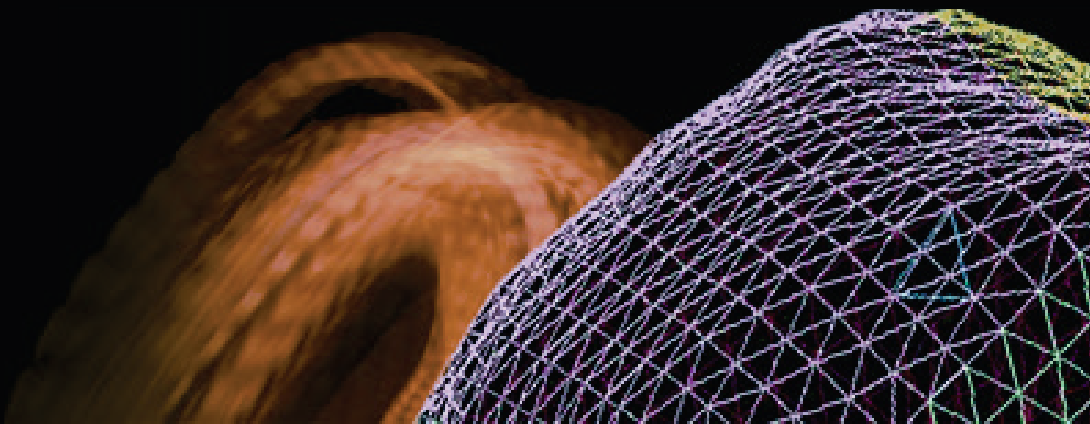
ISSN (Online) 2409-1669
ISSN (Print) 2409-7527

MULTIPHYSICS 2021

9 - 10 December 2021
Virtual



MULTIPHYSICS 2021
9-10 December 2021
Virtual

Conference Board

General Information

Scope of Conference

Understanding real physics and performing Multiphysics simulation are extremely important to analyse complex systems in order to better design and manufacture engineering products.

The objective of the conference is to share and explore findings on mathematical advances, numerical modelling and experimental validation of theoretical and practical systems in a wide range of applications.

The scope of the conference is to address the latest advances in theoretical developments, numerical modelling and industrial application, which will promote the concept of simultaneous engineering. Typical combinations would involve a selection from subject disciplines such as Acoustics, Electrics, Explosives, Fire, Fluids, Magnetic, Nuclear, Soil, Structures, and Thermodynamics.

Timing of Presentations

Each paper will be allocated 18 minutes. A good guide is 14 minutes for presentation with 4 minutes left for questions at the end.

Good timekeeping is essential, speakers are asked to keep strictly to 18 minutes per presentation.

Language

The official language of the conference is English.

Paper Publication

Authors are invited to submit full-length papers for publication in 'The International Journal of Multiphysics' by 31st January 2021.

There is 50% Article Processing Charge (APC) discount for one article per registration.

Sponsorship

The Conference Board would like to thank the sponsors for their support.

Keynote Speaker

H Khawaja
Associate Professor, UiT - The Arctic University of Norway,
and Vice President, The International Society of Multiphysics

BIOGRAPHY

Dr. Khawaja is currently serving as Associate Professor at the Department of Automation and Process Engineering and Research Group Leader of IR, Spectroscopy, and Numerical Modelling Research Group at UiT-The Arctic University of Norway. He is a Chartered Engineer (CEng), Professional Engineer (PE), and holds the posts of Vice President of the International Society of Multiphysics (ISoM), Honorary Secretary of the Association of Aerospace Universities (AAU), and Director & Chair at the Global Listening Centre (GLC). He is also Chairman of the Board and Founding Member of the start-up, 'Windtech AS'. Dr. Khawaja is the recipient of the Multiphysics Student Award back in 2009. He took the role of Coordinator for MULTIPHYSICS Conferences in 2010. He is also the Editorial Manager of The International Journal of Multiphysics (ISSN: 1750-9548). Dr. Khawaja studied his Doctorate at Fitzwilliam College, University of Cambridge, United Kingdom, thesis title: "CFD-DEM Simulations of Two Phase Flow in Fluidised Beds", which he defended in December 2012. He worked as a Post Doctorate on project title: "Multiphysics Investigation of Composite Shell Structures Subjected to Water Shock Wave Impact in Petroleum Industry", funded by the Research Council of Norway (NFR) PETROMAKS with Professor Moatamedi and Professor Souli until September 2013. He has collaborated with scientists/researchers in 22 higher education institutions in 13 countries: China, Ethiopia, France, Pakistan, Poland, Russia, Sweden, Saudi Arabia, Switzerland, United Arab Emirates, United Kingdom, and the United States. He has also collaborated with 13 industry/business/governmental organizations. He has published 65+ publications in the form of patent, books, journal articles, conference proceedings, and given 100+ conferences/seminars/keynote/plenary/invited talks lectures/presentations. He is also the recipient of distinguished prizes such as the W F Reddaway Prize, Commonwealth Scholarship, and President Gold Medal.

MULTIPHYSICS 2021

PROGRAMME

TIME (GMT)	Thursday 9 December 2021	Friday 10 December 2021
09:00 - 09:30	Virtual Platform Access	
09:30 - 11:00	**Keynote Address & Synopsis**	**Session 2.1** *Fluid Mechanics*
11:00 - 11:30	Break / Posters	
11:30 - 13:00	**Session 1.2** *Aerospace and Marine*	**Session 2.2** *Modelling Techniques*
13:00 - 14:00	Break / Posters	
14:00 - 15:30	**Session 1.3** *Coupling Methods*	**Session 2.3** *Environmental Science*
15:30 - 16:00	Break / Posters	
16:00 - 17:30	**Session 1.4** *Multiphysics Simulations*	**Session 2.4** *Posters*
17:30 - 18:30	Virtual Networking	Closing Remarks

Full Programme

Thursday 9 December 2021

09:00 - 09:30 Virtual Platform Access

09:30 - 09:45 Conference Opening

Opening of The 16th International Conference of Multiphysics 2021

T Rahulan, Conference Director, The International Society of Multiphysics

09:45 - 11:00 Session 1.1
Keynote Address & Synopsis

Chair: M Moatamedi, The International Society of Multiphysics

Keynote Address: Computational Fluid Dynamics - A Steppingstone of MULTIPHYSICS

H Khawaja, Associate Professor, UiT - The Arctic University of Norway, and Vice President, The International Society of Multiphysics

Synopsis Part 1: The International Journal of Multiphysics
Synopsis Part 2: The International Conference of Multiphysics 2022

H Khawaja, The International Society of Multiphysics

11:00 - 11:30 Break / Posters

Thursday 9 December 2021

11:30 - 13:00 Session 1.2
Aerospace and Marine

Chair: G Boiger, Zurich University of Applied Sciences, Switzerland

A CFD Investigation on the Impact of Projectile Spin on Base Turbulence Energy and Aerodynamic Drag of a Supersonic Projectile
N Bagalkot, A Fallah, A Keprate
Oslo Metropolitan University, Oslo, Norway

Coupled Aerodynamic and Magnetic Torques for Satellite Stabilization
M Azeem[1], R Varatharajoo[2], S Ahmad[1]
1. COMSATS University, Pakistan
2. University Putra Malaysia, Malaysia

Comparing Single Phase Flow Characteristics and Scavenge Performance of Aero-engine Bearing Chambers
B Chandra
University of the West of England, United Kingdom

Implementing LiDAR for Sea Spray Flux Estimation for the Evaluation of Marine Icing on Ships in Cold Regions
S Dhar, K Edvardsen, H Khawaja, M Naseri
UiT - The Arctic University of Norway

