Swansea University
Prifysgol Abertawe

UK Fluids Conference 2024

Swansea University
Bay Campus
9-11th September 2024

UK Fluids Conference 9th-11th September 2024

Table of Contents

Link to Conference

For comprehensive information about the UK Fluids Conference 2024, including the detailed conference program, profiles of keynote speakers, venue details, accommodation options, and information about the organizing committee, please visit our official website at https://www.swansea.ac.uk/science-and-engineering/research/upcoming-research-events/ukfc2024/. This page serves as the central hub for all event-related updates and resources.

Conference Venue

The conference will be held at the state-of-the-art Engineering Central building, located on the Bay Campus of Swansea University. The venue is situated on Fabian Way, offering a scenic view of Swansea, Wales. The full address is Engineering Central, Bay Campus, Swansea University, Fabian Way, Swansea, Wales, UK, SA1 8EN. The oral sessions will take place in rooms B001, B003, and B004. Posters will be displayed in room B002, where lunches and refreshments will also be served.

UK Fluids Conference Series

The UK Fluids Conference series is a premier annual event that brings together researchers, academics, and industry professionals from across the field of fluid dynamics. Established as a leading forum for the exchange of cutting-edge ideas and advancements, the conference covers a wide range of topics, including theoretical, experimental, and computational fluid mechanics. Each year, the event features keynote presentations by prominent experts, in-depth technical sessions, and opportunities for networking and collaboration. The series has grown in prominence over the years, attracting a diverse international audience and contributing significantly to the global discourse on fluid dynamics.

The UK Fluids Conference have previously taken place in London (ICL, 2016), Leeds (2017), Manchester (2018), Cambridge (2019), Southampton (online, 2021), Sheffield (2022), and Glasgow (2023). This year, the UK Fluids Conference will be held at Swansea University, during 9-11th September 2024.

The UK Fluids Conference 2024 is honored to feature an impressive lineup of keynote speakers, each a distinguished leader in their field. Dr. Marco Fossati, a Senior Lecturer at the Department of Mechanical and Aerospace Engineering at the University of Strathclyde, will share insights from his extensive research. Prof. Tiina Roose, who holds a Chair of Biological and Environmental Modelling at the University of Southampton, will bring her expertise in engineering and environmental systems to the forefront. Dr. Chris Combs, the Dee Howard Memorial Endowed Faculty Fellow in Mechanical Engineering and Graduate Advisor of Record at the University of Texas at San Antonio, will present on advancements in aerospace engineering. Richard Varvill, the Chief Designer and Technical Director at Reaction Engines Ltd, will discuss cutting-edge developments in propulsion technology. Prof. Maïwenn Kersaudy-Kerhoas, a leading Professor in Microfluidic Engineering at Heriot-Watt University, will delve into innovations in microfluidics. Finally, Dr. Douglas Pender, Technical Director at JBA Consulting and Service Lead for Marine and Coastal Risk Management, will share his expertise in coastal numerical modeling.

Prizes

Prizes at the conference are sponsored by the Journal of Fluid Mechanics (Cambridge University Press) and are available for students only. There will be two prizes for the best student presentation and the best student poster. It will be possible for all delegates to nominate students for these awards using the conference app

Instructions for Session Chairs & Presenters for Parallel Sessions

Each talk is scheduled for a 15-minute slot, consisting of 12 minutes for the presentation, 2 minutes for questions, and 1 minute for changeover. Session chairs are responsible for keeping the sessions on time. Their duties include providing a brief introduction to each talk, inviting and moderating questions, and ensuring the next speaker is set up. A synchronized clock will be available in each room, displaying the time allocated for presenting (12 minutes), Q&A (2 minutes), and changeover (1 minute). When the clock indicates Q&A time, chairs should ask the presenter to move to their final slide and conclude the talk. Chairs should arrive 5-10 minutes before the session begins to meet with the session assistant and speakers. If a speaker is absent, leave a gap in the schedule so that the subsequent talks start at the times listed in the program. Chairs are also required to complete the Student Prize Form at the end of their sessions.

Instructions for Keynote Session Speakers and Chairs

Each keynote talk is allocated 45 minutes, with speakers requested to present for no more than 35 minutes, leaving 10 minutes for questions. The chair will introduce the speaker and facilitate the Q&A session.

Instructions for Poster Sessions

Your poster has been assigned to one of the two poster sessions and can remain on display until the end of the conference. Please ensure you are present at your poster throughout your session and prepared to give a brief overview to visitors. Your poster should be in A0 Portrait format.

Conference Organising Committee

Prof Ben Evans	Swansea University
Prof Ruben Sevilla	Swansea University
Dr Xi Zou	Swansea University
Prof Oubay Hassan	Swansea University
Dr Alper Celik	Swansea University
Prof Antonio Gil	Swansea University
Prof Hamed Haddad-Khodaparast	Swansea University
Dr Bjornar Sandnes	Swansea University
Dr Jennifer Thompson	Swansea University
Dr Marco Fossati	University of Strathclyde
Dr Jason Jones	Swansea University
Dr Andrew Ross	University of Leeds

Conference Dinner

The dinner will be held on Tuesday, September 10th, at the Great Hall on Bay Campus, beginning at 7:30 PM.

Prof. Tiina Roose - University of Southampton

Prof. Roose holds a Chair of Biological and Environmental Modelling at the School of Engineering, Faculty of Engineering and Physical Sciences (FEPS) at the University of Southampton, where she also serves as the Deputy Head of School for Research. Additionally, she is a member of the Institute of Life Sciences (IFLS), which coordinates cross-faculty life science initiatives within the university, and one of the coordinators of Crop Systems Engineering at IFLS. With a Researcher ID of A-6352-2010, she is a dedicated scientist who combines mathematical modeling and experiments to explore the development and function of biological branching structures, such as plant roots, blood and lymph vessels, and the lung. As an academic mentor and supervisor, she takes pride in actively supporting her students' and postdocs' careers, helping them to publish and communicate their work to broad academic audiences.

Imaging and Image Based Modelling of Plant-Soil Interaction

We rely on soil to support the crops on which we depend. Less obviously we also rely on soil for a host of 'free services' from which we benefit. For example, soil buffers the hydrological system greatly reducing the risk of flooding after heavy rain; soil contains very large quantities of carbon, which would otherwise be released into the atmosphere where it would contribute to climate change. Given its importance it is not surprising that soil, especially its interaction with plant roots, has been a focus of many researchers. However the complex and opaque nature of soil has always made it a difficult medium to study.

In this talk I will show how we can build a state of the art image based model of the physical and chemical properties of soil and soil-root interactions, i.e., a quantitative, model of the rhizosphere based on fundamental scientific laws.

Needless to say, all fluid flows in soil are at low Re number.

Richard Varvill - Reaction Engines Ltd

Richard Varvill is a British engineer, and the Chief Designer (and Technical Director) at Reaction Engines Limited. He started his career with Rolls-Royce Military Engine Division, in the Advanced Projects division. He worked on preliminary ideas for what could have become the RB545 air-breathing rocket engine for HOTOL. He co-founded Reaction Engines in 1989 and is working on the successor to the RB545, SABRE (rocket engine).

Fluid dynamic challenges of winged reusable launch vehicles employing Sabre airbreathing propulsion

This talk will briefly summarise the design of the Sabre engine and the reusable spaceplane it is designed to propel (Skylon). It will then touch on the main fluid dynamics challenges in the engine and vehicle which will need solving to successfully realise an operational system.

Dr. Chris Combs - The University of Texas at San Antonio

Dr. Combs is the Dee Howard Endowed Associate Professor in Aerodynamics in the UTSA Department of Mechanical Engineering, where he is the Director of the Aerospace Engineering program, Director of the Center for Advanced Measurements in Extreme Environments, Director of the UTSA Hypersonics Lab, and Associate Dean Fellow for Research in the Klesse College of Engineering & Integrated Design. In the UTSA Hypersonics Lab, Combs leads a research group of over 20 graduate and undergraduate students studying problems related to hypersonic aerothermodynamics. He holds a BS degree in Mechanical Engineering from the University of Evansville and a Ph.D. in Aerospace Engineering from The University of Texas at Austin where he was a NASA Space Technology Research Fellow. His primary area of research interest is in the development and application of non-intrusive laser-based measurement techniques for compressible flows and he has extensive experience in investigations of hypersonic flow physics, with over 50 technical publications in this field and over $16M in research funding from various organizations including NASA, USAF, NSF, US Navy, JHTO, and DARPA. Recognized as a recipient of the prestigious National Science Foundation CAREER and Air Force Office of Scientific Research Young Investigator Program awards, Dr. Combs led the design and construction of the new Mach 7 wind tunnel facility at UTSA. He is a senior member of the American Institute of Aeronautics and Astronautics and is also active with the American Society of Mechanical Engineers and the American Physical Society. Dr. Combs serves as a board member for the Dee Howard Foundation and is also a member of a variety of technical and service committees including the AIAA Aerodynamic Measurement Technology Technical Committee, the San Antonio Chamber of Commerce Aerospace Committee, and the Dee Howard Foundation Education Advisory Council.

