks. To provide a liquid state of LNG, the temperature inside the tank should be equal -163□C under the pressure around 7 bar. Maintaining these parameters requires the use of a vacuum insulation system and the internal supports designed in order to minimize the heat leakage into the tank. Internal supports in mobile tanks should provide a small heat leak and must be able to transfer the complex mechanical loads. Combinations and values of directional mechanical loads as well as thermal loads to be accepted when designing mobile cryogenic tanks are determined by proper standards. In this paper, we proposed the construction of internal supports, which structure allows to reduce the heat leakage into the internal tank while transferring complex mechanical loads. The tests of the presented solution were carried out using thermo-mechanical FEM analysis.

Advanced design technique for serializing a single-channel pump based on the main performance parameters

Sung Kim[1], **Hyeon-Mo Yang**[2], **Young-Seok Choi**[3], **Jin-Hyuk Kim**[4]

[1]Korea Institute of Industrial Technology, Korea, Republic of (South Korea); [2]Korea Institute of Industrial Technology, Korea, Republic of (South Korea); [3]Korea Institute of Industrial Technology, Korea, Republic of (South Korea); [4]Korea Institute of Industrial Technology, Korea, Republic of (South Korea)

This paper presents a high-efficiency design technique for developing the serialized models of a single-channel pump based on the diameter, flow rate and head as the main performance parameters. The variation in pump performance by changing of the single-channel pump geometry was predicted based on computational fluid dynamics. Numerical analysis was conducted by solving three-dimensional steady Reynolds-averaged Navier–Stokes equations with the shear stress transport (SST) turbulence model. The tendencies of the hydraulic performance depending on the pump geometry scale were analyzed with the fixed rotational speed. These performances were expressed and evaluated as the functionalization for designing the serialized models of a single-channel pump.

Modelling of non-stationary pressure fluctuations during boiling in a minichannel

Romuald Mosdorf, **Hubert Grzybowski**, **Iwona Gruszczyńska**

Bialystok University of Technology, Poland

Boiling in a minichannel occurring at a low mass flow rate is accompanied by non-stationary two-phase flow. The analysis of pressure fluctuations during non-stationary boiling in minichannel shows that quasi-periodic changes in flow patterns can be observed in such fluctuations. We can define in such a way the sequences, which are called “oscillating boiling patterns”. In the present paper the model, which allows us to simulate the appearance of “oscillating boiling patterns” has been presented. In the proposed model the mass flow rate changes (because of evaporation and condensation) are modelled by compressible volumes representing various sizes bubbles. In the paper the good quality agreement between experimental data and simulation results has been achieved. Experimental data was collected during the boiling in open minichannel with inner diameter of 1 mm. The pressure drop was measured by silicon pressure sensor MPX12DP at rate of 1 kHz.

Numerical and experimental analysis of the velocity field of air flowing through swirl diffusers

Marek Borowski, **Michał Karch**, **Paweł Madejski**, **Marek Jaszczur**, **Rafał Łuczak**, **Piotr Życzkowski**

AGH University of Science and Technology Kraków, Poland

The air swirl diffusers are popular for the ceiling level air supply system and have been widely used for Indoor Air Quality (IAQ). They are nowadays one of the most popular diffusers commonly used in air-conditioning systems.

In the case of swirl diffusers, fluid flow is usually significantly influenced by the characteristics of different diffuser designs.

In this work, the airflow of swirl diffuser has been studied using Particle Image Velocimetry (PIV) method in order to analyse the mean swirling air flow and geometry influence on the results. The paper presents a comparison of the results of velocity field

measurements using the PIV method and the results obtained by means of numerical analysis CFD. Based on the analysis the velocity flow field was evaluated and the range of effective operation of the diffuser in the axis of flow have been determined.

Control of smoke flow using a jet-fan in an underground car park

Marek Borowski, Marek Jaszczur, Michał Karch, Tomasz Burdzy

AGH University of Science and Technology Kraków, Poland

The aim of this study is to design a fire ventilation system with jet-fan for an underground car park. There is a number of parameters that can affect the flow of smoke that need to be considered.

To avoid mistakes, necessary is visualize the fluid flow and also to directly compare the different design variants.

We can use computer software, specifically CFD (Computational Fluid Dynamics) simulations. By CFD it is possible to better analyse and keep control of the flow of fluid, heat transfer and other related phenomena. It also helps predict the contamination level of Carbon Monoxide, heat and smoke intensity and distribution.

In this study CFD simulations were used to design, test and compare alternatives of fire ventilation system.

An experimental and numerical Investigation of the thermal and non-thermal efficiency for counterflow heat exchanger

Marek Borowski[1], Michał Karch[1], Sławosz Kleszcz[1,2], Grzegorz Waryan[2]

[1]AGH University of Science and Technology Kraków, Poland; [2]Frapol Sp. z o.o., Kraków, Poland

Heating, ventilation and air conditioning systems are responsible for a significant part of total energy consumption in operated buildings. To decrease energy consumption and reduce harmful gases in the near future, all buildings will have mechanical ventilation with heat recovery. In such system heat exchanger is the essential part of the system responsible for effective energy recovery from exhaust air. However, the main disadvantage of this type of unit is heat exchanger freezing during winter. To solve this issue in the present research a special air distribution system in heat exchanger was developed and the results of an analysis are presented.

Proposed in the studies modification eliminate the negative aspect of unit freezing during the winter season as well as ensure proper air humidity.

Thermal performance of a U-shaped thermosyphon containing a PCM suspension

C.J. Ho[1], Rong-Horng Chen[2], H.Y. Hsu[1], Chi-Ming Lai[3]

[1]Department of Mechanical Engineering, National Cheng-Kung University, Taiwan; [2]Department of Mechanical and Energy Engineering, National Chiayi University, Taiwan; [3]Department of Civil Engineering, National Cheng-Kung University, Taiwan

This study numerically investigates the thermal performance of a U-shaped vertical thermosyphon system filled with a phase change material (PCM) suspension. The heating section and cooling section are located at the bottom and top of the loop respectively. Octadecane is used as the PCM. Parametric study of a loop with a fixed geometrical configuration were conducted in the following ranges: modified Rayleigh number= 5x10^3-5x10^5, modified Stefan number = 0.05-5, PCM suspension concentration= 0-10%, PCM particle dimensionless diameter=0.23-0.0023, and modified subcooling factor = 0.2-1.0. The results show that the modified Stefan number has a critical value under the given parametric conditions. When it is less than the critical value, the latent heat effect becomes significant and the temperature in the heating and adiabatic sections of the loop decreases considerably.