13:00-14:00 Break / Posters

Thursday 9 December 2021

14:00-15:30 Session 1.3
Coupling Methods

Chair: T Watanabe, National Fisheries University, Japan

A Parametric Study of Underground Fiberglass Pipeline Subjected to Fluid Induced Vibration Using Numerical Technique

A Alnahdi[1], E Al-Bahkali[1], M Souli[2], T AlBahkali[1], S Parvez[1]
1. Department of Mechanical Engineering, King Saud University, Saudi Arabia
2. Department Laboratoire de Mécanique, de Lille, Villeneuve d'Ascq, France

High Performance Computing Simulations for Multi-physics Fusion Phenomena

P Bonilla[1], M Mantsinen[1,2], J Lorenzo[1], O Ortega[1], E Goldberg[1], P Pastells[1], X Sáez[1], J Manyer[1], O Fernandez[1], G Houzeaux[1], O Lehmkuhl[1], S Gomez[1], H Calmet[1], A Soba[3,4], X Granados[5]
1. Barcelona Supercomputing Center (BSC), Spain
2. Catalan Institution for Research and Advanced Studies (ICREA), Spain
3. National Atomic Energy Commission (CNEA), Argentina
4. National Scientific and Technical Research Council (CONICET), Argentina
5. Institute of Materials Science of Barcelona (ICMAB), Spain

Simulation of Mechanical Relays under Coupled Electromagnetic and Structural Dynamic Physics

T Lankenau, M Graf
University of Applied Sciences Emden/Leer
Germany

KaleidoSim - Massive Simultaneous Cloud Computing for Multiphysics Simulations

G Boiger[1], D Sharman[1,2], D Drew[2]
1. ZHAW Zurich University of Applied Sciences, Winterthur, Switzerland
2. Kaleidosim Technologies AG, Switzerland

15:30-16:00 Break / Posters

Thursday 9 December 2021

16:00 - 17:30 Session 1.4
Multiphysics Applications

Chair: B Alzahabi, Kettering University, USA

Experimental Identification of the Gas Foil Bearings Operational Conditions - Study on Mechanical and Thermal Properties

J Roemer[1], P Zdziebko[1], A Martowicz[1], S Kantor[1], J Bryła[1], G Żywica[2], P Bagiński[2]
1. AGH University of Science and Technology, Krakow, Poland
2. Institute of Fluid-Flow Machinery. Polish Academy of Sciences, Poland

The Lymphatic Pumping Mechanism: A Numerical Study

M Pepona, G Helbecque, B Kaoui
Biomechanics and Bioenginering Laboratory, Universite de Technologie de Compiegne, Compiègne, France

Surrogate-based Modelling of Composite Panels with Microvascular Channels

S Upnere
Riga Technical University, Latvia

A Semi-Automated Multiphysics Simulation Software for Process Design in the Powder Coating Industry

S Bercan, G Boiger
Zurich University of Applied Sciences (ZHAW)
Winterthur, Switzerland

17:30 - 18:00 Virtual Networking

Friday 10 December 2021

09:30 - 11:00 Session 2.1
Fluid Mechanics

Chair: E Albahkali, King Saud University, KSA

Conservative Multirate Explicit Time Integration Method for CFD
R Messahel, G Grondin, J Gressier, J Bodart
ISAE-SUPAERO, Toulouse, France

Phenomena of Liquid Nitrogen Flashing under Rapid Depressurizations
A Nakano[1], T Watanabe[1], K Kambara[1], K Tokunaga[1], T Shiigi[1], K Shimojima[2]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan

Phenomena of Water Phase Change under Decompression State
K Kambara[1], T Watanabe[1], A Nakano[1], K Tokunaga[1], T Shiigi[1], K Shimojima[2]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan

Numerical Investigation of Incompressible Flow through Cavity attached in a Channel
R Kanna, O Alshaltouni, O Shaqra
Al Ghurair University, UAE

11:00 - 11:30 Break / Posters

Friday 10 December 2021

11:30 – 13:00 Session 2.2
Modelling Techniques

Chair: A Martowicz, AGH University of Science and Technology, Krakow

Comparison and Evaluation of Numerical Techniques and Physico-Chemical Algorithms for the Simulation of an Iodine Ion Thruster
I Barranco-Gomez, C Toomer
University of the West of England, United Kingdom

Geometric Optimisation of Stepped Labyrinth Seals
B Chandra
University of the West of England, United Kingdom

Multiphysics Analysis of Ice-Polyurethane Adhesion under Flexural Loading using FEM Analysis
H Eidesen[1], Z Andleeb[1], H Khawaja[1], M Moatamedi[2,3]
1. UiT-The Arctic University of Norway, Tromsø, Norway
2. Al Ghurair University, UAE
3. Oslo Metropolitan University, Oslo, Norway

Dynamic Response of Monolithic Ductile Membranes Subject to Underwater Shockwave
N Mehreganian[1], Y Novozhilov[2], P Ryabov[2], D Mikhaluk[2], A Fallah[3]
1. University of Liverpool, United Kingdom
2. CADFEM, Russian Federation
3. Oslo Metropolitan University, Oslo, Norway

13:00-14:00 Break / Posters

Friday 10 December 2021

14:00-15:30 Session 2.3 Environmental Science

Chair: R Khanna, Al Ghurair University, UAE

A Key for Reliable Simulation of Horizontal Axis Wind Turbine
M AbdulRaouf[1], E Al-Bahkali[1], S Parvez[1], M Souli[2]
1. Department of Mechanical Engineering, King Saud University, Saudi Arabia
2. Department Laboratoire de Mécanique, de Lille, Villeneuve d'Ascq, France

Phase Change Material, Doped by Nanomaterials for Thermal Energy Storage and Solar Cooling: Synthesis and Characterization
L Maifi, A Kamel, H Ouided, C Abdelhamid, K Tahar
Semiconductors Technology for Energetic Research Center (CRTSE)
Algeria

Windtech Device a Measurement Tool for 'Cold' Sensation
A Leyli, H Khawaja, S Antonsen, D Swart
UiT - The Arctic University of Norway

An Optimal Location of Cooling System Design for a Hybrid Photovoltaic Thermal System
A Abdulbari[1], E Al-Bahkali[1], S Parvez[1], M Souli[2]
1. Department of Mechanical Engineering, King Saud University, Saudi Arabia
2. Laboratoire de Mécanique, de Lille, Villeneuve d'Ascq, France

15:30-16:00 Break / Posters

Friday 10 December 2021

16:00 - 17:30 Session 2.4
Posters

Chair: A Fallah, Oslo Metropolitan University, Norway

Cold Heat Shock Loading to Parasite using the Liquid Nitrogen
T Watanabe[1], M Nakamura[1], K Tokunaga[1], T Shiigi[1], K Shimojima[2]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan

Implementation and Validation of a Thermal Radiation in a 5 axis CNC Aeroplane Milling Machine using COMSOL Multiphysics
P Kalsariya, D Schnee, A Shah, T Schiepp
Hochschule Furtwangen University, Germany

Mass Transfer from Multiple Core-shell Cylinders subjected to Flow: Towards Modeling Artificial Lung-on-a-Chip
P. Lammers, C. Bielinski, A. Bou Orm, B. Kaoui
Biomechanics and Bioengineering, Universite de Technologie de Compiegne, Compiegne, France

Numerical Simulation of Thermoacoustic Oscillation for Nuclear Application
T Watanabe
University of Fukui, Japan