Development of Optical Diagnostics for Hypersonics Research at UTSA

This presentation will provide an introduction to the new hypersonic experimental research capability at UTSA with a particular focus on hypersonic aerodynamics and non-intrusive diagnostic development. Completed in 2021, the UTSA Mach 7 Ludwieg tube is a unique facility within American universities—one of only six at a U.S. academic institution operating at a Mach number of 7 or higher and one of only two with the current combination of high-Reynolds number (up to 200×106 m-1) and high-Mach number capability. The added availability of high-speed diagnostics equipment at UTSA makes the experimental test capability exceptionally rare. This talk will provide an overview of the many diagnostic capabilities currently employed and under-development, while providing insight towards the relevant fluid physics challenges currently being explored.

Prof Maïwenn Kersaudy-Kerhoas - Heriot Watt University

Maïwenn Kersaudy-Kerhoas is a Professor in Microfluidic Engineering in the School of Engineering and Physical Sciences at Heriot-Watt University. She is the co-academic lead for the Global Research Institute in Health & Care Technologies with Prof Robert Thomson.
She holds a research MSc degree in micro and nanotechnologies from the Technical University of Lille (France), and an MSc degree from the Institut Superieur de l'Electronique et du Numerique (Brest-Lille, France). She was awarded a PhD at Heriot-Watt in July 2010. This study involved the development of a microfluidic chip for blood plasma extraction for detection of cell-free nucleic acids (DNA, RNA). In 2012 she was awarded a five year Royal Academy of Engineering Fellowship to develop microfluidic systems for prenatal diagnostics. In 2014 she was elected as a member of EPSRC Early Career forum in Manufacturing Research and entered the Young Academy of Scotland. In 2018 she was awarded a £1M EPSRC Healthcare Technology Challenge Award to develop total pre-analytical systems for cfDNA extraction at the point-of-need. Using this award and pre-commercialisation funding from Scottish Enterprise High Growth Spin Out programme she created with her team CNASafe, and automated platform and patented fluidic cartridge for cfDNA extraction. This technology is now under clinical pilot for use in an integrated workflow (iSEP-SEQ) for the identification of sepsis-causing pathogens.

Experimental microfluidic blood manipulations: exploring opportunities, limitations and biomedical applications

Pre-analytical manipulation or the transformation of raw blood samples prior to their analysis, remains an important challenge for point-of-care devices. Microfluidic technologies offer opportunities for integrated and contained manipulation of blood in continuous or static mode. My lab has designed and tested microfluidic devices leveraging natural physical laws at the microscale for separating plasma from blood, and devices using integrated actuators to manipulate blood in static chambers. Using these examples, I will highlight the need to push current limits in terms of modelling to design the next generation of microfluidic devices.

Dr. Marco Fossati - University of Strathclyde

Dr. Fossati is Senior Lecturer at the Department of Mechanical and Aerospace Engineering of the University of Strathclyde. His research interests are in the area of multiphysics computational aerodynamics. His expertise is in the field of high-speed and non-equilibrium flows, modal-based Reduced Order Modeling for aerodynamics, mesh optimization and generation. At Strathclyde he is a member of the Aerospace Centre of Excellence (ACE) and director of the Future Air-Space Transportation Technologies (FASTT). Prior to joining Strathclyde Dr. Fossati was Associate Director of the CFD Laboratory at McGill University in Canada where he holds an Adjunct Professor position. He has been visiting researcher at the Hong Kong University of Science and Technology. He holds a PhD in Computational Fluid Dynamics from Politecnico di Milano. Dr. Fossati is a member of the editorial board of Computers and Fluids and serves as a reviewer for the Journal of Computational Physics, Physics of Fluids, AIAA Journal, International Journal of Multiphase Flows, Aerospace Science and Technology, Mathematics and Computers in Simulation and the International Journal of Computational Fluid Dynamics.

Thermal non-equilibrium effects in shock interaction patterns

Understanding and predicting shock interaction patterns is fundamental to be able to design and safely operate hypersonic vehicles. This work focusses in particular on the shock interaction patterns generated by hypersonic flows over double wedge geometries and aims at understanding the impact of non-equilibrium and relaxation processes on the flow physics and waves generated at the interaction point and on surface quantities such as pressure and heat flux. The study will provide insight into the non equilibrium physics by comparing different Aerothermodynamics models, from perfect ideal gas to multi-temperature.

Dr. Douglas Pender - Technical Director at JBA Consulting

Douglas is a Technical Director at JBA Consulting and Service Lead for Marine and Coastal Risk Management and is responsible for coordinating coastal numerical modelling.
Prior to working in consultancy Douglas has spent time in academic research roles working on combined statistical and numerical modelling of coastal and fluvial hydrodynamics.

Environmental Fluids: An Industry Perspective on Advanced Numerical Model Evolution
JBA Consulting has been modelling fluids in the natural environment for over 25 years. Over this time, our use of numerical models as decision-making tools has evolved, driven by the need for quality, industry change, and external challenges. This lecture will explore this evolution in river and coastal engineering through case studies that showcase key shifts in perspective and the future opportunities these changes have created.

Paper ID AC-1 **Programme ID** 93

A preliminary study on novel low noise UAV Propellers

Muhammad-Nuramirul-Hijjaz, Barnaby M. Cooper, Jan W. Modrzynski and Alper Celik

This study concerns with exploring the flow structures around novel blade concepts for unmanned aerial vehicles. A total of three propellers were designed, inspired by some of the recent studies done in the same field, namely the Negative Staggered Loop Propeller, Tandem Loop Propeller and finally the Baseline Propeller, which served as benchmark for performance comparison. The flow field around the propellers were investigated through Particle Image Velocimetry measurements and smoke flow visualisation. The experiments were performed for rotational velocities ranging from 1000 RPM to 5000 RPM and inflow velocities ranging from 0 m/s to 4 m/s. The time averaged velocity field results show the evidence of formation of streamtube and slipstream but no significant difference towards the velocity increment behind the propeller disc in comparison for all of the propellers. Furthermore, smoke flow visualisations reveal that the vortex structures at the wake of Negative Staggered Loop Propeller and Tandem Loop Propeller portrayed a more structured and less chaotic tip vortex as compared to Baseline Propeller . However, the vorticity value marked by the 1st tip vortex shedded from the tip of the Negative Staggered Loop Propeller is slightly lower than the Tandem Loop Propeller. A comparative noise measurement for all propellers was performed in the closed test section. Although the data is contaminated with reverbations and shear layer effects, the results may be used to have an initial idea for further studies to be performed in anechoic rooms. The noise characteristics were expressed as the Power Spectral Density of the pressure fluctuations measured at the wind tunnel wall. The results show a reduction of broadband noise of about 2-7 db/Hz at low to mid frequency range for the Negative Staggered Loop Propeller when compared to Baseline Propeller and a reduction of about 1-9 db/Hz as compared to Tandem Loop Propeller even though utilizing the same looped tip concept mainly due to lack in vertical wake separation causing Tandem Loop Propeller to appear louder as opposed to the other propellers tested. However, increase in Tonal Noise at the 1st Blade Passing Frequency by about 17-22 Db/Hz could be observed for Negative Staggered Loop case and Tandem Loop case compared to the Baseline case mainly due to the lack of skew on the blade as only straight blade was used to form the loop propellers.