CFD analysis of the steam superheater transient state

Mariusz Granda, Marcin Trojan, Jan Taler, Artur Cebula

Cracow University of Technology, Poland

Nowadays, industry requirements referring to the optimization, not only demand better efficiency or durability but also the lower total cost of the project. Maximum temperatures, allowable stress and economics determine materials used during the engineering process. Moreover, different materials in different parts of superheater can be used. Regarding the steady-state, calculations can be derived without major difficulties but transient-state is a much more complex issue, where Computational Fluid Dynamics can be applied. CFD as an engineering tool, that gives a better understanding of the problem, is more and more popular during the optimization process. Appropriate knowledge about heat transfer, fluid dynamics, finite element method is required to find the solution to the given question.

The paper presents a CFD analysis of the transient-state of the steam boiler superheater when attemporator is running. Temperature distribution of the steam, flue gas and maximum wall temperature were determined.

Assessment of the overall thermal resistance of the thermoelectric module system assembly based on rapid and steady state measurement.

Ryszard Buchalik, Krzysztof Rogozinski, Grzegorz Nowak

Institue of Power Engineering and Turbomachinery, Silesian University of Technology, Poland

The paper discusses issues on thermoelectric module measurements whose aim is to determine its performance characteristics. The research is based on the experimental investigation to get the most important parameters (thermal and electric quantities) determining module's operation. One of the unknown considered parameters is the total thermal resistance composed of thermal gaps and conduction within the cell. Research focus on simulation of the entire thermoelectric cell system with attached heat exchangers, its behavior and characteristics relevant to the operating conditions. The measurements were done both in steady state and rapid changes of the electric load (rapid state measurement) to determine the thermal resistance. The experimental results connected with some numerical simulations are presented, interpreted and compared with the manufacturer's data. An influence of subsidiary elements such as clamping force was taken into account.

Prediction of spindle displacement under thermal influence based on regression neural network

Chih-Jer Lin[1], Her-Terng Yau[2], Mao-Chin Houng[2]

[1]Graduate Institute of Automation Technology, National Taipei University of Technology; [2]Department of Electrical Engineering, National Chin-Yi University of Technology

During machining process of a machine tool, its precision is affected by room temperature and the temperature variation on the spindle, the motor and the lead screw. The rising temperature of the components will make the dimension of the main structure deformed thermally. This study focuses on thermal displacement and thermal effect of the spherical tool machine using Poussin belt spindle. The temperature variation and the axial displacement of the main shaft are monitored at the different rotation speeds. The temperature sensor is embedded in the vicinity of the front bearing, the rear bearing to measure the thermal effect and the thermal displacement in the axial direction is measured using KEYENCE laser displacement meter. The multivariable regression analysis (MRA) and the regression neural network (RNN) are used to establish the relationship between thermal deformation and temperature. The results show that the RNN has better prediction accuracy than the MRA.

Analysis of structural indicators of low-temperature water boilers

Jarosław Bartoszewicz, Adam Nygard

Poznan University of Technology, Poland

Main purpose this paper to evaluate excellence in boilers construction. Results contained in this paper are based on analysis of production program of 130 coal boilers manufacturers from Poland. Attributes which are the main subject of comparison and their classify with reference to compatibility with standards requirements consists with parameters indicating energetic properties of construction such as fuel consumption, heating surface, thermal load of heating surface, boiler efficiency, boiler mass, water capacity, slow-burning, and others. Results are compiled in tables and figures which allows to compare construction features boilers in the same standard class.

The latest world research includes e.g.: research on the influence of air distribution in the combustion process on the growth of brown coal combustion effects [3], application of exergy analysis in boiler optimization [1] and use of biomass in commercial boilers [2].

Experimental validation of heat transfer model in underground power cable system

Paweł Ocłoń[1], Janusz Pobędza[2], Paweł Walczak[2], Piotr Cisek[1], Andrea Vallati[3]

[1]Institute of Thermal Power Engineering, Cracow University of Technology, Poland; [2]Laboratory of Technoclimatic Tests and Work Machinery, Cracow University of Technology, Poland; [3]Department of Astronautical, Electrical and Energetic Engineering, University of Rome, Italy

This paper presents the laboratory test stand that is used to experimental validation of underground power cable system models. Determination of temperature distribution in the vicinity of the cable is the main goal of the study. The paper considers a system of three power cables situated in the in-line arrangement buried in the sand. Three electrical heaters of special construction are used in order to simulate heat flux that is generated in the power cables during its operation. The test stand will be placed in the thermoclimatic chamber, which will allow testing the system in various thermal conditions – ambient temperature changes from 20°C to 30°C. The numerical computations of the steady-state temperature fields are performed using the Finite Element Method. Additionally, to include the dry-zone formation effect on the temperature distribution, the sand thermal conductivity is considered as a temperature dependent.

Mechanical pretreatment of lignocellulosic biomass to improve biogas production: Comparison of results for giant reed and wheat straw

Pier Paolo Dell'Omo, Vincenzo Andrea Spena

Università di Roma "La Sapienza", Italy

Effects of a mechanical pretreatment were determined on the methane yield of giant reed stems and wheat straw. Feedstocks were pretreated using a two stages dry milling device, fitting medium and large-scale biogas plants requirements. Untreated and pretreated materials were anaerobically digested in batch reactors. The cumulative biogas production exceeded 212 Nm3 t-1 of volatile solids, showing a 137% gain as compared to raw material. Significant decrease in the acid detergent fiber content was observed in the processed stems. Pretreated straw reached a cumulative methane yield of 250.3 Nm3 t-1 of volatile solids, +49.2% compared to the feedstock.

With reference to the use of biogas in CHP plants, net electric energy output grew by 111.7% and 38.8% compared to the feedstocks for reed and straw, respectively. Energy cost for the extra electric energy produced was estimated at 0.034 and 0.048 € kWh-1 for giant reed and straw, respectively.

A comparative study on the cavitation characteristics between pump and turbine

Keum-Young Jung[3], Md Rakibuzzaman[1], Sang-Ho Suh[1,2]

[1]Department of Mechanical Engineering, Soongsil University, Seoul, 06978, Korea; [2]Department of Mechanical Engineering, Soongsil University, Seoul, 06978, Korea; [3]KSB Korea Ltd., 13, Hannam-daero 20-gil, Yongsan-gu, Seoul, 04419, Korea

Cavitation is an abnormal physical phenomenon which occurs in relatively low-pressure regions in turbomachinery such as pumps and hydraulic turbines. A comparison between the pump and turbine cavitation behavior is a significant and essential process. The work investigates a comparative study of the cavitation characteristics on a centrifugal pump and a hydraulic Francis turbine numerically and experimentally. The current work adopted the Rayleigh-Plesset cavitation model as the source term for inter-phase mass transfer to predict cavitation characteristics. The experimental data were compared with the numerical results and were found to be in good agreement. Results of the comparative study showed that cavitation first occurred at the suction leading edge on the impeller blades and attached cavitation observed on the impeller blade at the lower suction head; however, for the turbine, the development of attached cavitation happened at the runner outlet near the trailing edge on the runner blades.