Numerical Study of Viscous Fingering in a Heterogeneous Porous Medium
H Djebouri, S Zouaoui
L.M.S.E. Laboratory, Mechanical Engineering Department, University of Tizi Ouzou, Tizi-Ouzou

Shock Loading Pre-processing for Freeze-drying
T Watanabe[1], M Nakamura[1], K Shimojima[2], S Tanaka[3], K Hokamoto[3], S Itoh[4]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan
3. Institute of Industrial Nanomaterials, Kumamoto University, Japan
4. Institute of Shockwave Applied Technology, Japan

17:30 - 18:00 Close of Conference

SESSION 1.1

KEYNOTE ADDRESS & SYNOPSIS

THURSDAY, 9 DECEMBER 2021
09:30 – 11:00

CHAIR

M Moatamedi
The International Society of Multiphysics

Thursday, 9 December 2021
09:30 – 11:00

Keynote Address

Computational Fluid Dynamics - A Steppingstone of MULTIPHYSICS

H. Khawaja
Associate Professor, UiT - The Arctic University of Norway,
and Vice President, The International Society of Multiphysics

The fluids have been around us for as far as we know and it enabled us to cross the oceans, ride the winds, shortens the distances, and much more. Computational Fluid Dynamics (CFD) has extended our understanding of fluids. Until today, the analytical solution of fundamental equations of fluid mechanics is a million-dollar award (referred to Clay Mathematics Institute). As engineers, let's leave that to mathematicians to claim but that does not mean that we cannot enjoy the power offered by these equations. Smart numerical techniques developed over years such as discretization and time-stepping methods have enabled us to solve them with the help of computers. With the increase of computing power, CFD applications are ever-growing, and literally and figuratively, sky is the limit. Keynote address walks through the evolution of CFD and discusses the historical researchers who contributed to its development. Next, the word of caution concerning numerical modelling and the topic of colourful fluid dynamics is discussed. In relatively recent times, CFD is being solved coupled with other physical equations, e.g., thermal conduction (heat equation), electromagnetics (Maxwell's equations), structures (Hooke's law), and more. This has given birth to a new field known as MULTIPHYISCS. This emerging field of science and engineering can be imagined as the coupled mathematical world, where many overlapping and interactive problems of physics are being solved on the fundamental level.

The keynote address begins with the introduction of the speaker, few personal life experiences, and mainly focuses on evolution and inspiring examples from speaker's past work that enlighten the application of CFD as a steppingstone of MULTIPHYSICS. The examples are: micro-fluidic valve, a collaborative project of NUST and CALTECH; CFD-DEM simulations of two phase flow in fluidised beds, a collaborative project of CAMBRIDGE and ETH ZURICH; multiphysics investigation of composite shell structures subjected to water shock wave impact in petroleum industry, funded by Norwegian Research Council and in collaboration with LILLE University; applied investigation of viscosity-density fluid sensors based on torsional resonators, partially funded by Innosuisse - Swiss Innovation Agency and in collaboration with RHEONICS GmbH and ZHAW; to determine the sensation of 'cold' via conjugate heat transfer, a collaborative project with WINDTECH AS.

Synopsis Part 1: The International Journal of Multiphysics

H Khawaja
The International Society of Multiphysics

The International Journal of Multiphysics publishes peer-reviewed original research articles, review papers and communications in the broadly defined field of Multiphysics. The emphasis of this journal is on the theoretical development, numerical modelling and experimental investigations that underpin Multiphysics studies.
The scope of the journal is to address the latest advances in theoretical developments, numerical modelling and industrial applications which will promote the concept of simultaneous engineering. Typical combinations would involve a selection from subject disciplines such as Acoustics, Electrics, Explosives, Fire, Fluids, Magnetic, Soil, Structures, and Thermodynamics. This journal aims to publish high-quality findings of basic research and development as well as engineering applications.
The International Journal of Multiphysics is indexed in Elsevier® Scopus (SNIP, CiteScore, SJR), Elsevier® Engineering Village (EI Compendex), Clarivate Analytics® Emerging Sources Citation Index (ESCI), Clarivate Analytics® Web of Science, SHERPA RoMEO: Green, Directory of Open Access Journals. The International Journal of Multiphysics received Elsevier® Scopus CiteScore of 1.0 in 2020 (in comparison to 0.8 in 2019).

For more information, visit: www.multiphysics.org/journal

Synopsis Part 2: The International Conference of Multiphysics 2022

H Khawaja
The International Society of Multiphysics

The objective of The International Conference of Multiphysics is to share and explore findings on mathematical advances, numerical modelling and experimental validation of theoretical and practical systems in a wide range of applications.
The scope of the conference is to address the latest advances in theoretical developments, numerical modelling and industrial application which will promote the concept of simultaneous engineering. Typical combinations would involve a selection from subject disciplines such as Acoustics, Electrics, Explosives, Fire, Fluids, Magnetic, Nuclear, Soil, Structures and Thermodynamics.
In the past, Multiphysics Conferences have been organised Online (Virtual), Dubia United Arab Emirates, Krakow Poland, Beijing China, Zurich Switzerland, London United Kingdom, Sofia Bulgaria, Amsterdam The Netherlands, Lisbon Portugal, Barcelona Spain, Kumamoto Japan, Lille France, Narvik Norway, Manchester United Kingdom and Maribor Slovenia. Researchers from all around the world participated in these events. The Organisers and the Management Committee are thankful to all attendees for making these events successful.

For more information, visit: www.multiphysics.org/conference

SESSION 1.2

AEROSPACE AND MARINE

THURSDAY 9 DECEMBER 2021
11:30 – 13:00

CHAIR

G Boiger
Zurich University of Applied Sciences
Switzerland

A CFD Investigation on the Impact of Projectile Spin on Base Turbulence Energy and Aerodynamic Drag of a Supersonic Projectile

N Bagalkot, A Fallah, A Keprate
Oslo Metropolitan University, Oslo, Norway

High-speed computational fluid dynamic (CFD) study was carried out on a spinning 25-mm M910 training projectile. The study's primary objective was to understand how the spinning of the projectile influences the various aerodynamic coefficient and the turbulence energy dissipation at the base profile. ANSYS CFX was used to solve compressible, mass-averaged Navier-Stokes's equations using the k-ω and SST turbulence model. The simulation methodology was validated using the previous experimental and CFD simulation data. The simulations were carried out for a range of projectile spin, at different Mach numbers ranging from M = 0.8 to M = 3.5. Since the turbulence associated with the spinning of the projectile is transient, a transient CFD simulation was carried out. The results indicated that the spinning of the projectile is more significant to turbulence kinetic energy than aerodynamic coefficients.

Turbulence Energy, Aerodynamic Drag, Supersonic Projectile, Mach number

Coupled Aerodynamic and Magnetic Torques for Satellite Stabilization

M Azeem[1], R Varatharajoo[2], S Ahmad[1]
1. COMSATS University, Pakistan
2. University Putra Malaysia, Malaysia

For a Low-Earth-Orbit (LEO) satellite with nadir-pointing requirements, a three-axis attitude stabilisation is typically required. At altitudes below 500 km, satellites are primarily subjected to aerodynamic drag. As a result of the drag force, the disturbance torques will act around the satellite's centre of mass. When a satellite moves through the Earth's magnetic field, it experiences a magnetic disturbance due to its residual magnetic moment. Magnetic disturbance can be also used to preserve the satellite fuel. In addition, by exploiting the aerodynamic torques, the satellite's attitude can be stabilised. In this work, the magnetic disturbance torques and aerodynamic torques are coupled to stabilise the satellite's roll, pitch, and yaw attitudes. To employ the aerodynamic torques, the flaps must be placed on the satellite's body and slanted at specific angles to provide the necessary control torques. This combination approach is investigated using numerical simulations based on their developed governing equations. The results of the simulations demonstrate coupling the magnetic disturbances into the current satellite attitude stabilisation regimes are effective especially for small satellites.