Paper ID AC-2 **Programme ID** 110

Unveiling the Potential of Nanoscale Vibrations for De-Icing

Saikat Datta and Rohit Pillai

Undesirable ice accumulation on aeroplane wings is a pilot's worst nightmare; icing can increase the risk of dangerous aerodynamic stalls affecting an aircraft's aerodynamic performance. Uneven icing on the rotating components of the engine produces stress on the engine mounts and blades, leading to possible failure. The unpredictable shedding of the large ice chunks inside the engine can also cause mechanical impact damage and decrease engine performance. Existing de-icing methods depend on interventions like i) use of thermal systems, ii) mechanical separation, and iii) chemical applications, each with drawbacks such as high-power requirements, inefficiency, potential structural damage, and environmental concerns, respectively. An innovative approach involves leveraging ultrahigh-frequency surface vibrations, specifically using surface acoustic wave (SAW) devices. Our recent study indicates that nanoscale vibrations generated by SAW devices can induce significant acoustothermal energy, rapidly heating ice layers within a few nanometres of the surface, all within one billionth of a second. This localized heating can prevent the agglomeration of pre-critical ice crystals, leading to reduced ice adhesion and facilitating the removal of formed ice by gravity or natural forces. Our results reveal that different regimes of mechanism for the vibration-driven melting are present depending on the amplitude and frequency of vibration. While acoustothermal heating is predominant at lower amplitudes and higher frequencies of vibration, as the amplitude increases, the vibration causes a change of thermodynamic states at the vicinity of the surface-ice interface; this, in turn, produces rapid pressure-driven (i.e. non-acoustothermal) melting and dislodging of the ice layer. As SAW devices are costeffective, easily mass-produced, and energy-efficient, our results highlight their potential as a promising alternative to conventional thermal systems for anti-icing applications.

Paper ID AE-1 **Programme ID** 8

Impacts of truncation and flight altitude on the flow and thrust of a plug nozzle

Henry Lizcano and Manuel Del Jesús Martínez

Nozzles are devices that allow the mechanical energy of expanding gases to be harnessed. One type of nozzle is the plug or aerospike, whose performance at low flight altitude is better compared to the other nozzle types, in terms of thrust. If the Plug nozzle could maintain its thrust while rising through the atmosphere, i.e., while the atmospheric pressure is decreasing, it would be a different alternative to the traditional bell nozzle. As the plug nozzle ascends through the atmosphere, several phenomena affect flow behavior, resulting in thrust losses. In the present work, a study was carried out on the effects of flight altitude and truncation in a plug nozzle, with an initial truncation, to avoid an excessively large nozzle. Changes in ambient temperature and pressure are taken into account during ascent through the atmosphere to simulate flight conditions as accurately as possible. Truncation effect was analyzed at various ranges, finding that thrust losses begin when the truncation is greater than 60%. In addition, the effects of the shocks are analyzed, showing pressure peaks at some points on the plug wall, due to the interaction of the shocks. Recirculation zones was identified and the flight altitude influence over its location at the plug wall. The flow through the plug nozzle is modeled with computational fluid dynamics software.

Paper ID AE-2 **Programme ID** 55

Impacts of truncation and flight altitude on the flow and thrust of a plug nozzle

Patrick Haywood, Cho Yoon, Burak Turhan and Mahdi Azarpeyvand

This study presents an experimental investigation into the aerodynamic characteristics of Boundary Laver Ingestion (BLI) propellers. The custom-designed test rig, intended to enhance propeller research capabilities at the University of Bristol, was used to test an isolated 1-bladed propeller (15.5 x 12 inches) in boundary layer flow within the Boundary Laver Wind Tunnel facility. Thrust and torque measurements were recorded for two inflow speeds, 12 m/s and 6 m/s, at four tip clearance heights and rotational speeds ranging from 2000 to 5500 RPM in 250 RPM intervals, corresponding to an advance ratio (J) from 0.18 to 0.91. Additionally, thrust and torque measurements were taken at one tip clearance height, with yaw angles varying from 0 to 38 degrees at the same inflow speeds and RPM range. The research focuses on two main aspects: determining how aerodynamic performance changes with increasing submersion depth into the boundary laver (BL) and increasing yaw angle relative to the free stream. When the propeller is fully inside the turbulent flow (the smallest s/D), the thrust coefficient and propeller efficiency increase by approximately 12% and 10%, respectively, compared to when the propeller is outside the turbulent flow (the largest s/D). This trend, when compared to the boundary layer velocity and turbulent Kinetic Energy (TKE) profiles, can be attributed to lower inflow speeds and lighter kinetic energy in the smallest s/D. Additionally, as the yaw angle increases, the efficiency decreases, and the effective advance ratio at which maximum efficiency occurs also decreases. The findings from this study contribute to a better understanding of the aerodynamic characteristics of BLI propellers, a key technology for future Urban Air Mobility solutions and new civil aircraft configurations, such as Distributed Electric Propulsion.

Paper ID AE-3 **Programme ID** 61

Aerodynamic Performance of a Blade Section with Droop Leading Edge

Shanshan Xiaoa, Mark Jabbalb, Humberto Medinac and Mohammadreza Amoozgar

The use of droop leading edge (DLE) design, as a method to change the effective geometry of the propeller, has been proven to be effective in delaying dynamic stall, enhancing lift, reducing drag, and improving propeller efficiency. Potential applications for DLE include wind turbines and low-speed rotorcraft including eVTOL platforms. During operation, the characteristic Reynolds number of the local rotor section varies linearly along the span of the blade as either the speed increases or the chord changes. This implies that the blade will be subjected to different flow regimes and, thus, transitional effects will play an important role. Recently, a number of phenomenological transitional models that exploit the concept of Laminar Kinetic Energy have been developed 1-3 and show promising results when applied to canonical transitional test cases. However, more work is needed to understand their performance and limitations when applied to more complex geometries. In this work, the $k-\omega$ *LKE* model will be benchmarked when applied to DLE configurations. Comparison with wind tunnel experiments will be made for a range of droop angles, $0<\delta<30$, and Reynolds numbers in the range, $50{,}000<Re<500{,}000$. A wind tunnel test programme consisting of several 3D-printed, pressure-tapped aerofoils with fixed DLE angles will be mounted on a force balance to investigate changes in pressure, lift and drag coefficient. The experimental results will also be used to validate the transitional model.

Paper ID AE-4 **Programme ID** 114

Aerodynamic interactions between a propeller and a transonic wing-strut junction

Bryn Jones, Zachary Ciera, Scott Bennie and Marco Fossati

In the drive for increasingly ambitious noise, pollutants, and emissions targets in the aviation sector, it has become necessary to explore aircraft configurations beyond the conventional tubeand-wing. Two such promising technologies are the high aspect-ratio strut-braced wing (SBW), and distributed hybrid-electric propulsion (DHEP), which combined reduce total emissions; and pollutants/noise near the airport, respectively. The wing-strut junction causes entrainment of the flow, which can lead to intense shock waves in transonic flight. For a DHEP SBW, it may be necessary also to have a propeller in the neighbourhood of the wing-strut junction. In this work, we use Reynolds-averaged Navier-Stokes simulations and actuator methods to classify and quantify the aerodynamic interactions between a propeller and a wing-strut junction in transonic flow for a DHEP SBW. A clean wing configuration is directly compared with one where propellers are present, showing the incremental contribution to the flow due to the propeller, and changes in the shock wave and separation, especially in the wing-strut junction. Preliminary results show the propeller weakens the shock wave and changes the pattern of flow separation in the wing-strut region, suggesting it would be necessary to include the effect of the propeller in detailed DHEP SBW design studies. The weakening of the shock wave is possibly caused by an increase in pressure in the region due to the propeller swirl increasing the wing local effective angle of attack.

Paper ID AE-5 **Programme ID** 72

Avian flapping flyer response to discreet gusts

Charles Proe, Emily Shepard and Alper Celik

Birds are able to operate in turbulence levels that are currently prohibitive for UAM vehicles. Studying how birds respond to gusts can potentially help improve stability and safety of UAM vehicles operating in urban environments through bio-inspired design. To date, studies have quantified flight stability in birds flying in reverse Von Karman streets, downstream from turbulence grids and through upwards vertical gusts. Nonetheless, the control inputs that birds employ in response to vertical gusts, and the consequences for displacement and instability remain poorly understood. We use a novel experimental setup to quantify this in pigeons (Columba livia) , installing a gust generator across a wind tunnel test section that generates controlled sinusoidal gusts, where the amplitude and the frequency can be modified. Using a motion capture system to capture the body movements of the pigeons, changes in flapping frequency, flapping amplitude, tail fan angle, body displacement and acceleration. All flight test are conducted in a 1.5mx1.8m wind tunnel operating at 14 ms-1. The operating Reynolds number with regards to the root chord is 110,000 with pigeons having 6-8 wingbeats per second. Preliminary analyses suggest that birds experience greater displacements to gusts over 15 degree deflection and that this was associated with a reluctance to fly in these types of gusts. Further testing will assess the response to different turbulence integral length scales generated via turbulence grids. Overall, this will provide insight into the strategies that flapping fliers use to reject gusts and the displacement they experience according to the gust length scale and velocity. Following this, a UAM body would be made to test the effect of incorporating the gust response strategies into a fixed wing UAM design.