Numerical simulations and laboratory measurements for permeability analysis in low porous rock samples

Paweł Madejski[1], Paulina Krakowska[2]

[1]AGH University of Science and Technology, Poland, Faculty of Mechanical Engineering and Robotics; [2]AGH University of Science and Technology, Poland, Faculty of Geology, Geophysics and Environmental Protection

The paper presents results of fluid flow simulation in tight rocks being potentially gas-bearing formations. Core samples are under careful investigation because of the high cost of production from the well. Numerical simulations allow determining absolute permeability based on computed X-ray tomography images of the rock sample. Computational fluid dynamics (CFD) give the opportunity to use the partial slip Maxwell model for permeability calculations. Detailed 3D geometrical model of the pore space was an input data. These 3D model of the pore space was extracted from the rock sample using highly specialized software poROSE (porous materials examination SoftwarE), which is the product of close cooperation of petroleum science and industry. The changes in mass flow, depended on pressure difference and tangential momentum accommodation coefficient, was delivered and used in further quantitative analysis. The results of fluid flow simulations were combined with laboratory measurement result using gas permeameter.

Numerical simulation of turbulent combustion in pulverized coal-fired boiler using mixture fraction method

Paweł Madejski[1], Piotr Żymełka[2,3]

[1]AGH University of Science and Technology, Poland, Faculty of Mechanical Engineering and Robotics; [2]PGE Energia Ciepła S.A., Production Economics Department, Rybnik, Poland; [3]Silesian University of Technology, Faculty of Energy and Environmental Engineering, Institute of Thermal Technology, Gliwice, Poland

The paper presents results of numerical simulation of the combustion process inside the industrial boiler. Simulation of pulverized coal combustion process was conducted using CFD model (Computational Fluid Dynamics) of fluid flow and heat transfer together with modeling of turbulent reactive flow mechanism. Modeling of gas phase combustion products was made using a mixture fraction approach and the two-reaction rates model of the devolatilization process (Kobayashi model). The input data used in the simulation were collected during boiler operation and based on the ultimate and proximate analysis of coals. The simulation results show the quality of boiler operation and the analysis was conducted to identify different behaviors of burners and furnace operation, as well as, to present the impact of solid fuel properties on the combustion process. The results have been verified using available data measuring tests of boiler operation and using balance calculation of combustion and heat transfer processes.

Influence of thermal and flow conditions on the temperature distribution in the evaporator tubes

Marek Majdak, Sławomir Grądziel

Cracow University of Technology, Poland

The article presents the results of thermal and flow analysis of the working conditions of neighboring waterwall tubes, loaded with heat streams of different values. The numerical model used for the analysis, allowing to calculate the temperature distribution of the tubes and the fluid flowing through them at each time step depending on the thermophysical parameters of the fluid and the material from which the tubes were made. By using the algorithm it is possible to precisely determine the temperature distribution for tubes, allowing to determine the places where the most divergent temperatures occur and in which thermal stresses of the highest value may occur. Analysis for several adjacent tubes will allow for the effect of temperature differences in the tubes to the temperature of the fin which is connecting them and to collect data that may be used for the determination of stress distribution in the tubes and fins.

Large eddy simulation of co-axial jet non-premixed combustion using FGM model

Jinghua Li

Nanjing University of Aeronautics and Astronautics, China, People's Republic of

In order to study non-premixed combustion flow field of gas turbine model combustors, Large Eddy Simulations (LES) of a methane/air co-axial jet non-premixed combustor are performed coupled with flamelet generated manifolds model and partially premixed steady flamelet model respectively, and the LES results are compared with experimental data. The results shows that: the results of FGM model like velocity profiles, mixture fraction profiles, product and CO mass fraction profiles agree better with experimental data; the two models can both capture the lift-off flame phenomenon in the combustor; the structure of the flame is complicated, which premixed combustion and diffusion combustion coexist, and diffusion flames are mostly around stoichiometric iso-lines while premixed flames are mainly in lean regions.

Liquid metal turbulent heat transfer in cross-flow bundles for advanced nuclear reactors

Jasper Meeusen[1], Alessandro Tassone[2], Fabio Giannetti[2], Vincenzo Narcisi[2], Gianfranco Caruso[2]

[1]KU Leuven, Oude Markt 13, 3000 Leuven, Belgium; [2]DIAEE – Sapienza University of Rome, Corso Vittorio Emanuele II 244, 00186 Rome, Italy

Heavy liquid metals (HLM) are attractive coolants for innovative heat exchangers in both nuclear fission and fusion applications due to their excellent thermal properties. In this paper, the ANSYS Fluent CFD code is used to characterize the fluid dynamics and heat transfer for the case of HLM ($Pr=0.021$) turbulent cross-flow in square and triangular rod bundles, for both loose ($S=1.45$) and tight ($S=1.45$) lattice arrangements. Extensive code validation is performed for water and LM cross-flow cases, finally selecting the k-ω SST model for the purpose of the study. Steady-state simulations are performed for a test geometry with at least 10 rod ranks for uniform wall temperature and heat flux boundary conditions, in the range $Pe=767\div1150$. Numerical results are compared with simplified theoretical models based on experimental data, observing large underprediction (40%÷54%) and slight overprediction (12%÷31%) for the average Nusselt number in square and triangular bundles, respectively.

Method for monitoring transient thermal stresses for a three-dimensional temperature field

Magdalena Jaremkiewicz, Jan Taler
Cracow Uniwersity of Technology, Poland

The paper proposes an effective method of determining thermal stresses in structural elements with a three-dimensional transient temperature field. When determining thermal stresses using the finite element method, it is necessary to know the fluid temperature and the heat transfer coefficient on the internal surface. Both values are very difficult to determine under industrial conditions. In this paper, space marching inverse methods were proposed for the determination of transient fluid temperature and heat penetration coefficient on the internal surface. The temperature and heat flux on the internal surface were determined by measuring the transient temperature in a small area on the external surface of a pressure element that is easily accessible. In order to validate the method, computational and experimental tests were carried out. The proposed method can be used to monitor thermal stresses in elements of the power unit in thermal power plants, both conventional and nuclear power plants.

Gasification of coal dust in a cyclone furnace in an O2/H2O atmosphere

Robert Zarzycki, Justyna Jedras, Rafal Kobylecki, Zbigniew Bis
Czestochowa University of Technology, Institute of Advanced Energy Technologies, J.H. Dabrowskiego 73, 42-201 Czestochowa

This paper presents the results of modelling of coal dust gasification using oxygen and water steam in a cyclone furnace. Combustion process leads to release of the gas composed mainly of CO and H2. Composition of this gas can be controlled through changes in content of O2 and H2O in the gasifying agent. It is possible for the process conditions adopted in the study to obtain the CO content at the level of 50% with the content of the gasifying agent O2/H2O of 50/50%. In the case of the composition of the gasifying agent O2/H2O of 80/20%, the H2 content was 50%. The maximal calorific value of the gas was obtained for O2/H2O 50/50% (11.2 MJ/kg).