Satellite Stabilization, Aerodynamic Torques, Magnetic Torques

Comparing Single Phase Flow Characteristics and Scavenge Performance of Aero-engine Bearing Chambers

B Chandra
University of the West of England, United Kingdom

A good scavenge performance in aero-engine bearing chamber is necessary to reduce the risk of oil leakage and ensure effective heat transfer. Research on scavenge flow has been conducted in the past employing experimental and computational efforts. Whilst some progress has been made in understanding the major factors affecting the scavenge performance, however it remains relatively expensive and laborious to assess a sump geometry experimentally or computationally. It was observed that at reasonably high shaft speed, the windage generated by the shaft rotation dominates, thus the rotating air drives both the liquid and gas phases. This paper presents a pioneering work to examine if there is a relationship between single phase air-only flow field in a bearing chamber with a given sump geometry and its corresponding scavenge performance. CFD simulations of single-phase air-only flow were performed on bearing chamber with several sump variants. The characteristics of the single-phase flow field including if any, the size and strength of secondary recirculation zones, were compared to the measured residence volume, and in some cases, previously observed severity of the hydraulic uplift of the liquid pooling in the sump region. It is proposed that this work can lead to an alternative method to easily assess future sump designs.

Aerospace, Gas Turbine, Bearing Chamber, Sump, Scavenge Flow

Implementing LiDAR for Sea Spray Flux Estimation for the Evaluation of Marine Icing on Ships in Cold Regions

S Dhar, K Edvardsen, H Khawaja, M Naseri
UiT - The Arctic University of Norway

Accretion of freezing sea spray formed by ship-wave interaction or by wind-generated is the primary contributor to ice accumulation on ships. It involves multiphase flows of air and sea spray, heat transfer, phase change from water droplets to ice, and finally, impingement and adhesion on ship structures; hence, it is a perfect example of a Multiphysics problem. The efficiency of the present ship-icing models to estimate the amount of ice accumulation is largely dependent on the accuracy of the input parameter of incoming spray flux. These models mostly use spray-flux formulations based on past empirical observations from fishing trawlers. The past attempts to measure sea spray flux were carried out by collecting impinging spray on specific parts of the total spray cloud, which doesn't provide information such as its complete flux distribution, its evolution, velocity, and frequency. In this study, the authors implement a measurement technique well-established in other disciplines in a field new to Lidars: studies of sea sprays. LiDAR is capable of visualizing the evolution of the sea spray drift with a high spatial and temporal resolution. In order to recognize this potential, a novel LiDAR prototype named MarSpray LiDAR (MSL) is designed, built, and tested. The MSL is a mono-static multi-axial 905nm wavelength LiDAR, specifically designed for short-range spray analysis. It generates laser pulses of short width (FWHM) of 3 to 4ns, with a spatial resolution able to highlight local variation concentrations on the order of 45 to 60cm inside the spray with a temporal resolution of 1ms. With certain future modifications of the MSL, the primary application for this remote sensing equipment can be made suitable for shipborne use to profile and retrieve marine spray properties capable of gathering real-time data and has the potential to bridge the gap that the past techniques failed to measure.

Sea Spray, LiDAR, Marine Icing

SESSION 1.3

COUPLING METHODS

THURSDAY 9 DECEMBER 2021
14:00 - 15:30

CHAIR

T Watanabe
National Fisheries University
Japan

A Parametric Study of Underground Fiberglass Pipeline Subjected to Fluid Induced Vibration Using Numerical Technique

A Alnahdi[1], E Al-Bahkali[1], M Souli[2], T AlBahkali[1], S Parvez[1]
1. Department of Mechanical Engineering, King Saud University, Saudi Arabia
2. Department Laboratoire de Mécanique, de Lille, Villeneuve d'Ascq, France

Fiberglass is a lightweight, corrosion-resistant, cost-effective alternative for concrete, steel and other plastic pipes, especially in large-diameter, high-pressure applications. Nowadays fiberglass pipes are being used in many fields and applications that are represented in transfer or storage a wide variety of fluids. They had replaced the steel pipes for the advantages of easiness of installation, lightweight, corrosion resistance, strength-weight ratio, low maintenance and easy to repair if damaged. Pipeline failures are numerous, one of these failures is due to vibration. The vibration is excited in the system due to the fluid flow. It is known as fluid induced vibration (FIV). This kind of problem is identified as fluid structure interaction (FSI). This study aims to investigate the optimal values of specific parameters (e.g., pipe diameter, fluid velocity and operating pressure) for a main water header fiberglass pipeline buried under ground using numerical technique. It is exhibit FIV with different vibration mechanisms based on its natural frequencies. The fluid is considered one-phase flow, with steady/unsteady condition.

Fiberglass, FIV, FIS, Natural Frequency

High Performance Computing Simulations for Multi-physics Fusion Phenomena

P Bonilla[1], M Mantsinen[1,2], J Lorenzo[1], O Ortega[1], E Goldberg[1], P Pastells[1], X Sáez[1], J Manyer[1], O Fernandez[1], G Houzeaux[1], O Lehmkuhl[1], S Gomez[1], H Calmet[1], A Soba[3,4], X Granados[5]

1. Barcelona Supercomputing Center (BSC), Spain
2. Catalan Institution for Research and Advanced Studies (ICREA), Spain
3. National Atomic Energy Commission (CNEA), Argentina
4. National Scientific and Technical Research Council (CONICET), Argentina
5. Institute of Materials Science of Barcelona (ICMAB), Spain

Nuclear fusion reactors are highly complex systems from a multi-physics and modelling perspective. The simulation of interdependent physics phenomena requires involved numerical models and costly computations in several time and space scales. Integration of these multi-physics models is key to understand how the different physics processes affect each other. Furthermore, only optimal efficiency of these coupled physic models enables whole domain simulations to validate the models and integrate them in the community tools. In this work, we present nuclear fusion applications based on Alya computational mechanics multi-physics HPC software (Vázquez, 2016). Alya's main advantages are its supercomputing resource optimisation and its in-built modular multi-physics implementation that overcomes the necessity of data converters which can significantly hamper a multi-physics simulation. This paper highlights the current state of the development of the main Alya modules applied to fusion, i.e., those of magnetism, fluid dynamics, thermodynamics and neutron transport. All the modules allow or are planned for future couplings between them within the Alya system. Regarding the test cases for validation of the modules, first the magnetism module developed to predict the performance of High Temperature Superconductors (HTS) cables in fusion magnets is compared against analytical models and numerical benchmarks in the literature. Then, the fluid dynamics and thermodynamics modules are tested on a simulation of the heat dissipation on ITER's first wall with a given source term to account for the nuclear deposition. Finally, the advances in the neutron transport module are validated against benchmarks from a Nuclear Energy Agency database SINBAD in which tests for simple geometries are developed (Soba, 2021).