Paper ID AI-1 **Programme ID** 20

Machine Learning Based Intelligent CFD Surrogates for Interactive Design Exploration of Built Environments

Usamah Adia

Understanding the effect of airflow in enclosed/indoor environments is of great interest due to its close relationship to occupant safety, thermal comfort, and energy efficiency. Optimally placed air conditioning could increase the comfort of the inhabitants. In hospitals, airflow can distribute germs and can pose a significant health hazard. Indoor airflow patterns can be very complicated, and computer simulations are an invaluable tool for understanding their characteristics during the initial design phase when designers can explore various design scenarios, which take into account multiple constraints (infection risk, energy efficiency) without compromising the critical factor of indoor air quality (IAQ). However, the complex and dynamic nature of the problem makes it challenging to perform the fluid simulations quickly to explore 100 to 1000s of design options interactively to identify the optimum.

The project will combine computational fluid dynamic (CFD) simulations with machine learning (ML) algorithms to develop a novel data-driven and interactive physics-aware design optimisation method applied to the built environment. The outcome of this project is to identify which features are generalizable and how far we can use the data to create interactive CFD simulations. Find the limitations and challenges of using this method. So far work has been done on using different reduced-order models to reduce the dimensionality of CFD data.

Paper ID AI-2 **Programme ID** 24

Data-driven approach to modelling momentum deficit in a turbulent boundary layer over a rough surface

Martina Formichetti, Uttam Cadambi Padmanaban, Sean Symon and Bharathram Ganapathisubramani

Turbulent boundary layers over rough walls have been extensively studied for decades; however, they still pose a significant challenge for low-fidelity CFD solvers, especially for cases with varying roughness and/or transitionally rough regimes. Consequently, due to the complexity of creating a model that is both physically meaningful and generalisable, most of the research done until now has employed either high-fidelity CFD methods or experimental techniques. Nonetheless, the cost of both of the aforementioned techniques is substantial and, in order to improve the efficiency of studying such fluid dynamics problems from both an industrial and academic perspective, a reliable model is necessary that can be used in conjunction with low-fidelity techniques.

The data-driven method that we chose to model the effect of rough walls with streamwise variations in roughness on the turbulent boundary layer is a variational method used with a RANS set-up and PIV data. Moreover, DAFoam is used to perform the variational data assimilation step by using the discrete adjoint method. However, before diving into modelling the effect of roughness changes, we first decided to increase our confidence in the tool by testing the variational method with homogeneous roughness PIV data and a smooth wall RANS set-up. In this case, an unknown momentum forcing is optimized for a smooth wall TBL case that reduces the discrepancy between the mean velocity fields computed using the low-fidelity method and experimental data. The computed forcing can then be interpreted as a momentum deficit in the log region of the TBL exclusively due to the presence of wall roughness.

Preliminary results of the data assimilation step show profiles that are extremely close to the experimental data (within 1%), and the corresponding forcing term added by the variational method to account for the momentum deficit due to roughness can be quantified using the existing known form of the turbulent viscosity term.

Paper ID AI-3 **Programme ID** 38

Hard Constraint Projection in a Physics Informed Neural Network

Miranda J. S. Horne, Peter K. Jimack, Amirul Khan and He Wang

Machine learning provides a promising framework to simulate fluid dynamics at a fraction of the computational cost of traditional numerical methods[1]. Furthermore, the incorporation of domain knowledge into a neural network can improve the prediction accuracy, increase the model's explainability, and result in a neural network that is less reliant on training data.

Typically, the incorporation of the physical constraints into a neural network is only weakly enforced, for example, a PINN[2] weakly enforces the governing equation by incorporating a penalty term (often the equation's residuals) into the loss function. In the cases where a physical constraint is strongly imposed, the enforced governing equation is often either linear[3], weakly nonlinear, or an additional conservation law (such as the incompressibility constraint[4]).

In this work, we extend the research of Chen et al.[3] to embed hard constraints in a physics informed neural network (PINN). A PINN is used to estimate the stream function and pressure of a system governed by the 2D incompressible Navier-Stokes equation. The stream function is differentiated using automatic differentiation to recover an incompressible velocity field, and an unlearnable hard constraint projection (HCP) layer projects the velocity and pressure to a hyperplane that admits only exact solutions to a discretised form of the governing equations.

[1] S. L. Brunton, B. R. Noack, and P. Koumoutsakos, "Machine learning for fluid mechanics," Annual Review of Fluid Mechanics, vol. 52, 2020.

[2] M. Raissi, P. Perdikaris, and G. Karniadakis, "Physics-informed neural networks: A deep learning framework for solving forward and inverse problems involving nonlinear partial differential equations," Journal of Computational Physics, vol. 378, 2019.

[3] Y. Chen, D. Huang, D. Zhang, J. Zeng, N. Wang, H. Zhang, and J. Yan, "Theory-guided hard constraint projection (HCP): A knowledge-based data-driven scientific machine learning method," Journal of Computational Physics, vol. 445, 2021.

[4] A. T. Mohan, N. Lubbers, M. Chertkov, and D. Livescu, "Embedding hard physical constraints in neural network coarse-graining of three-dimensional turbulence," Phys. Rev. Fluids, vol. 8, 2023.

Multi-fidelity machine learning for improving RANS simulation data using DNS samples applied to the periodic hill

Harshinee Goordoyal, Katharine H Fraser and Andrew N Cookson

Computational fluid dynamics is commonly used in the design of cardiovascular devices. While Reynolds Averaged Navier Stokes (RANS) simulations offer a less computationally expensive option to solving the Navier Stokes equations, they can be inaccurate. Direct Numerical Simulations (DNS) are more accurate, but they are prohibitively expensive for parameter sweeping during design optimisation of complex 3D geometries.

In this study, data-driven machine learning is used to learn the approximation between low-fidelity RANS data and high-fidelity DNS data applied to the case of the periodic hill. This simple geometry was chosen as it is a representative separation-flow model that demonstrates important features observed in cardiovascular flows. The multi-fidelity MPINN architecture used in this study consists of a low fidelity approximator and a high fidelity approximator consisting of a linear single layer branch and a non-linear 4-layer branch. The training dataset consisted of velocity components obtained from RANS simulations and DNS mapped onto the same grid consisting of 14751 elements. The model was trained on all RANS points but only requires DNS data from 1000 randomly sampled points from the domain.

The model was applied to learn approximations between four different RANS models and DNS. The RANS models were K-Epsilon, K-Epsilon-Phi-Tf, K-Omega and K-Omega SST. The K-Omega SST case yielded the lowest root mean squared error (RMSE) compared to DNS, while the K-Epsilon case yielded the largest improvement in RMSE. Figure 1 shows the horizontal velocity field prediction by our model over the periodic hill. Figure 2 and Figure 3 show the horizontal and vertical velocity profiles respectively at the outlet for the multi-fidelity model compared to DNS and RANS (K-Omega SST). In all cases, the model prediction was better than the RANS simulation. The highest errors were observed along the lower wall boundary.

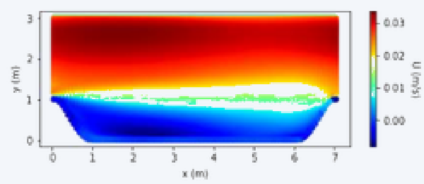


Figure 1: Horizontal velocity field prediction

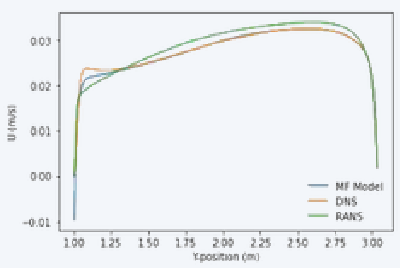


Figure 2: Horizontal velocity at the outlet (k-Omega SST)

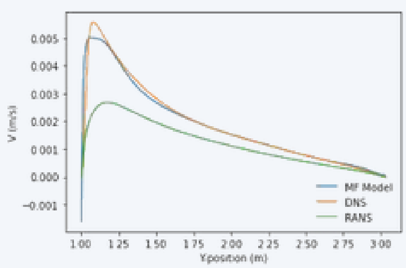


Figure 3: Vertical velocity at the outlet (k-Omega SST)

In this study, a data-driven multi-fidelity machine learning method was successfully employed to improve RANS simulation data. Future work will involve improving the sampling of points from the DNS dataset to better capture different flow characteristics and embedding knowledge of the boundaries in the model. The model will also be extended to different geometries.