Analytical model for maximum heat transfer in wet fins subject to all nonlinearity effects

Debasis Barman, Balaram Kundu
Jadavpur University, India

New Adomian decomposition method is employed to analyze wet fin heat transfer for straight fins with the consideration of convective heat transfer coefficient, convective mass transfer coefficient, and thermal conductivity as a function of local fin temperature. A psychrometric relationship between humidity ratio and fin surface temperature is taken to establish an actual study. From the results obtained from the present analytical analysis, it can be highlighted that the variable heat transfer coefficients decreases both the fin performance and heat transfer rate as well. This effect is significantly pronounced at the optimum design condition for the maximization of heat transfer rate for a given fin volume. Therefore, for establishing an actual design analysis with the consideration of the optimization analysis, inclusion of nonlinearity parameters is extremely important where not only the actual optimum design information but also the error associated with the consideration of any approximation is to be known.

Basic design relationships in an auger reactor for biomass carbonization

Zbigniew Bis[1], Rafał Kobyłecki[1], Robert Zarzycki[1], Bogusław Usowicz[2]
[1]Czestochowa University of Technology, Institute of Advanced Energy Technologies, J.H. Dabrowskiego 73, 42-201 Czestochowa; [2]Institute of Agrophysics of the Polish Academy of Sciences, Doświadczalna 4, 20-290 Lublin

Auger reactors are the most commonly used devices for biomass carbonization, lignocellulose waste and their mixtures with e.g. hard or brown coal. The solution of such a reactor allows for easy control of the carbonization process and obtaining biochar with the required degree of carbonization. The article presents an innovative and simple solution for a reactor called the ABV (Autothermal Biomass Valorization). This solution was used to derive general thermal and flow relationships. After the introduction of the criterion of a stable autothermal carbonization process, these relationships allowed for the determination of critical values of the reactor's geometrical parameters and the equation of the value of the index of the limit mass load to the heating surface of the reactor. These relationships were verified in the model reactor designed by the authors of this study.

Convertion of steam power plant into cogeneration unit - case study

Marcin Panowski[2], Robert Zarzycki[1]
[1]Politechnika Czestochowska, Poland; [2]Politechnika Czestochowska, Poland

This study presents the concept, simulation calculations of the system for heat recovery from flue gas integrated with heat storage facility. The role of the system is to ensure heat supply to the heating system during the heating season and to produce heat for production of hot water. The installation discussed in the paper consists of dedicated heat exchangers allowing for heat collection from flue gases in the process of moisture condensation. An absorption heat pump was installed to increase the heat flux collected from the flue gases, allowing for additional cooling of the flue gases and improving the temperature of the heating medium. The installation of the system proposed in the study allows for the implementation of thermal insulation of conventional condensation units with the lowest possible reduction of their electrical power. Heat recovery from flue gas significantly reduces emissions of CO2 and other harmful substances into the atmosphere.

Influence of Epoxy Composite Coatings on Hydrodynamics and Heat Transfer Processes for Compact Small Diameter Tube Bundles

Valery Gorobets[1], Yurii Bohdan[2], Viktor Trokhaniak[1], Ievhen Antypov[1], Alla Bohdan[2]

[1]National University of Life and Environmental Sciences of Ukraine; [2]Kherson State Maritime Academy, Ukraine

Tube bundles with cross flow heat carrier chart are widely used in many power systems due to their relatively simple design, low cost, low pressure drops, high efficiency etc. In order to exploitation these tube bundles in dirty and aggressive applications, protective coatings can be applied to the heat transfer surface to enhance effectiveness and minimize pollutant deposits. Protective coatings generally using for improving wear resistance, corrosion resistance, thermal conductivity to heat transfer surface and thermal insulation of frame surfaces of heat exchangers.

In this work physico-mechanical and thermophysical properties of heat conduct coating materials for heat transfer surface are investigated. Experimental and numerical investigations of hydrodynamics and heat transfer processes for compact small diameter tube bundles coated by epoxy composite with high thermal conductivity nanofillers are presented.

Experimental investigations of investigated tube bundles were carried out in open circuit section type wind tunnel of subsonic speeds.

On the thermofluid characteristics of a nanofluid for electric motor cooling

Ali Deriszadeh, Filippo de Monte

Università dell'Aquila, Italy

This paper studies fluid flow and heat transfer characteristics of nanofluids as advance coolants for the cooling system of electric motors. Investigations are carried out using numerical analysis for a cooling system with spiral channels. To solve the governing equations, the finite-volume method is used. The base fluid is water with laminar flow. The fluid Reynolds number and turn-number of spiral channels are evaluation parameters. The effect of nanoparticles volume fraction in the base fluid on the heat transfer performance of the cooling system has also been studied. The numerical results show that, with increasing Reynolds number and turn-number of spiral channels, the motor temperature decreases. Investigating the effect of nanoparticles on the base fluid also suggests that by increasing the volume fraction of nanoparticles, heat transfer is improved, but the addition of nanoparticles causes an increase in pressure drop. This paper aims at finding a trade-off between effective parameters.

Solution of inverse problem of non-stationary heat conduction using a Laplace transform

Andrzej Frąckowiak, Michał Ciałkowski

Poznan University of Technology, Poland

This paper presents solved one-dimensional unsteady inverse heat conduction problem of the Cauchy type. Based on known courses of temperature and heat flux on one boundary, a solution in the domain of the Laplace's transform was determined.

Problems related to the stability of the solution to the Cauchy problem in time domain were discussed. Efficiency of the presented method was examined for the inverse problem of reconstruction of shock temperature change and heat flux distribution on the region's boundary.

Results obtained for the shock temperature change on the opposite boundary of the region indicate great stability of the solution. Presented methods enables determining minimal time interval needed for obtaining stable solution to the inverse problem. This interval is shorter for stable course of temperature than for stable course of heat flux.