Fusion, Magnetism, Fluid Dynamics, Thermodynamics, Neutron Transport; High Performance Computing

Simulation of Mechanical Relays under Coupled Electromagnetic and Structural Dynamic Physics

T Lankenau, M Graf
University of Applied Sciences Emden/Leer
Germany

The function of mechanical relays depends largely on the mechanics and electromagnetics. Both can be calculated separately using the finite element method. In this specific application the electromagnetic field, which is induced by a coil in the relay, generates forces in the component that lead to deformations in the geometry, which in turn influences the electromagnetic forces in the relay. The aim of this simulation is to couple both physics so that mechanics and electromagnetics can be calculated in a combined model. For this purpose, the properties for mechanical deformations and magnetic fields are simulated with the software "COMSOL Multiphysics". The electromagnetic forces are calculated in the electromagnetic model and coupled into the structure in the mechanical model using Maxwell's surface tension tensor. The relay is pretensioned by a spring force and is then operated by an electric current fed into the coil. In this way, a work cycle of the relay can be simulated. The calculation can be used to calculate electromagnetic forces that are necessary to trigger the relay. Structural dynamic calculations in the electromagnetic field are possible and the magnetic flux in the corresponding areas can be optimized. Furthermore, interfering stray fields in the course of deformation can be determined and the properties of the coil and the iron core can be optimized. By this, the function of a mechanical relay can be significantly improved.

Relay, Simulation, Electromagnetic-Structure Interaction, Maxwell's Surface Tension tensor

KaleidoSim - Massive Simultaneous Cloud Computing for Multiphysics Simulations

G Boiger[1], D Sharman[1,2], D Drew[2]
1. ZHAW Zurich University of Applied Sciences, Winterthur, Switzerland
2. Kaleidosim Technologies AG, Switzerland

The cloud-computing software KaleidoSim has been developed. While the current web-platform focuses on compatibility with the open source CFD toolbox OpenFOAM, KaleidoSim enables MSCC Massive Simultaneous Cloud Computing functionality for any Multiphysics simulation software. Thereby the idea is to provide an easy-to-use, web-browser-based software, which allows the user to run any Multiphysics simulation in the cloud within minutes. Rather than just replacing local hardware, KaleidoSim has been specifically designed to tackle the issue of horizontal scaling in the cloud. While vertical scaling (parallelization) can well be conducted according to a standard selection of cloud-computers with up to 224 cores each, KaleidoSim's key functionality is about the orchestration of multiple cloud-machines running simultaneously. KaleidoSim has been tested for workflows involving the seamless coordination of up to 500 individual, simultaneous simulation runs. Features include: i) The Self-Compile Option where OpenFOAM-based, self-composed software can be directly uploaded, compiled and run in the cloud; ii) The Katana File Downloader, which enables the selective download of any computed cloud data; iii) The In-Cloud Post-Processing Option for Paraview-based, automated post-processing via Python-trace utilities; iv) The Kaleidoscope Feature enabling the automated creation of large parameter studies; v) The API Application Programming Interface, for command-line-based access of the entire spectrum of KaleidoSim-functionality and for easy integration into third-party software. These capabilities allow the conduction of parameter studies, optimization runs and generally the creation of vast amounts of simulation data with unprecedented speed. This means that Kaleidosim effectively democratized vast computational resources and enables the integration of numerical studies of significantly higher data-intensity, investigative depth and thus quality, into every-day Multiphysics investigations. Application examples of several thermo-fluiddynamic scenarios can be demonstrated in which work-flow-speed-up-factors of no less than 50-100 can be achieved.

Cloud, MSCC Massive Simultaneous Cloud Computing, Horizontal Scaling, Browser-Based, Parameter Studies

SESSION 1.4

MULTIPHYSICS SIMULATIONS

THURSDAY 9 DECEMBER 2021
16:00 - 17:30

CHAIR

B Alzahabi
Kettering University
USA

Experimental Identification of the Gas Foil Bearings Operational Conditions - Study on Mechanical and Thermal Properties

J Roemer[1], P Zdziebko[1], A Martowicz[1], S Kantor[1], J Bryła[1], G Żywica[2], P Bagiński[2]
1. AGH University of Science and Technology, Krakow, Poland
2. Institute of Fluid-Flow Machinery. Polish Academy of Sciences, Poland

The paper demonstrates identification of temperature and strain fields of the gas foil bearing (GFB) support layer. The effectiveness of GFB was successfully verified for many demanding applications. Even now, however, the principle of operation in transient states (run-up/run-down) is not yet fully understood. It leads to the increased risk for malfunction, especially along the strict size and weight requirements and adverse working conditions. For this reason, the customized bearing with integrated temperature and strain gauges was presented. The feasibility of the solution was proved with an experimental test.

The authors acknowledge the project "Mechanisms of stability loss in high-speed foil bearings – modelling and experimental validation of thermomechanical couplings", no. OPUS 2017/27/B/ST8/01822 financed by the National Science Centre, Poland.

GFB, Gas Foil Bearings, Rotating Machinery, Mechanical Engineering, Inconel

The Lymphatic Pumping Mechanism: A Numerical Study

M Pepona, G Helbecque, B Kaoui
Biomechanics and Bioenginering Laboratory, Universite de Technologie de Compiegne, Compiègne, France

Elucidating the lymphatic pumping mechanism is of crucial significance towards a better understanding of the lymphatic system-related diseases, such as lymphedema and cancer, and subsequently the development of more efficient treatments. To this end, we are currently developing a numerical framework to computationally model the interaction between the lymph fluid, the deformable lymphatic vessel walls and the lymphatic two-leaflets valves. Our numerical model accounts for the reaction-diffusion of calcium ions (Ca2+) and the advection-diffusion of nitric oxide (NO) regulating the lymphangions contraction-dilatation. Our numerical approach consists of: the lattice Boltzmann method to solve the lymph fluid flow and species mass transport governing equations, a spring-network model for the lymphatic vessel walls and valves, and the immersed boundary method allowing for the coupling of the fluid and structure solvers. Results on two-dimensional configurations will be presented. The current implementation paves the way to the non-trivial extension to three dimensions, and the study of the effects of various fluid, geometrical and mechanical properties on the lymphatic pumping mechanism.

Computer simulations, Fluid Structure Interaction, Mechanobiology, Lymph pumping

Surrogate-based Modelling of Composite Panels with Microvascular Channels

S Upnere
Riga Technical University, Latvia

Bio-inspired vascular networks are widely used in engineering fields such as battery cooling, nanosatellite panels, self-healing and self-cooling materials. This study demonstrates a fast and computationally resource-efficient methodology for modelling microvascular channels in a composite material. Computational fluid dynamics simulations are performed to analyse the cooling efficiency of the panels using several two-dimensional channel network designs with different coolant flow rates at specific sampling points. The obtained results are used as input parameters to create surrogate models. Modelling confirms that microchannels can significantly reduce the maximum temperature of heated composite panels.

Coolant Flow, Heat Transfer, Microchannels, Surrogate Model

A Semi-Automated Multiphysics Simulation Software for Process Design in the Powder Coating Industry

S Bercan, G Boiger
Zurich University of Applied Sciences (ZHAW)
Winterthur, Switzerland

Powder coating is a widely used industrial process to obtain functional or aesthetic surface properties on manufactured parts. In a previous contribution we have introduced a Multiphysics methodology to simulate real life coating applications with a Eulerian-Lagrangian solver within the OpenFOAM framework, incorporating multiple periodically moving coating pistols. In the scope of the current study, the aforementioned methodology has been semi-automated to develop the software coatSim. The aim of the software is to provide a sequential workflow for setting up and simulating practical coating applications for all users, regardless of their expertise in the field of computational science. A database of pre-generated coating chambers and pistol geometries is at the disposal of the user. The database can be complemented by any substrate geometry uploaded. Then the user can interactively perform geometric operations via a combination of Paraview and Python libraries to manipulate STL files. An elaborate meshing scheme allows the user to automatically mesh each of the pistol regions and the coating chamber with the possibility of coarsening or refining the mesh through a single scaling parameter. As the material and process parameters are prescribed, the software performs the necessary operations to set up the case in the correct sequence. The finalized cases are uploaded to the cloud platform KaleidoSim through its extensive API (Application Programming Interface) capabilities in order to perform large series of simulations (e.g.: parametric studies, optimization runs) online. The coatSim software bridges the gap between the practical application and the theoretical expertise required to accurately simulate a coating process in OpenFOAM. Hence, CFD becomes a viable predictive tool for process design in the powder coating industry, as coatSim allows an intuitive workflow without in depth expertise in the field of CFD.