Paper ID AI-5 **Programme ID** 66

Mixed data-source transfer-learning for a turbulence model augmented physics-informed neural network

Christian M Toma

Given the cost and complexity of capturing experimental data or running high-fidelity simulations, there is often a trade-off between accuracy and cost. With large quantities of unused data from various cases, data-driven methods aim to provide value to future projects in design or control. Presented here is a method for combining multiple data sources of varying fidelity and spatial resolutions using physics-informed neural networks. The model is trained from sparse measurements to predict the full flow field. We have also incorporated a turbulence model and transfer-learning, a method of applying the previously learnt trends to a similar problem, to enhance prediction performance when training with sparse experimental data.

So far, we have applied this method to three cases. For the first case, we considered a flow past a NACA 0018 airfoil at a Reynolds number of Re = 10,250 using the data from Symon et al. Starting from training the PINN with RANS data for the same conditions, we predicted the pressure side of the wing, which was previously obstructed from the laser sheet by the airfoil. In our second case, we used the data from Carter et al. for a NACA 0012 at a Reynolds number of Re = 75,000. Here the technique was applied to a significantly smaller window size relative to the domain and was able to predict the previously unseen wake region. The third case applies transfer learning between a variety of high Reynolds numbers for a NACA 0012 airfoil at post-stall angles of attack. Unlike the previous cases, the PINN is trained using an initial dataset with a lower Reynolds number and then fine-tuned with data from a higher Reynolds number.

Neural Network-driven Degree Adaptivity for Unsteady Incompressible Flows

Agustina Felipe, Ishaan Pradhan, Rubén Sevilla and Oubay Hassan

Accurate simulations of incompressible flow problems, governed by the Navier-Stokes equations, present a significant challenge in computational fluid dynamics. High-order solvers, such as the hybridisable discontinuous Galerkin (HDG) method, can yield precise solutions [1]. However, maintaining computational efficiency while ensuring accuracy necessitates adaptive strategies. The discontinuous nature of the HDG method supports using variable degrees, applying higher-order approximations only to elements demanding greater precision [2].

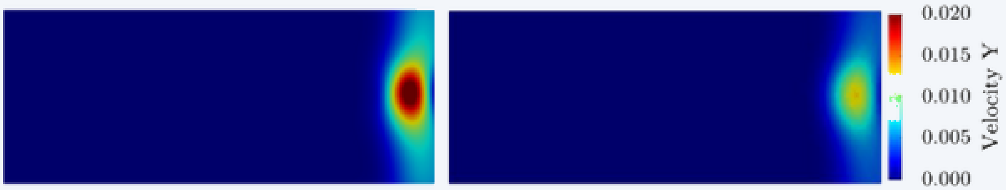


Fig.1: Comparison between vertical velocity fields with high-order elements (left) and degree adaptivity (right) on a gust simulation.

Figure 1 illustrates a traditional degree adaptivity method in a gust simulation. For a uniform degree of approximation, numerical dissipation is minimal, and the gust perturbation is accurately propagated, as seen in Figure 1a. Nevertheless, this approach is not optimal due to the excessive degrees of freedom in regions where the solution remains nearly constant. While degree adaptivity can mitigate this issue, it typically operates at a fixed time step, failing to consider the solution at the next time step. Consequently, numerical dissipation occurs, as illustrated in Figure 1b.

This work proposes a neural network approach to guide the adaptivity process in transient simulations. The core innovation is the neural network's capability to learn and predict solution behaviour from the current to the next time step. This enhances adaptivity in high-order solvers by dynamically adjusting mesh resolution based on the predicted solution, optimising computational resources, and maintaining accuracy in incompressible flow simulations. Examples will be provided to demonstrate the effectiveness and potential of the proposed approach.

[1] Giacomini, M., Sevilla, R. and Huerta, A., "Tutorial on Hybridizable Discontinuous Galerkin (HDG) Formulation for Incompressible Flow Problems". Modeling In Engineering Using Innovative Numerical Methods For Solids And Fluids., pp. 163-201 (2020), https://doi.org/10.1007/978- 3-030-37518-8 5.

[2] Giorgiani, G., Fernández-Méndez, S. and Huerta, A., "Hybridizable Discontinuous Galerkin with degree adaptivity for the incompressible Navier–Stokes equations". Computers & Fluids, Volume 98, (2014), https://doi.org/10.1016/j.compfluid.2014.01.011

Paper ID AI-7 **Programme ID** 50

Investigating Guiding Information for Adaptive Collocation Point Sampling in PINNs

Jose Florido

Physics-informed neural networks (PINNs) provide a means of obtaining approximate solutions of partial differential equations and systems through the minimisation of an objective function which includes the evaluation of a residual function at a set of collocation points within the domain. The quality of a PINNs solution depends upon numerous parameters, including the number and distribution of these collocation points. We considered a number of strategies for selecting these points and investigated their impact on the overall accuracy of the method. In particular, we suggest that no single approach is likely to be "optimal" but we show how a number of important metrics can have an impact in improving the quality of the results obtained when using a fixed number of residual evaluations. We illustrate these approaches through the use of two benchmark test problems: Burgers' equation and the Allen-Cahn equation.

An Operator Learning-based Approach for Modelling Convective Zonal Jets

Ankan Banerjee, Calum S. Skene and Steven M. Tobias

Zonal jets are ubiquitous throughout astrophysical and geophysical fluid flows. For example, they can arise due to nontrivial interactions between rotation and turbulent fluctuations in planetary bodies [1], where the dynamics is also greatly influenced by topographical effects due to tangent boundary planes. Here, we develop a reduced-order model to investigate the dynamics of zonal jets in the Busse annulus, a model for rotating convection that is parameterized by the rotation rate, topographical effects, and buoyancy due to convection. In this system, direct numerical simulations (DNS) using the pseudo-spectral solver Dedalus [2] shows that zonal jets can arise and exhibit a range of phenomena, including single jets, multiple jets, and bursting.

For our reduced order model we train a neural operator, capable of learning mappings between functional spaces, on the simulated data. Here, our neural operator is a Markov neural operator (MNO) [3], which predicts the one-step evolution of our system on a timescale larger than that of the DNS using a Fourier neural operator [4]. Temporal convergence is incorporated utilizing the dissipativity and Markovian properties of the solution space.

The resultant model is able to reproduce the invariant of the solution, such as zonal jets, over more than one time unit, as evident from Hovmöller plots of x-averaged temperature ($\langle\vartheta\rangle$x) and vorticity ($\langle\zeta\rangle$x), in figure 1. The x-direction is analogous to the azimuthal direction in spherical geometry.

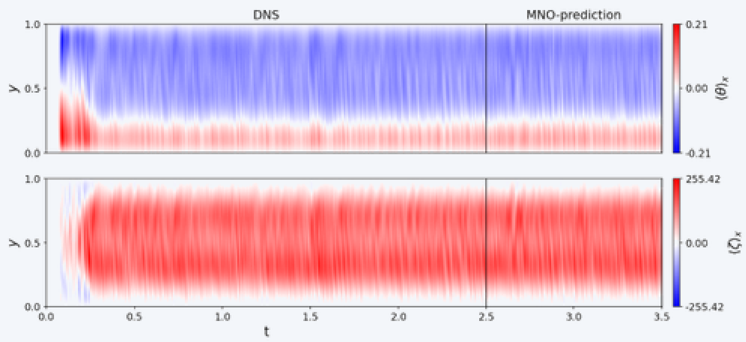


Figure 1. Comparisons between simulated and model-predicted temperature($\langle\theta\rangle_x$) and vorticity($\langle\zeta\rangle_x$), averaged along the x-direction, in the (t, y)-plane. The y-direction corresponds to the opposite radial direction in spherical geometry, and t is the dimensionless time.

[1] F. H. Busse, Convection Driven Zonal Flows and Vortices in the Major Planets, Chaos 4 (1994).

[2] K. J. Burns et al., Dedalus: A flexible framework for numerical simulations with spectral methods, Physical Review Research 2 (2020).

[3] Z. Li et al., Learning Dissipative Dynamics in Chaotic Systems, NeurIPS (2022).

[4] Z. Li et al., Fourier Neural Operator for Parametric Partial Differential Equations, ICLR (2021).