Prediction of hydrogen flame propagation in a channel with exit contraction

Ali Cemal Benim, Björn Pfeiffelmann

Duesseldorf University of Applied Sciences, Germany

The propagation of a flame front in a homogeneous and initially quiescent hydrogen-air mixture in a channel with exit contraction is numerically analyzed by means of Computational Fluid Dynamics. For the given configuration, the compressibility effects are important, the average pressure increases in time due to the exit contraction, and pressure waves occur, which affect the flame propagation. Flow turbulence is modelled by the Realizable k-ε model. In modelling combustion, turbulence-chemistry interactions are neglected. Predictions are compared with the measurements for evolution of the flame shape, propagation speed and pressure. It is observed that the flame propagation speed, and, thus, the rate of pressure increase are over-predicted by the present approach. Still, a fair qualitative agreement to measurements is observed

Validation of combustion models for lifted hydrogen flame

Ali Cemal Benim, Björn Pfeiffelmann

Duesseldorf University of Applied Sciences, Germany

A computational investigation of a turbulent lifted H2/N2 flame is presented. Various turbulent combustion models are considered that treat turbulence-chemistry interaction in different ways, namely the Eddy Dissipation Model (EDM), the Eddy Dissipation Concept (EDC) and the Flamelet Generated Manifold (FGM). Turbulence is modelled within the framework of Reynolds Averaged Numerical Simulation (RANS), using the Renormalization Group Theory (RNG) k-ε model, which has proven to offer a good accuracy, based on a preceding validation study for an isothermal H2/N2 jet. Results are compared with the published measurements for a lifted H2/N2 flame, and the relative performance of the turbulent combustion models are assessed.

Crank-Nicolson numerical simulation of temperature distribution in gas turbine blade.

Dariusz Jakubek, Paweł Ocłoń

Faculty of Mechanical Engineering, Cracow University of Technology

The elements of turbine power plants work under severe thermal conditions. Intensive exploitation occurs damage and decreasesthe lifespan of cooperating elements. To manage difficult conditions, it is necessary to apply cooling systems that decrease the temperature of the elements most exposed to damage. This paper concentrate on temperature distribution in the gas turbine blade equipped with cooling holes systemon transient heat transfer. The present study requires the specification of internal and external boundary conditions. What is more, in Crank-Nicolson algorithm four different time steps were applied which affect the final result.The heat transfer coefficient of the cooling working surface the of heat pipes was 1,600 W/(m2K).It was found that in comparison of results from Ansys analysis and Crank-Nicolson method different values of temperature were obtained.

The simulation is based on Finite Element Method which uses Crank Nicolson algorithm. The thermal analysis were performed by using software ANSYS-17.2 and Matlab.

Comparison and evaluation of the Shan-Chen model and most common equation of states

Saleh Saeed Baakeem, Saleh Bawazeer, A. A. Mohamad

University of Calgary, Canada

A novel approach for incorporating the Equation of State (EOS) into the Shan-Chen (SC) model is developed. The difference between the previous and the current approaches is in determining the parameters of the EOS. The novel approach has a solid physical foundation than previous approaches. Popular Equation of States including van der Waals, Redlich-Kwong, Carnahan and Starling, and Shan-Chen are compared and evaluated. A modified relationship of the parameter that controls the strength of the inter-particle force, G, as a function of temperature is developed. The modified relationship eliminates the difference between pressures calculated by the EOS and the SC model. Laplace’s law test is considered in this work to evaluate the stability of the scheme. The evaluation of the maximum density ratio of each EOS is presented and compared with previous approaches. The effect of the coexistence density range of each EOS on the surface tension is discussed.

SUGGESTION FOR THE PROPER TREATMENT OF BUOYANCY FORCE EFFECT ON SINGLE PHASE THERMAL STRATIFACTION PHENOMENON

Gong-hee Lee[1,2]

[1]Korea Institute of Nuclear Safety, Korea, Republic of (South Korea); [2]University of Science and Technology, Korea, Republic of (South Korea)

When emergency core cooling system (ECCS) is operated during loss of coolant accident (LOCA) in a pressurized water reactor (PWR), pressurized thermal shock (PTS) phenomenon can occur. Insufficient flow mixing may cause temperature stratification and this will reduce the life of the reactor vessel. Because temperature difference between the hot coolant at the inlet of the cold leg and the cold cooling water at the inlet of the ECCS injection line is 200 K or more, buoyancy force due to density difference might have significant effect on thermal-hydraulic characteristics of flow. Therefore, in this study, the necessity to consider properly buoyancy force term in governing equations, especially turbulent transport equations, for accurate prediction of single phase thermal stratification in cold legs by ECCS injection was numerically studied using ANSYS CFX.

Fabrication and performance tests for a ultra-thin copper flat heat pipe

Bo Shi, Le Xu

Nanjing University of Aeronautics and Astronautics, China, People's Republic of

In this work, a new structural ultra-thin copper flat heat pipe is fabricated and experimentally tested. The overall structure is composed entirely of copper, with deionized water as the working fluid. The total thickness of the heat pipe is less than 0.7 mm, including 0.3mm vapor chamber. The thermal performance of heat pipes is studied under various heating power, inclination angles and liquid-filling ratios. Experiments reveal that when the tilt angle is 90 degrees and the filling rate is 1.05, the FHP has the best heat transfer performance. The optimal operating power of the ultra-thin flat heat pipe varies from 7.1 to 13.7W and with the lowest thermal resistance ranging from 1.12 to 1.26 with different tilt angles.

Comparison of a CCGT plant and the Westinghouse AP1000 NPP based on exergy analysis

Luisa Ferroni, Antonio Natale, Giovanni Molinari, Francesco Vitillo

Sapienza Università di Roma, Italy

The paper shows and analyzes the results of the comparison between the exergetic efficiencies of a CCGT Plant and the Westinghouse AP1000 Nuclear Power Plant, both power plants of 1100 MWe, also highlighting the exergy destruction rate and the exergy destruction ratio of each of the components of the two plants. In particular, the exergy efficiency of the Nuclear Reactor is calculated using a methodology, developed by the authors, which asses the Carnot factor on the detailed modeling of the heat exchange phenomena inside the reactor aimed to obtain the most realistic trend of the fission temperature inside the core.

Simulation of cooling system for photovoltaic-thermal modules

Paweł Ocłoń[1]**, Piotr Cisek**[1]**, Chiara Colucci**[2]**, Vincenzo Spena**[2]**, Jan Taler**[1]**, Artur Cebula**[2]**, Andrea Vallati**[2]

[1]Cracow University of Technology, Poland; [2]Sapienza University

The PV cooling allows one to increase the photovoltaic conversion efficiency, as well as to obtain the waste heat production which may be utilized for various purposes. This paper presents a numerical simulation of the PV system cooling of novel construction. The paper presents TRNSYS transient model of PV cooling circuit. The simulation results are compared with experimental data, collected at a test stand. The measurement results and the numerical simulation shows quite good agreement.

SE-23: Day 4 Session 23

Time: Friday, 06/Sep/2019: 9:00am - 11:00am · *Location:* Uniroma Tre
Session Chair: Pieter Rousseau

Plenary room

9:00am - 9:15am

Identification of improper use of boilers under real operating conditions.