Powder coating, coatSim, Multiphysics simulation software, Cloud computing, KaleidoSim

SESSION 2.1

FLUID MECHANICS

FRIDAY 10 DECEMBER 2021
09:30 – 11:00

CHAIR

E Albahkali
King Saud University
KSA

Conservative Multirate Explicit Time Integration Method for CFD

R Messahel, G Grondin, J Gressier, J Bodart
ISAE-SUPAERO, Toulouse, France

In the context of high-fidelity simulation of compressible flows (LES and DNS) at extreme scale (small time steps) on massively parallel supercomputers, explicit time integration methods for which the Courant–Friedrichs–Lewy (CFL) condition has to be satisfied everywhere in the computational domain can lead to an unnecessarily expensive computational approach. We propose further improving the efficiency (CPU wall-time reduction) of explicit Runge–Kutta methods by developing a multirate explicit time integration method through flux interpolation at the boundary between cells evolving with different time-steps while enforcing the conservation properties of the legacy schemes. The presented mulltirate method is assessed both in terms of numerical accuracy and stability and of computational efficiency in a parallel framework (CPU wall-time reduction) on one-dimensional benchmarks (linear advection/shock tube) and the developed turbulent channel flow test case (Re_tau=392).

Multirate Explicit Time Integration, Finite Volume, Spectral-Differences, High Performance Computing, Direct Numerical Simulation

Phenomena of Liquid Nitrogen Flashing under Rapid Depressurizations

A Nakano[1], T Watanabe[1], K Kambara[1], K Tokunaga[1], T Shiigi[1], K Shimojima[2]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan

Flashing experiment of liquid nitrogen in a pressure vessel was conducted under rapid depressurization rate. Observations of the explosive boiling behaviour caused by quick opening of an electro-magnetic valve were undertaken by using a video camera. Pressure and temperature changes in the vessel were measured. Experimental results showed that the initial temperature distributions and depressurization rates have more intrinsic influence on these flashing phenomena. Relationship between pressures undershoot and depressurization rate was obtained. The rate of pressure recovery was found to increase with the rate of depressurization, which also resulted in the increase of the increment of pressure recovery.

Liquid Nitrogen, Flashing, Depressurization

Phenomena of Water Phase Change under Decompression State

K Kambara[1], T Watanabe[1], A Nakano[1], K Tokunaga[1], T Shiigi[1], K Shimojima[2]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan

The latent heat of the evaporation for water is very big compared with other fluid. Therefore, when evaporation is promoted by depressurization, water lowers the temperature and freezes finally. We made them freeze without applying heat by decompressing water using a vacuum pump. Even if water caused boiling, the pressure kept falling, but when freezing finally, pressure build-up was measured. It was expected that pressure build-up is caused by increase of the evaporation amount by the latent heat release when freezing. An experiment was made in detail. This is the transition phenomenon when switching over from evaporation to sublimation. This seems to bring useful information when considering a phase change model.

Water, Phase Change, Decompression State

Numerical Investigation of Incompressible Flow through Cavity attached in a Channel

R Kanna, O Alshaltouni, O Shaqra
Al Ghurair University, UAE

Flow over cavity is having many applications such as electronics cooling, heat exchangers, glass tempering are few. It involves fundamental flow physics such as flow separation, boundary layer, vortex formation and reattachment etc. Cavity may arise in various form like inclined cavity, T-shaped cavity, shallow cavity and many more based on the requirement. The present investigation to simulate incompressible fluid flow through a channel which drives fluid inside the cavity. An in-house CFD solver will be used to simulate two-dimensional situation to simulate laminar flow. Non-uniform computational domain will be used to capture where gradients are significant. The momentum transferred viz. fluid spread over the channel will shear the fluid in the cavity. The interesting flow physics will be revealed in connection with Reynolds number and aspect ratio. Results will be presented for velocity contours, vorticity contours and streamlines.

Channel Flow, Cavity, Numerical Simulation, Vortex, Velocity Contours

SESSION 2.2

MODELLING TECHNIQUES

FRIDAY 10 DECEMBER 2021
11:30 – 13:00

CHAIR

A Martowicz
AGH University of Science and Technology
Krakow, Poland

Comparison and Evaluation of Numerical Techniques and Physico-Chemical Algorithms for the Simulation of an Iodine Ion Thruster

I Barranco-Gomez, C Toomer
University of the West of England, United Kingdom

An electrostatic ion thruster, modelled on the BIT-3, is simulated numerically using the software "Starfish". It is assumed that the thruster is in vacuum conditions propelling a CubeSat in low Earth orbit. A model for using xenon propellant and a new model for iodine propellant are developed and tested. The plasma in the ion thruster and the associated electric fields are simulated using a particle-based kinetic code in which a hybrid approach of Particle in cell and Direct Simulation Monte Carlo methods has been used. In modelling these flows, elastic and inelastic collisions can occur and charge and momentum exchanges occur. Such collision models use a number of assumptions, e.g., concerning the collision cross-section, and in this paper we present results where the physico-chemical modelling is improved reducing the level of assumptions used. Results are also presented concerning the numerical methods used for the iterative convergence scheme, stochastic sampling and the importance of the constraints for the mesh size and timestep. The subsequent effects on the plume and the thrust produced of using different approaches in modelling the electron temperature distribution are also evaluated to produce a more rigorous modelling methodology.

Ion thruster, Iodine propellant, PIC, DSMC, collision models

Geometric Optimisation of Stepped Labyrinth Seals

B Chandra
University of the West of England, United Kingdom

Sealing technologies have been one of the most crucial parts of turbomachinery due to the operational working principle of gas turbines. There are at least fifty locations in a jet engine that utilises a seal therefore the collective influence of the seals has a massive contribution to the overall efficiency of the engine. There has been a great advancement in other major components within jet engines but labyrinth seal technologies have lagged. Even with the emergence of new types of seals, such as brush seals, labyrinth seals are still preferred at certain positions within the jet engine as the minimum clearance between the stationary and rotational components accommodates the differential thermal expansion while reducing undesired leakages. With the greater push for more efficient aircraft engines and the reduction of carbon emission contribution by the aviation industry, all components within the engines must be optimised to operate at their maximum efficiency. The work focuses on the geometric design optimisation of a two-dimensional labyrinth seal stepped seal design configuration. A numerical Computational Fluid Dynamics (CFD) study of a two-dimensional axisymmetric labyrinth seal to explore the effects of the geometrical parameters at operational conditions on the leakages in a jet engine was undertaken using ANSYS Design Explorer. A base geometry with associated experimental results was used to perform verification and validation of the simulation results was chosen from previous literature studies. Three geometric parameters namely, clearance, tooth width and step height were set as input parameters and mass-flow rate as the performance measure for the optimisation task. Latin-hypercube sampling Design of Experiments (DOE) was utilised to explore the design space with upper and lower limits set on the input parameters to constrain the size of the optimisation problem. An automated workflow is created to allow automatic generation of geometry using the parameters and using the set CFD boundary conditions to run the numerical calculation on each design. A goal-driven Multi-Objective Genetic Algorithm (MOGA) was used to optimise the DOE without constraints on parameters to avoid over constraining the optimiser. The main conclusion is that the input geometric parameters interaction requires a trade-off for the most optimum design in regards to minimum leakages each design allows. The clearance was found to have the greatest influence on the leakage output. The results showed the step height is inversely proportional to the mass flow. Design optimisation techniques that make use of data-driven methods and numerical techniques can allow designers broader design space search at much shorter lead times.