Paper ID AI-9 **Programme ID** 98

Neural Operators for Stable Prediction of Time-Dependent Partial Differential Equations

Seun Coker

Time-dependent partial differential equations (PDEs) play a vital role in the understanding and modelling of various scientific and engineering phenomena involving systems, such as weather forecasting. Efficient and accurate solution of these PDEs is crucial for optimizing designs and processes. Traditional numerical methods can be computationally expensive, but deep learning offers a promising alternative.

Neural operators, a class of deep learning models capable of learning mappings between infinite-dimensional functions, offer a powerful approach for solving PDEs. They directly learn the solution operators, allowing evaluation at different resolutions. However, learning the evolution of these operators for time-dependent PDEs presents a challenge, particularly the instability associated with autoregressive prediction methods. This work addresses this challenge by proposing novel training strategies that improve the stability and accuracy of neural operators when applied to time-dependent PDEs.

Paper ID AN-1 **Programme ID** 36

Extended linear theory of an oscillating foil propulsor

Amanda S. M. Smyth, Takafumi Nishino and Andhini N. Zurman-Nasution

Linear unsteady aerofoil theory such as the Theodorsen function has been used for rapid analytical prediction of unsteady aerofoil lift for many decades. However, its equivalent for calculating unsteady thrust, the Garrick function, substantially over-predicts propulsive efficiency. In this study we test the hypothesis that the central shortcoming of linear small amplitude models such as the Garrick function is the failure to account for thrust-induced flow acceleration, a well-known phenomenon in open propeller/turbine modelling. Taking inspiration from these devices, in this study an analytical model is developed by coupling the Garrick function to an actuator disc model, in a manner analogous to blade-element momentum theory used for wind turbines and propellers. This amounts to assuming the Garrick function to be locally valid and, in combination with a global control volume analysis, enables the prediction of the flow acceleration at the aerofoil. Both steady and cycle-averaged actuator disc frameworks are developed, and it is demonstrated that if inviscid and irrotational flow with a planar wake topology is assumed, the cycle-averaged actuator disc equations differ from the steady equivalents through only two terms: one representing oscillation energy added by the foil, and the other the added mass energy at the exit face of the control volume. The new model is demonstrated to substantially improve the agreement with Large-Eddy Simulations of an aerofoil in combined heave and pitch motion. Discrepancies remain, which may result from leading-edge vortex separation which is not captured by Garrick's model. The outcomes of this study provide a new analytical framework for propulsive foil modelling, suggesting that linear small-amplitude aerofoil theory can be used to model foil propulsion problems to first-order accuracy

Paper ID AN-2 **Programme ID** 43

Eliminating the Kelvin Wake

Jack Keeler

Everyone has seen the v-shaped Kelvin wake-pattern visible in the wake of a moving object on the surface of water. These patterns are a rare example of a fluid dynamics phenomena well-known to scientists and layman alike. However, the wake is undesirable for a number of reasons; it can cause erosion to river banks and cause wave-drag, thus reducing the fuel efficiency. Therefore, the design of a moving body that can reduce or even eliminate these waves is important for sustainability. A typical approach is to model the boat by an imposed pressure distribution in the free-surface Bernouilli condition. In this talk we show, using a simple mathematical argument, that by a judicious choice of a pressure distribution, wave-free solutions are possible in the context of a model system; the forced Kadometsev-Petviashvili equation. Strikingly, we show that these solutions are stable, so they could potentially be visualised in a physical experiment. The answer of the question in the title is therefore; 'Yes', it is possible to eliminate Kelvin waves.

Paper ID AN-3 **Programme ID** 62

Sharp Corner Singularity of the White-Metzner Model

Christian Jones

Polymer melts and solutions are important materials in industry, and are categorised as viscoelastic, exhibiting properties of both viscous fluids and elastic solids. Modelling these fluids requires complex constitutive relations that extend beyond the classical Newtonian stress-strain relationship. The use of these materials in processing applications is widespread, and stress-singular geometries are commonly encountered, for example in contraction flow, die swell and cross-slot flow. Understanding these stress singularities is; therefore, important to the quality of any processed products.

In this talk, we study one such singularity via the flow of a White-Metzner fluid around a re-entrant corner. A natural stress decomposition of the extra-stress tensor is used, allowing for a complete description of the leading order flow around the corner. This is achieved via the method of matched asymptotics, which is used to construct selfsimilar solutions for the stress and velocity fields, with matching to both upstream and downstream boundary layers. Our theoretical findings are confirmed using numerical simulations, with discussion of the validity and limitations of our asymptotic approximations.

Transient shear wave propagation in a solid-liquid coupled system

Aaron D'Cruz and Pierre Ricco

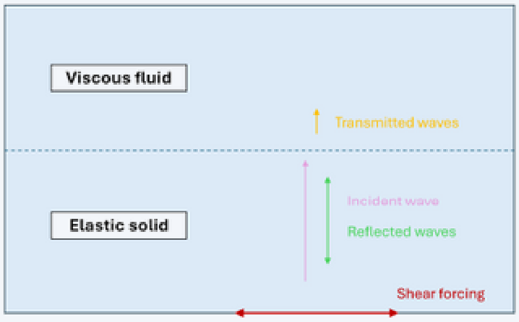


Figure 1: Schematic of the shear-wave propagation through the solid-fluid system due to forcing at the lower boundary.

We present a novel closed-form analytical solution for the response of a forced solid-fluid system, shown in Figure 1. It consists of a wide finite elastic solid located underneath a Newtonian fluid. The bottom surface of the solid is forced horizontally for a finite time period. The system is governed by coupled partial differential equations, describing the shear displacement in the elastic solid and the shear velocity in the fluid. The boundary conditions represent the continuity of shear velocity and shear stress at the solid-fluid boundary, the lower boundary shear forcing by continuous imposed displacement or shear rate, and the decay of the shear velocity of the fluid at large distance from the solid-liquid interface. Initial conditions of zero displacement and velocity are used.

The transient response of the system is obtained analytically using integral transform techniques. The solution clearly shows a set of superposed elastic waves in the solid: an incident wave engendered by the driving action at the lower boundary, partially reflected waves at the solid-fluid interface, and perfectly reflected waves at the lower boundary. The transmitted waves which enter the viscous fluid via the interface coupling have an exponentially damped oscillatory profile, with a penetration depth that depends on the fluid viscosity. This response is highly reminiscent of the classic periodic 'Stokes layer', and the transient extensions of similar problems[1].

These waves have interesting applications in active mixing[2,3], turbulent drag reduction[4,5], and sensing for biological and chemical flows[6]. The analytical solution gives physical insight which cannot be obtained through a purely numerical solution of the governing system, and is more comprehensive than studies of shear wave propagation using simplified eigenmode analysis, or analogical models not derived from first-principles physical laws.

1 Panton, J. Fluid Mech. 31 (4) (1968).
2 Ricco and Hicks, J. Eng. Math. 111 (2018).
3 Shilton et al., J. Appl. Phys. 104 (1) (2008)
4 Bird and Morrison, Flow Turbulence Combust 100 (2018).
5 Quadrio and Ricco, J. Fluid Mech. 667 (2011).
6 Lange et al., Anal. Bioanal. Chem. 391 (2008).

Paper ID AN-5 **Programme ID** 74

Free-surface flow over generalised topographies using an arclength formulation

Henry Writer and Phil Trinh

We are interested in understanding the mathematical behaviours of free surface waves generated by flow passing over a varying bottom topography in two-dimensional flow problems. Such wave-structure interaction problems are relevant in the design of flood control infrastructure and, at very different length scales, the behaviour of air currents generated by winds passing over mountains.

Historically, in modelling the fluid as a potential flow, there is a large amount of prior work investigating waves using a conformal mapping and/or boundary integral approach. For simple geometries, such as flow over a step, a substantial amount of analytical insight can be drawn. In this talk, we present an alternative formulation, which uses an arc-length characterisation of the problem; this method allows us to consider more general bottom topographies. The challenge of this approach is that flow equations now include multiple coupled integral equations. In this talk, we discuss the limitations of prior methods, introduce the arc-length formulation, and discuss numerical solutions and asymptotic results.