Adam Nygard, Jarosław Bartoszewicz
Poznan University of Technology, Poland

The paper presents works aiming at an analysis of a method of identification of a wrong use of solid fuel boilers under actual conditions. In experiment, additionally to the manufacturer-recommended fuel, wastes given in strictly defined batches were co-incinerated. During the tests, the amount of oxygen in the exhaust gas was monitored. In addition, the increase in carbon dioxide depending on the type and amount of waste discharged was analysed. It was observed that there is a correlation between the level of oxygen and carbon dioxide in the exhaust gas. It was observed that even slight attempt to operate the boiler incorrectly could be detected based on the analysis of the oxygen level in the exhaust gas. Executed tests gave the foundation for defining a method of detection of wastes co-incineration based on oxygen monitoring in the exhaust gas.

9:15am - 9:30am

Experimental and numerical analysis of a bubbling fluidized bed gasifier for renewable energy production

Anna Prati[1], Francesco Selmo[2], Alessandra Nigro[1], Marco Baratieri[1], Michele Larcher[1]
[1]Free University of Bozen, Italy; [2]University of Trento, Italy

The aim of this work consists in investigating the fluid dynamic behavior of a cold bubbling fluidized bed gasifier both from an experimental and a numerical point of view. The experimental analysis has been carried out considering three heights of the bed material and several fluidization velocities. The experimental set up is shown in Fig.1. The experimental average pressure drops have been compared to the corresponding theoretical values, showing a good agreement (Tables. 1-2). The numerical simulations have been performed with a commercial software by using the Kinetic Theory of Granular Flow (KTGF) and the two-fluid Euler-Euler´s model. To validate the numerical models used to perform the simulations, the Power Spectral Densities (PSD) of the numerical and experimental pressure drop time series have been compared, showing a general good agreement. Numerical results provide a deep insight on the bubbles' behavior inside the fluidized bed, as shown in Fig.2

9:30am - 9:45am

Protection by noise in quarrying activities : test methods comparison of the hearing protection devices efficiency

Guido Alfaro Degan[1], Gianluca Coltrinari[1], Dario Lippiello[1], Pietro Nataletti[2], Diego Annesi[2]
[1]Department of Engineering, Roma Tre University, Italy,Rome; [2]Department of occupational and environmental hygiene, INAIL, Monte Porzio Catone (Rome), Italy

The work operations in the quarry activities are the source of many physical agents such as dust and vibrations but especially noise. This last can have relevant negative effects they cause serious problems for the worker's health. In order to prevent the hearing loss, the employer must provide to the workers the HPD (Hearing Protection Devices). They are inserted inside the ear following a specific procedure to ensure the maximum protection by the noise. For this reason, the use training represents an essential aspect. Also the material, which the device is made of, can influence strongly the actual noise attenuation. The study is based on tests with different HPD, some made of polyurethane and others by silicone. Two experimental campaigns were carried out under different conditions with the same measurement methodology. Results highlight the importance of the methodology used in the test and the relevant role of the use training

9:45am - 10:00am

Full and simplified set of dimensionless parameters for fluidized bed hydrodynamic scaling: preliminary simulations

Francesco Selmo[1], Anna Prati[2], Alessandra Nigro[2], Marco Baratieri[2], Michele Larcher[2]
[1]University of Trento, Italy; [2]Free University of Bozen, Italy

This work aims at comparing two different procedures used to downscale a real fluidized bed reactor (of the National Renewable Energy Laboratory, NREL, showed in Fig.1 and described in Tab.1) for biomass gasification. According to the literature, two sets of non-dimensional parameters have been exploited: while the so-called full one takes into account all the necessary dimensionless coefficients, the so-called simplified one doesn't involve the ratio between the particle diameter and the reactor section's main dimension. The two reactors have been simulated with Ansys by means of an Eulerian-Lagrangian scheme Dense Discrete Phase Model – Discrete Element Method, DDPM-DEM). Results focus on the hydrodynamic behaviour of the granular flow and specifically on the bubbles evolution inside the reactor. The two fluidized beds show similar behaviour, but the Power Spectral Density (PSD) analysis of the pressure fluctuations time series higlights relevant differences that cannot be ignored in this application (Figg.2-3).

10:00am - 10:15am

Motion process of horizontal particles laden jet flow

Tooran Tavangar[1], Hesam Tofighian[2], Ali Tarokh[3]

[1]Lakehead university, Canada; [2]Mechanical Engineering Dept., Amirkabir University of Technology, Tehran, Iran; [3]Mechanical Engineering Dept., Lakehead University, Thunder Bay, ON, Canada

Particles laden jet flows can be observed in many industrial applications. In this investigation, the horizontal motion of particles laden jets is simulated using Eulerian-Lagrangian framework. The 2-way coupling is applied to the model to simulate the interaction between discrete and continuum phase. In order to track the continuum phase, a passive scalar equation is added to the solver. Eddy Life Time (ELT) is employed as dispersion model. The influences of different parameters including Stokes number, Reynolds number and mass loading ratio on the flow characteristics are studied. The results of simulation are verified with the available experimental data. It is revealed that by decrease the Reynolds number, the particles path is deflected from the jet because of the gravity force.

10:15am - 10:30am

Effect of droplet superficial velocity on mixing efficiency in liquid-liquid droplet flow

Xiao-juan Li[1], Chang Qiu[1], Zan Wu[2], Jin-yuan Qian[1,2,3], Bengt Sunden[2]

[1]Institute of Process Equipment, College of Energy Engineering, Zhejiang University, Hangzhou, 310027, PR China; [2]Department of Energy Sciences, Lund University, P.O. Box 118, SE-22100 Lund, Sweden; [3]State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University, Hangzhou 310027, PR China

Enhancement of mixing efficiency in droplets is of great importance to extend the industrial applications of the liquid-liquid two-phase flow in micro-scale. In this paper, mixing efficiency in droplets is investigated via the VOF (volume of fluid) method coupled with UDS (user defined scalar) model. The cross-shaped T junction with a square cross-section is designed and used for droplet formation. Different droplet length and droplet formation frequency are obtained by changing the droplet superficial velocity. Relative velocity fields in different droplet superficial velocity are plotted. Effect of droplet superficial velocity on mixing efficiency in droplets are quantitatively analyzed based on the scalar distribution uniformity in droplets. Results show that the mixing efficiency varies with the different droplet superficial velocity. A higher droplet superficial velocity makes a higher mixing efficiency due to the faster inner circulation and shorter droplet length.