Aerospace, Gas Turbine, Seal, Labyrinth Seal

Multiphysics Analysis of Ice-Polyurethane Adhesion under Flexural Loading using FEM Analysis

H Eidesen[1], Z Andleeb[1], H Khawaja[1], M Moatamedi[2,3]
1. UiT-The Arctic University of Norway, Tromsø, Norway
2. Al Ghurair University, UAE
3. Oslo Metropolitan University, Oslo, Norway

This paper presents the Finite Element Analysis (FEA) Multiphysics technique, applied to study the strength of ice adhesion between the surface of polyurethane and ice. The theoretical study of this work is based on the Euler-Bernoulli beam theory that is used to solve a four-point bending problem to give the correlation of displacements with load and longitudinal stresses. The physical samples were prepared by freezing ice over the polyurethane surface and were tested experimentally in a four-point flexural setup. In the experiment, masses were added to the four-point bench until the ice separates from the surface. The results revealed that the ice adhesion on the surface of polyurethane is in the same range as with other polymers. The displacement at the time of separation was recorded, and the same conditions were used to perform numerical simulations in ANSYS® Workbench. The meshed ice-Polyurethane Finite Element Method (FEM) model was tested for sensitivity. A good agreement was found between theoretical, experimental, and numerical simulation results.

Ice-Polyurethane adhesion; Four-Point Flexural Test; FEA; Numerical Simulations; FEM, Multiphysics

Dynamic Response of Monolithic Ductile Membranes Subject to Underwater Shockwave

N Mehreganian[1], Y Novozhilov[2], P Ryabov[2], D Mikhaluk[2], A Fallah[3]
1. University of Liverpool, United Kingdom
2. CADFEM, Russian Federation
3. Oslo Metropolitan University, Oslo, Norway

Extensive shocks from underwater explosions induce catastrophic effects on the hull of the marine and submarine vessels or as well as offshore buildings. As a result of the initial shock and the subsequent bubble pulsation, a submerged underwater target is subjected to repeated harmonic loading, which can exhaust the target's energy absorption capacity to the point of failure and damage. The present work encompasses the development of analytical and numerical modelling schemes to study the nonlinear dynamic response of a rectangular membrane subjected to an extensive underwater pressure wave (UNDEX) and the subsequent pulsating bubbles. The target membrane is assumed to be monolithic, isotropic, homogenous, initially flat, and ductile. A direct consequence of the excessive pressure generated due to UNDEX on a lightweight membrane is the significant contribution of the plastic response to the total response compared to that of the elastic one. Hence, the material response is idealised as rigid-perfectly plastic. However, since the membrane stresses substantially influence the response, the influence of finite displacements is retained in the study. A pulse pressure load is assumed and truncated into a single term of its multiplicative representation of the spatial and temporal functions. The profile of the former is assumed as uniform across the target while that of the latter mimics a sequential composition of a series of exponentially decaying pulses which damp out gradually with the motion of the bubble and its disintegration. Considering the Fluid-Structure Interaction (FSI) effects, the analytical solutions are derived by assuming a kinematically admissible velocity profile in the three phases of motion corresponding to the pulse shape of the load. The analytical models corroborated well against the numerical methods which considered the same FSI effects.

Bubble pulsation, UNDerwater Explosion (UNDEX), rigid-perfectly plastic, membrane, FSI, shock wave

SESSION 2.3

ENVIONMENTAL SCIENCE

FRIDAY 10 DECEMBER 2021
14:00 – 15:30

CHAIR

R Kanna
Al Ghurair University
UAE

A Key for Reliable Simulation of Horizontal Axis Wind Turbine

M AbdulRaouf[1], E Al-Bahkali[1], S Parvez[1], M Souli[2]
1. Department of Mechanical Engineering, King Saud University, Saudi Arabia
2. Department Laboratoire de Mécanique, de Lille, Villeneuve d'Ascq, France

A complete 3D rotor model was designed using periodicity and Multiple Reference Frame (MRF) approach solutions to full Computational fluid dynamics (CFD) that is utilizing the Reynolds-averaged Navier–Stokes equations (RANS). The effect of the hub and tower of the wind turbine in simulation results were studied to reduce the computational time cost of wind turbine simulation. Simulations of a full-scale design of a horizontal axis wind turbine are first done and then compared to the wind tunnel experimental results using different methods. The results of these methods are compared to identify the best model that gives the least simulation time with the closest to the experimental results. Then, the simulation of the verified model was compared with the same design but without hub and tower to measure the differences in results in the two cases. Furthermore, the ratio of the two simulation times was identified to define which case takes less time.

Turbines, CFD, RANS, Wind, Tunnel

Phase Change Material, Doped by Nanomaterials for Thermal Energy Storage and Solar Cooling: Synthesis and Characterization

L Maifi, A Kamel, H Ouided, C Abdelhamid, K Tahar
Semiconductors Technology for Energetic Research Center (CRTSE)
Algeria

Among the parameters that limit the performances of photovoltaic panels when used in Saharan environments is the latent temperature under illumination, which destroys the diffusion length of the porters in the semiconductor. One solution to this problem is to use phase change materials to remove heat from the panel to keep it at the correct temperature. In this work we use a paraffin type $C_{25}H_{52}$ as phase change material. We are studying the evolution of the phase change time as a function of the mass of pure paraffin. To improve this time, we have doped this paraffin using TiO_2, $ALCL_3$ and AL_2O_3 as dopants. The study then consists of following the evolution of the phase change time depending on the nature of the dopant and the doping rate. The latter has shown that the best dopant is aluminium chloride type $ALCL_3$.

Paraffin, Dopant, Time, Temperature, Phase Change, TiO2, ALCL3, AL2O3

Windtech Device a Measurement Tool for 'Cold' Sensation

A Leyli, H Khawaja, S Antonsen, D Swart
UiT - The Arctic University of Norway

The operation conditions in the harsh cold climate bring several challenges for the workforce; therefore, a tool that can give an indication of how the people working in these conditions shall implement safety measures is of importance. The Windtech team has devised a novel tool for measuring the sensation of 'cold'. The 'cold' is measured via the response of an electrical heater attached to a temperature sensor. The heater is exposed to different weather conditions, and it will be stabilized within a short period of time which is referred to as 'heated temperature'. The higher values of 'heated temperature' indicate lower heat loss, and less values mean higher heat loss. The heat flow over the device's heater is simulated with ANSYS®. Also, the heat flow is modelled experimentally and analysed by IR thermography. The 'heated temperature' values are affected by environmental parameters, including ambient temperature, humidity, wind velocity, and irradiance. Windtech device is equipped with sensors that measure associated parameters. The measured values of 'heated temperature' are further implemented in combination with ambient temperature to provide values referred to as 'representative temperature'. The 'representative temperature' values are used to determine risk classification based on ISO standards.