Paper ID AN-6 **Programme ID** 86

The Relation between rotating curved channel flow and Taylor-Dean flow

D. P. Wall

Taylor-Dean flow (TDF) describes the flow of fluid confined between two co-axial cylinders driven by the rotation of the cylinders about their axis and an imposed azimuthal pressure gradient. Rotating curved channel flow (RCCF) describes the flow through a curved channel that is subject to a spanwise system rotation. A number of theoretical and experimental studies of the RCCF system have been undertaken, since the rotation of the channel can be chosen so that the Coriolis force opposes the centrifugal force, allowing for investigation of these two instability mechanisms [1]. The similarity of these two flow systems has previously been noted [2],[1], but has not been explained. In the present study we show that there exists a mathematical equivalence between RCCF and a class of TD flows. We describe the transformation that converts this class of TD flows to RCCF, and demonstrate the equivalence by example calculations. The equivalence allows for further insights into both flows; in the present study we re-interpret the stability of RCCF flow by examining the flow in a TD configuration.

[1] O. John E. Matsson and P. Henrik Alfredsson. Curvature- and rotation-induced instabilities in channel flow. Journal of Fluid Mechanics, 210:537563, 1990.

[2] Innocent Mutabazi, Christiane Normand, and Jos Eduardo Wesfreid. Gap size effects on centrifugally and rotationally driven instabilities. Physics of Fluids A: Fluid Dynamics, 4(6):1199–1205, 06 1992.

Paper ID AN-7 **Programme ID** 100

On Constructing Lagrangians for Dissipative Systems: Application to Hydrodynamics

Josiah-Shem Davis

When physical systems convert energy to heat in a moving mechanical system these systems are said to be dissipative dynamical systems. In the 1930s it was shown to be impossible to derive equations of motion for a constant mass Newtonian fluid which included dissipative terms, from a variational principle - that is, using Euler-Lagrange equations containing only integer-order derivatives of coordinates [1]. However, since Fred Riewe in 1995 it has been shown that with the addition of non-integer order "fractional" derivatives it is possible to derive equations of motion for dissipative dynamical systems from a variational principle [2,3].

The following presentation is an exploration of a novel functional-derivative-based approach to the stationary action principle, wherein by making use of fractional derivatives, equations are derived describing a general system of linear dynamical equations.

Application is then made in the description of the flow of a coarse-grained, and linearised version of hydrodynamics, which is compared to those of Smoothed Dissipative Particle Dynamics (SDPD) equations for the fluid flow of a coarse-grained, and linearised version of the Navier-Stokes' equations [4].

[1] Bauer, Paul S. (1931). Proceedings of the National Academy of Sciences, 17(5), 311-314.
[2] Riewe, Fred (1995). Physical Review E, 53(2), 1890-1899.
[3] Lazo, Matheus J. and Krumreich, Cesar E. (2014). Journal of Mathematical Physics, 55(12), 122902.
[4] Español, Pep and Revenga, Mariano. (2003). Physical Review, 67(2), 026705.

Paper ID AN-8 **Programme ID** 128

A Quantitative Microbial Risk Assessment (QMRA) framework for comparing infection risk from aerosol generation during toilet flushing

Ciara A. Higham, Dr Martín López-García, Prof. Catherine J. Noakes and Dr Louise Fletcher

Flushing the toilet is an aerosol generation procedure, dispersing microorganisms that can transmit diseases (via external room airflows and settling due to gravity), such as gastrointestinal and respiratory infections. Despite the detection of these aerosols and microorganisms, a quantitative analysis linking aerosol dynamics to infection risk remains underexplored. We developed a framework to evaluate the infection risk to a second individual using a toilet after an infected individual has flushed. This framework integrates experimental measurements of aerosol particle concentrations in a controlled chamber with a mathematical model based on Quantitative Microbial Risk Assessment (QMRA). By focusing on the dispersion of the aerosols generated, we analyse how variables like cubicle space and cubicle entry timing affect risk for pathogens like SARS-CoV-2 and norovirus. The model reveals that aerosol generation via flushing the can have a non-negligible impact on infection risk, especially for pathogens with higher concentrations in faeces (e.g., norovirus exhibited a 136 times greater maximum risk than SARS-CoV-2). Mean and median risks were reduced when the second individual entered 60 seconds post-flush compared to entering within 60 seconds post-flush. The timing of entry proved more critical in mitigating risk than the duration of occupancy. To mitigate infection risk from toilet aerosols, effective ventilation to manage aerosol dispersion before entry is crucial. Allowing sufficient time between toilet usage allows aerosols to be removed via ventilation or settling due to gravity. This reduces infection risk more effectively than simply reducing occupancy time. Our model underscores the importance of understanding aerosol fluid dynamics from toilet flushing to inform public health measures. However, further quantitative data is needed, particularly in high-risk environments like hospitals, to accurately quantify absolute risks and optimise mitigation strategies.

Paper ID AN-9 **Programme ID** 95

On the evaluation of complex flow around a transparent non axi-symmetric corotating system mounted in an enclosure

Ibrahim Abubakar Masud

The geometry and dynamics of co-rotating systems, such as turbomachinery, rotating machines, and hard disk drives (HDDs), play a crucial role in their efficiency related to fluid flow. These systems can experience long-term failures due to issues arising from fluid dynamics. In HDDs, for example, flow-induced vibrations occur not only because of the system's geometry and dynamics but also due to the movement of the arm from the shroud to the disk area. In our refractive index matched HDD model, the disk and arm are the primary rotating and moving components, respectively. The non-axisymmetric shrouded wall acts as a barrier to external atmospheric conditions by directing airflow, reducing turbulence, and managing pressure distribution. The rotating disks create centrifugal forces that drive the fluid radially outward, establishing a boundary layer and a pressure gradient, resulting in a complex airflow pattern. The arm interacts with this airflow, experiencing drag forces and contributing to the overall fluid velocity within the drive, particularly with the inflow from the shroud opening. This inflow exerts shear stress on the disk region, contributing to momentum exchange, especially due to the accelerated flow from the arm-hub region. Our two-dimensional average velocity results were revealed in Cartesian coordinates and analyzed in cylindrical coordinates. In this symposium, we intend to demonstrate the velocity and vorticity distributions, which showed irregular profiles resulting from the momentum exchange. Rigid body rotation around the hub varies according to regions with forced vortex regions existing within the flow. Moreover, in some regions of the flow, there are indications of secondary flow suspected to exist in the axial plane below the upper disk and above the lower disks, respectively.

Paper ID AN-10 **Programme ID** 70

The Saffman-Taylor viscous fingering problem in a wedge

Phil Trinh and Cecilie Andersen

The analysis of Saffman-Taylor viscous fingering is a classic challenge in potential flow theory. In this problem, an inviscid fluid is injected into a Hele Shaw channel filled with a viscous fluid; an interface forms, and in the steady limit, a single finger occupies some proportion of the width of the channel. Asymptotic analysis as the surface tension parameter approaches zero demonstrates a continuum of permissible finger widths; however, experiments demonstrate that a unique zero-surface-tension limit is selected, and for many years, it was not clear how this selection mechanism was derived. It was eventually shown that exponential asymptotics can be applied to derive the precise selection of the Saffman-Taylor finger in the zero surface-tension limit.

In this talk I discuss a generalisation of the Saffman-Taylor problem to a wedge geometry. This is motivated by desires to understand the more physically relevant Saffman-Taylor instability in a circular geometry, with injection of the inviscid fluid outwards from a central source. I discuss the new challenges and differences introduced by the geometry, and the necessary exponential asymptotics to resolve the problem.

Paper ID BI-1 **Programme ID** 73

A novel perspective on cerebrospinal fluid flow in perivascular spaces

Gregory Holba

Cerebrospinal fluid (CSF) bathes the brain and flows within and around it, delivering nutrients and carrying away waste products. Without effective waste clearance, neurodegenerative conditions can develop, such as Alzheimer's and Parkinson's diseases, which affect over 50 million people worldwide. Given the increasing prevalence of such conditions, understanding the causes of these diseases is becoming imperative.

The flow of CSF is of particular interest within the perivascular spaces (PVS) of the brain. These are thought to be the entry and exit points of the glymphatic system, the brain's waste clearance system. Where blood arteries penetrate the brain, they are surrounded by PVS, which act as annular conduits for CSF flow into the brain matter.

Given that CSF flow is known to be strongly influenced by the cardiac cycle, a common approach to modelling CSF flow in the PVS has been to analyse the flow driven by the motion of blood vessel walls, assumed to exhibit a sinusoidal motion. A wide range of flow rates have been predicted, often thought to be too small in themselves to account for effective waste clearance.