SE-24: Day 4 Session 24

Time: Friday, 06/Sep/2019: 9:00am - 11:00am · *Location:* Uniroma Tre
Session Chair: Mirosław Seredyński

Room 20

9:00am - 9:15am

Response of viscoelastic fluid flow in micro cross-slot channel to external stimulation

Chao Yuan[1,2], Hong-Na Zhang[2], Xiao-Bin Li[2], Feng-Chen Li[2]

[1]School of Aeronautics and Astronautics, Sun Yat-Sen University, People's Republic of China; [2]Sino-French Institute of Nuclear Engineering and Technology, Sun Yat-Sen University, People's Republic of China

The flow of viscoelastic fluid in a micro cross-slot channel with a typical length of 100 microns under constant boundary conditions can be divided into three flow regimes: steady-state symmetric flow, steady-state asymmetric flow and random asymmetric flow. In this paper, the response of cross-slot channel flow under external sinusoidal stimulation perturbation is studied by numerical simulation using DNS method. When the viscoelastic fluid viscosity ratio is 0.1 and the relaxation time is 0.5 s, we found that no matter the time of stimulation signal passing through two critical Weissenberg numbers is larger or smaller than the relaxation time of viscoelastic fluid, the steady state asymmetry flow pattern does not appear instead of a steady symmetric flow pattern.

9:15am - 9:30am

Parametric numerical investigations of a cooling of a hot plate by an array of microjets

Piotr Łapka, Adrian Ciepliński

Warsaw University of Technology, Faculty of Power and Aeronautical Engineering, Institute of Heat Engineering, Poland

In this paper the influence of a distance between a nozzle and hot plate and a microjet diameter-based Reynolds number on the heat transfer coefficient on the surface of hot plate was numerically investigated. A square array of 8x8 submerged microjets was considered. The numerical model which was based on the steady-state compressible Navier-Stokes equations and SST k-omega turbulence model was applied for the analysis. During simulations the ratio of distance between the nozzle and hot plate to the microjet diameter was H/d = 3.125, 25 and 50, while the microjet diameter-based Reynolds number was equal to Re_d = 690, 1100 and 1510. The microjet pitch to the microjet diameter ratio was fixed to s/d = 31.25. It was found that H/d ratio and Re_d significantly influence flow patterns in the gap between the nozzle and hot plate as well as heat transfer coefficient on the surface of hot plate.

9:30am - 9:45am

Effect of magnetic field on bifurcation in natural convection through horizontal annulus

Muhammad Usman, Il Seouk Park

Kyungpook National University, Korea, Republic of (South Korea)

In this study the bifurcation phenomenon for natural convection has been analyzed through application of circular magnetic field. The Prandtl number is maintained at 0.3 and diameter ratio is kept equal to 2. For initial values of Hartmann number a single solution in upward or downward flow exists at very low or high Rayleigh number, however over a specific range of Rayleigh number the flow solution is bifurcated and multiple solutions are obtained. For higher values of Hartmann number the solution exists in upward flow region for all values of Rayleigh number. For intermediate values of Hartmann number the solution started to bifurcate with the increase in Rayleigh number and remained in the bifurcation region even for undefined values of the Rayleigh number. The study presented a bifurcation map over varying values of Hartman number and Rayleigh number. Further, the changes in the equivalent thermal conductivity are monitored with increase of Hartman number.

9:45am - 10:00am

Noise reduction of extinguishing nozzle using response surface method

Yo-Hwan Kim[1], Myoungwoo Lee[1], Youn-Jea Kim[2]

[1]Graduate School of Mechanical Engineering, Sungkyunkwan University, Korea, Republic of (South Korea); [2]School of Mechanical Engineering, Sungkyunkwan University, Suwon 16419, Republic of Korea

An inert gas such as nitrogen is used as an extinguishing agent to suppress unexpected fire in places such as computer rooms and server rooms. The gas released with high-pressure causes about above 130dB noise. According to recent studies, the loud noise with the above 120dB has strong vibrational energy and that leads to a negative influence on electronic equipment with a high degree of integration. In this study, a basic fire extinguishing nozzle with absorbent was selected as the reference model and numerical analysis was conducted using the commercial software, ANSYS FLUENT ver. 18.1. Total 28 experiment points were selected by using DOE (design of experiment) method. An optimum point was derived by using RSM (response surface method). Results show that the vibrational energy of the noise reduced by minimizing the turbulence kinetic energy. The pressure and velocity distributions were calculated and graphically depicted with various absorbent configurations.

10:00am - 10:15am

Uncertainty quantification of film cooling performance of an industrial gas turbine vane

Andrea Gamannossi[1], Alberto Amerini[2], Lorenzo Mazzei[2], Tommaso Bacci[2], Matteo Poggiali[2], Antonio Andreini[2]

[1]Department of Engineering and Architecture, University of Parma, Via Università 12, Parma – 43121, Italy; [2]Department of Industrial Engineering, University of Florence, Via S. Marta 3, Firenze – 50139, Italy

This work presents an uncertainty quantification approach applied to CFD. An additive manufactured prismatic model of an industrial gas turbine vane, with EDM manufactured shaped holes, is studied. The achieved adiabatic effectiveness was experimentally characterized in a previous campaign. CFD analyses are conducted on this test case. RANS approach with k-ω

SST turbulence model is used for the simulations. A mixture approach is adopted to measure the effectiveness: air is used for the mainflow, while coolant is fed with CO_2 An uncertainty quantification analysis is performed varying the dimension, streamwise angle, and inlet fillet radius of the holes. Polynomial-chaos approach with probabilistic collocation method is used. This method is able to thoroughly reproduce what Monte Carlo analysis does with remarkable fewer evaluations. Results show good agreement with the experimental findings and prove how these uncertainties are extremely important for the effectiveness evaluation.

10:15am - 10:30am

Conjugate heat transfer analysis using a weak arbitrary grid interface

Mehmet Shala

Ricardo UK Ltd, United Kingdom

This paper presents a weak Arbitrary Grid Interface (AGI) for the coupling of fluid and solid domains. The resulting computational domain is then solved implicitly. The weak-AGI method is particularly useful in dealing with meshes that are non-conformal along the fluid-solid interfaces or in situations where conformal joining is difficult to achieve. A full description of the weak-AGI algorithm is given. Advantages/disadvantages of each joining method and simulation time are also included. An advanced collocated Finite Volume Method is used to discretise the governing equations of fluid flow and heat transfer in unstructured grids using a cell interpolation scheme for poor quality cells. Simulations are performed using Ricardo's VECTIS-MAX multi-domain CFD software. The method is validated against fully conformal grids and published data. Furthermore, the weak-AGI is applied to an engine cooling problem. Results obtained are in good agreement with published results.