Cold climate, Heat loss, CFD, CHT, Infrared thermography, Risk

An Optimal Location of Cooling System Design for a Hybrid Photovoltaic Thermal System

A Abdulbari[1], E Al-Bahkali[1], S Parvez[1], M Souli[2]
1. Department of Mechanical Engineering, King Saud University, Saudi Arabia
2. Laboratoire de Mécanique, de Lille, Villeneuve d'Ascq, France

In this study, a cooling technique for a hybrid solar Photovoltaic/Thermal (PV/T) system has been modelled and simulated via Computer-Aided Engineering (CAE) software in order to cool and maintain the temperature of the surface of the PV panel at a certain allowable range by optimizing the elevation, water flow rate and temperature distribution across the panel. Multiple analytical tests have been run on the proposed cooling technique to target operational aspects in order to determine the optimum cooling layout. These aspects are PV module elevation, pipes shape, pipes configuration, water flow rate, temperature distribution on the surface of the PV panel, and finally inlet and outlet temperature within a certain controlled volume of the system. Optimum pipes configuration was determined; then different elevations were considered to conduct the comparison. The increase in the efficiency and the amount of saved power resulted from applying the selected cooling technique were estimated in two different periods of the year depending on previous studies' readings. The output power was also compared with the readings of a not-cooled PV module to observe the technique's influence on the system.

Photovoltaic, Thermal, FIS, Power, Cooling

SESSION 2.3

POSTERS

FRIDAY 10 DECEMBER 2021
16:00 – 17:30

CHAIR

A Fallah
Oslo Metropolitan University
Norway

Cold Heat Shock Loading to Parasite using the Liquid Nitrogen

T Watanabe[1], M Nakamura[1], K Tokunaga[1], T Shiigi[1], K Shimojima[2]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan

In Japan where people like raw food of the marine products, a problem of the parasite often occurs. A similar problem occurs for feed of the dolphin that is very popular in aquarium. A parasite is contained in marine disposal of waste and feed, the method of the management is a subject. There is a guideline of the Ministry of health, labour and welfare in Japan. But the information that is detailed to the parasite isn't known well. We carried out the research on the actual condition of the parasite. We tried to check about the low temperature tolerance of the parasite for usual freezing and low heat shock by liquid nitrogen experimentally.

Cold Heat Shock, Liquid Nitrogen, Parasite

Implementation and Validation of a Thermal Radiation in a 5 axis CNC Aeroplane Milling Machine using COMSOL Multiphysics

P Kalsariya, D Schnee, A Shah, T Schiepp
Hochschule Furtwangen University, Germany

The aim of this paper is to carry out thermal radiation for a 5 axis CNC aeroplane milling machine using COMSOL Multiphysics. The paper aims to explain the process flow of the simulation from choosing the initial values till the thermal radiation results post simulation. A validation of the simulation results was carried out by checking the temperature results got by sensors that were implanted in the machine in real time.

Thermal radiation, FEMM

Mass Transfer from Multiple Core-shell Cylinders subjected to Flow: Towards Modeling Artificial Lung-on-a-Chip

P. Lammers, C. Bielinski, A. Bou Orm, B. Kaoui
Biomechanics and Bioengineering, Universite de Technologie de Compiegne, Compiegne, France

We study systematically mass transfer from a stationary array of cylinders covered with a semi-permeable shell (that adds an interfacial resistance) and subjected to flow. Two-dimensional computer simulations based on two-component lattice Boltzmann method are used to compute the flow around the cylinders and the mass transport of a solute. Mass transfer efficiency is characterized with the Sherwood number (the dimensionless transfer coefficient) as a function of different parameters, such as the shell permeability and the cylinders spatial arrangement. The model and the knowledge of this study can be applied to model oxygen exchange in an array of cylinders in artificial lungs.

Computer simulations, Transport phenomena, Organ-on-chip, Artificial lung

Numerical Simulation of Thermoacoustic Oscillation for Nuclear Application

T Watanabe
University of Fukui, Japan

High-speed computational fluid dynamic (CFD) study was carried out on a spinning 25-mm M910 training projectile. The study's primary objective was to understand how the spinning of the projectile influences the various aerodynamic coefficient and the turbulence energy dissipation at the base profile. ANSYS CFX was used to solve compressible, mass-averaged Navier-Stokes's equations using the k-ω and SST turbulence model. The simulation methodology was validated using the previous experimental and CFD simulation data. The simulations were carried out for a range of projectile spin, at different Mach numbers ranging from M = 0.8 to M = 3.5. Since the turbulence associated with the spinning of the projectile is transient, a transient CFD simulation was carried out. The results indicated that the spinning of the projectile is more significant to turbulence kinetic energy than aerodynamic coefficients.

Turbulence Energy, Aerodynamic Drag, Supersonic Projectile, Mach number

Numerical Study of Viscous Fingering in a Heterogeneous Porous Medium

H Djebouri, S Zouaoui
L.M.S.E. Laboratory, Mechanical Engineering Department, University of Tizi Ouzou, Tizi-Ouzou

This work deals with the numerical study of an immiscible water-oil displacement through a porous medium. This type of flow finds its application in many industrial processes [1]. The purpose of this work is to see the effect of the heterogeneity of the porous medium on the instability of the interface of the two fluids. This instability develops by the formation of viscous fingering at the interface [2]. This phenomenon is undesirable in the enhanced oil recovery (EOR) because it reduces the sweep efficiency [1, 3]. Faced with a double complexity, that of the nature of the porous medium and that of the nature of the flow, most of the researchers concentrated on simple geometries and on the qualitative aspect of the phenomenon [1]. In this work, we consider a five-point configuration with an injection well located at the centre and four injection wells at the corners of a square. The qualitative aspect has been given to this phenomenon. In order to study the effect of the heterogeneity of the medium, different cases are considered. The case of a fractured porous medium was also taken into account. The evolution of the water-oil interface is shown in the saturation fields and the number of viscous fingers is represented as a function of time.

Multiphase Flow, EOR, Heterogeneity, Porous Medium, Viscous Fingering

Shock Loading Pre-processing for Freeze-drying

T Watanabe[1], M Nakamura[1], K Shimojima[2], S Tanaka[3], K Hokamoto[3], S Itoh[4]
1. National Fisheries University, Japan
2. Okinawa National College of Technology, Japan
3. Institute of Industrial Nanomaterials, Kumamoto University, Japan
4. Institute of Shockwave Applied Technology, Japan

In the food industry, it is hoping for high value-aided product and the increase in efficiency of food processing. On the other hand, we get an experimental result that the load of the shock wave improves an extraction of food and soften food. We tried to examine the effectivity of the shock wave as pre-processing for freeze-drying from the result in permeation character seen in the radish and so on. In the case of freeze-drying, the object tends to be limited to the small or thin one with size, from the sublimability in processing, the performance in case of the restoration and the viewpoint of the cost performance ratio. Therefore, we used comparatively large, beheaded shrimps and squids and attempted to review the effectivity of the shock wave processing about being freeze- drying. The improvement of the sublimation speed was gotten from the result that the pressure change during freeze-drying processing and the improvement of the reconstitution was gotten from the result using hot water. It was expected that the reconstitution of the freeze-dried food is improved and that a processing time is abridged, by shock wave loading as pre-processing for freeze-drying. This phenomenon will be modelled, and it will be compared with a result of an experiment.

Shock Loading, Pre-processing, Freeze-drying

www.elsevier.com

www.aau.ac.uk

www.oslomet.no

www.zhaw.ch

www.agu.ac.ae

www.kaleidosim.com

www.windtech.no

www.nafems.org

www.uit.no

MULTIPHYSICS®

www.multiphysics.org