SE-25: Day 4 Session 25

Time: Friday, 06/Sep/2019: 9:00am - 11:00am · *Location:* Uniroma Tre
Session Chair: Tadeusz Wójcik

Room 21

9:00am - 9:15am

Prediction of heat and fluid flow in microchannel condensation

Anil Basaran[2], Ali Cemal Benim[1], Ali Yurddas[2]

[1]Duesseldorf University of Applied Sciences, Germany; [2]Celal Bayar University, Manisa, Turkey

The condensing flow in microchannel has gained importance. In the present study, numerical simulations on condensing flow inside the microchannel is conducted to analyze the heat transfer characteristics. Circular microchannel geometries with various diameters are investigated. The Volume of Fluid (The VoF) model is used to model two-phase flow during condensation. The main purpose of using the VoF method is to track the liquid-vapor interface which changes during condensation. The phase-change at the liquid-vapor interface at saturation temperature is modeled with Lee Model. In the considered microchannel geometries, different from the conventional channels, shear stress and surface tension forces can be dominant compared to gravitational forces. Therefore, surface tension is taken into account in the simulations. The numerical simulation results are validated by comparions with the experimental data that exist in the literature. Based on the validated model, working mediums with favorable thermophysical properties but have received less attention are analyzed and assessed.

9:15am - 9:30am

A 2D-numerical study on slot jet applied to a wind turbine as a circulation control technique

Ivano Petracci[1], Luca Manni[2], Matteo Angelino[3], Sandra Corasaniti[1], Fabio Gori[1]

[1]Department of Industrial Engineering , University of Rome "Tor Vergata", Via del Politecnico 1, 00133 Rome, Italy; [2]Cecom srl, Via Tiburtina, km 18,700, 00012 Guidonia Montecelio, Rome, Italy; [3]Loughborough University, Aeronautical and Automotive Engineering, Stewart Miller Building, Loughborough LE11 3TU, UK

A preliminary study of the feasibility of the Circulation Control (CC) technique for wind turbines applications is proposed. The CC was born in aeronautic field, to improve the lift force of the wings allowing the short take-off and landing of aircraft. It consists in blowing air at a relatively high speed over a rounded trailing edge. The thin jet of air uses to remain attached to the convex curved surface and it imposes a certain curvature to the outer streamlines, hence, increasing the lift force of the airfoil. Aim of this study is to numerically investigate the advantages on a wind turbine based on the S809 airfoil, taking into account the energy-related considerations as the cost of the jet production. After a thorough evaluation of the increase of the generated power, it has been found that the application of this technique could be promising with respect to energy-harvesting aims.

9:30am - 9:45am

Modelling of membrane transport in a regenerative hydrogen-vanadium fuel cell

Catalina A. Pino-Muñoz, Nigel P. Brandon

Imperial College London, United Kingdom

A Regenerative Hydrogen-Vanadium Fuel Cell (RHVFC) is an energy storage system based on an aqueous vanadium electrolyte and hydrogen. During discharging, V(V) is reduced to V(IV) and hydrogen is oxidized, while the reverse process occurs in charging mode and hydrogen is stored. This hybrid system has the advantage of fast hydrogen kinetics and absence of cross-mixing.

In order to characterise the capacity loss of the RHVFC, a membrane crossover model is developed including the transport of water and vanadium and sulphuric acid species. The ionic transport through the membrane due to advection, diffusion and migration is modelled with the Nernst-Planck equation. A space charge region at the electrode/membrane interface is allowed by mean of the Poisson equation, which relates space charge density and ionic potential. A study of the dominant crossover processes and the continuous Donnan effect at the interface is presented at different operating conditions.

9:45am - 10:00am

Modeling of dendrite growth in undercooled solution sodium acetate trihydrate

Chanchal Kumar[2], Aniket Dilip Monde[1], Anirban Bhattacharya[3], Prodyut Ranjan Chakraborty[1]

[1]IIT Jodhpur, India; [2]Defence Laboratory, DRDO Jodhpur, India; [3]Department of Mechanical Engineering, IIT Bhubaneswar, India

Sodium acetate trihydrate (SAT) is an aquas solution of sodium acetate popularly known as hot ice. The SAT commonly used as energy storage phase change material in heating pads for body or hand warmer in cold climates. The undercooled melt of sodium acetate trihydrate kept at room temperature results in an exothermic reaction when solidification seed is nucleated. In present work, modeling of denritic growth in an undercooled solution of sodium acetate trihydrate has been carried out. The enthalpy method has been used to compute solid liquid interface growing in undercooled melt. The interface temperature, concentration and grain growth have been modeled considering curvature effect and solutal undercooling. A 2-D computational grid of square control volumes has been used and discreatized governing equations were solved explicitly. The crystal anisotropy was imposed explicitly. The results are validated using experimental data.

10:00am - 10:15am

Heat Shield in the Fairing of a Launcher

Sandra Corasaniti[1], Nicla Di Stefano[1], Fabio Gori[1], Luca Manni[2], Ivano Petracci[1]

[1]Department of Industrial Engineering, University of Rome "Tor Vergata", Via del Politecnico 1, 00133 Rome, Italy; [2]CECOM s.r.l. - Via Tiburtina km 18,700 - 00012 Rome, Italy

The payload fairing is the protection of the payload, typically represented by a satellite. The fairing is designed in order to protect it from the atmospheric agents, the aerodynamic forces and micrometeorites. The jettison of the fairing takes place at an altitude of about 110-120 km, where it is split in two halves, dropped in the radial direction, through a pyrotechnic mechanism. The aim of the present work consists in the numerical analysis of a fairing made up by panels of fiber-reinforced polymer and by an aluminum-honeycomb core. The outermost layer of the fairing is coated with a thermal-resistant blanket of cork to preserve the payload from overheating. The numerical simulation is carried out to evaluate the temperature distribution inside the fairing by setting a known profile of heat flow on the external surface as a boundary condition. A hand-made code allowes to evaluate the thickness of the ablated cork.

Other Abstracts

Numerical and experimental investigation of thermal energy storage processes during phase transformations of heat-accumulating materials with nanoparticles

Valery Gorobets[1], Ievhen Antypov[1], Viktor Trokhaniak[1], Yurii Bohdan[2]

[1]National University of Life and Environmental Sciences of Ukraine; [2]Kherson State

Nowadays for accumulating thermal energy broadly used materials with phase or chemical transformations. The use of these materials gives an opportunity to significantly increase concentration of accumulated thermal energy per unit mass compared with well-known solid and liquid materials. One of the methods for increasing the energy efficiency of heat-accumulating materials is the addition of solid nanoparticles with a large coefficient of thermal conductivity.

The work presents results of numerical and experimental investigation on the influence of nanoparticles of different materials and sizes on the processes of thermal energy storage in paraffin. The thermophysical properties of paraffin with nanoparticles were found by optical spectroscopy method. As a result of experimental and numerical studies, the heat accumulating capacity of the heat-accumulating materials, the dynamics and profile of the melting boundary around cylindrical heat sources are determined. Comparison of heat-storage capacity for heat-accumulating materials with and without nanoparticles is made.