In this work, we propose a more complex pumping mechanism that drives the flow in the PVS. We consider a combination of an elastic peristaltic blood vessel wall motion and an additional physiological driver that induces oscillations at the outer walls of the PVS. We use lubrication theory to derive the equations of motion and to investigate the effects of the additional driver on the pressure and flow rate of the CSF in the PVS.

Paper ID BI-2 **Programme ID** 89

Study of arterial transit times in the brain modelled as a porous medium

Isla Henderson, Vijay Nandurdikar, Ajay Harish, Alistair Revell, Laura Parkes and Adam Greenstein

Cerebrovascular diseases such as stroke and Alzheimer's disease are a leading cause of death and disability worldwide, and are often linked to chronic reduction in blood supply to the brain [1]. An important marker of brain vascular health is arterial transit time (ATT), the time taken for blood from large arteries in the neck to travel to the capillaries of a specific region of brain tissue. Longer ATT is often associated with ageing and various neurovascular diseases, making it useful for early detection of cerebrovascular impairment [2]. Despite its importance, the determining factors of ATT remain poorly understood, necessitating accurate modelling to further our understanding.

Due to disease-induced physiological changes, capillary networks are pruned and regenerated. This results in a non-uniform, time-dependent network distribution across the brain tissue. As it is computationally intractable to individually model the billions of blood vessels within the brain, we model the capillary bed as a homogenised porous medium connected to arterial flow. Thus by varying the effective properties of the capillary bed, we can simulate microvascular changes with neurovascular disease, and their effects on ATT.

Currently, a computational model of the cerebral vasculature is under development using OpenFOAM, connecting a pulsatile arterial flow to a probabilistic model of the capillary bed. Latin hypercube sampling is used to apply a range of permeability values to the domain, capturing the heterogeneity of brain tissue. Once established, this model aims to investigate changes to ATT with diseases like vascular dementia and Alzheimer's, guided by clinical insight and experimental data.

[1] Payne, S. et al. Review of in silico models of cerebral blood flow in health and pathology, Progress in Biomedical Engineering 5 022003 (2023).

[2] Feron, J. et al. Determinants of cerebral blood flow and arterial transit time in healthy older adults, bioRxiv (2023).

Paper ID BI-3 **Programme ID** 97

Automated Workflow For Constructing Virtual Twins for Haemodynamic Analysis of Stenosed Native Aortic Valves

Cristina Teleanu

Aortic stenosis is characterised by calcium deposits that narrow and stiffen the aortic valve (AV), resulting in elevated transvalvular pressure, reduced blood flow, and increased cardiac workload. It is known that fluid-structure interaction (FSI) simulations offer the most accurate representation of the interaction between the AV and its surrounding flow field, and associated pathologies. FSI simulations, however, particularly involving complex, patient-specific geometries, present challenges such as time-consuming image-based anatomy reconstruction, complex meshing, long set-up times, and difficulty of automation. We present an automated workflow designed to create FSI simulation-ready virtual twins of the aortic root, to analyse blood flow in stenosed AVs. Using computed tomography angiography (CTA) scans, echocardiography, and Doppler imaging data, patient-specific haemodynamic simulations are obtained using the workflow below. Detailed models of the aortic root, including the aorta, left ventricle outflow tract, and calcifications, are generated from CTA scans. Valve leaflets, not directly obtained in segmentation due to imaging spatial and temporal resolution limitations, are inferred from detected anatomical landmarks for each patient. These steps are performed using SimplewareTM Software (Synopsys Inc, USA), and an in-house code. Fully coupled FSI simulations of the AV haemodynamics are then conducted. This process is applied to several patients using Ansys LS-DYNA, augmented with custom Python scripts to provide performance predictions for comparison with clinical measurements. Model outputs are validated against Doppler traces. This automated workflow, incorporating FSI simulations, aims to provide a tool for understanding the physiology of aortic stenosis and the underlying disease mechanisms. With further refinement, it can form a key building block of in-silico trials of cardiac devices and interventions.

Paper ID BI-4 **Programme ID** 99

An investigation of the fluid structure interaction in articular cartilage across disparate scales

Emily Butler, David Head, Mark Walkley, Michael Bryant and Greg de Boer

Articular cartilage (AC) is found at opposing surfaces in mammalian joints. It provides a smooth bearing surface, promoting low friction articulation and facilitating continuous operation under relative motion. A lack of cells within AC renders a low capacity for intrinsic healing or repair. This leaves it prone to degeneration and disease, resulting in a high clinical demand for cartilage repair. To streamline treatment, an accurate computational model of the tissue is essential to inform rapid prescreening of therapeutic interventions. Current models generally use a single-scale approach, which fails to capture the complex multi-scale features of the tissue, including its intrinsic heterogeneity and depth dependant properties. Instead, this project aims to couple an immersed fibrous network (micro-scale) model with a continuum (macro-scale) model, to create a multi-scale poroelastic representation of the fluid-structure interaction within AC across disparate scales.

A continuum-continuum model has been developed for validation of the multi-scale poroelastic framework. Damage is incorporated in the micro-scale continuum model to replicate fibrillation of surface layers and calcification of the deep zone. The resulting transient macro-scale response is determined for progressively degrading micro-scale damage of the tissue.

A fibrous network representation of the micro-scale behaviour is being integrated into the multi-scale model to include anisotropy, zonal structuring, and depth dependent properties. The regular spring lattice structure, in which bond occupation and spring stiffness can be freely varied, is advantageous for damage modelling, as fibre anisotropy and damaged tissue can be incorporated by selective modification of the micro-scale elastic elements. Developing an understanding of the change in behaviour and mechanical properties of damaged AC tissue is a critical step to inform clinical interventions for cartilage repair and future replacement.

Paper ID BI-5 **Programme ID** 117

Mathematical Modeling of Varicose Veins

Giannopoulou Ourania

Cardiovascular diseases affect over half of the global population, manifesting in various forms throughout the body. Varicose veins are one such condition, characterised by high blood pressure exerting stress on the vessel walls, leading to their dilation. This disrupts normal blood circulation and, if left untreated, can progress to more severe stages, including the development of ulcers. Given the fact that there is general consensus on the critical role of pressure in the onset of varicose veins, this study examines the role of pressure within the framework of non-Newtonian fluids. Unlike Newtonian fluids, non-Newtonian fluids more accurately represent blood flow in veins, where the radius to length ratio is small, allowing for a better understanding of shear thinning effects on vessel walls and consequently the wall pressure gradients. Furthermore, the study investigates the influence of pressure-regulating hormones, such as progesterone and oestrogen, whose imbalances are considered a primary cause of the disease. The governing equations of the model under investigation are the incompressible Navier Stokes equations for an incompressible fluid combined with a non - Newtonian viscosity law. For the discretization of the resulting system of equations we utilise the open source software Fenics based on finite elements.

Paper ID CA-1 **Programme ID** 23

Development and testing of moving boundary theories for modelling dynamic contact lines on precursor films

Meg Richards

The flow of thin films driven by capillary forces occur widely in the natural world and within industry. Here, we present a novel mathematical model describing the generalised time-dependent evolution of contact lines driven by surface tension over a thin precursor film. Whilst most previous analytical descriptions rely on a quasistatic theory to develop an asymptotic model, the novelty in our moving boundary model comes from applying a quasistatic approximation in the nose region only and keeping a fully general model elsewhere. Whilst the existing quasistatic models are specialised to a droplet configuration, the new model is more general and can be applied to both the spreading of a droplet and injection into a lock exchange configuration.

Paper ID CA-2 **Programme ID** 106

Quantifying stick-slip motion of droplets on chemical patterns

Nick Creasy

Controlling droplet transport and localisation on solid surfaces is important for many technologies, such as microchemical reactors, liquid-repellent surfaces, and micro-printing. Consequently, considerable effort has been devoted to designing solid surfaces that control wetting and droplet's contact area and/or direct droplet's motion. A key aspect here is the concept of stick-slip contact line motion, which arises because of impurities on the surface.

In this work, we study droplets that are subjected to external, slow time-dependent variations, such as evaporation and condensation, to quantify the emergence of stick-slip motion. We consider a two-dimensional droplet that is slowly evaporating on a smooth chemical pattern. Under these conditions, it has recently been shown that the motion and shape of the droplet is quantified in terms of a hierarchy of bifurcations. By gradually decreasing the smoothness of the pattern, we describe the emergence of stick-slip mechanisms in terms of bifurcation transitions. We provide both analytical and computational results based on a diffuse interface model, observing good agreement.

Paper ID CA-3 **Programme ID** 30

Capillary effects in vorticity-driven surface waves

Josh Shelton

This talk concerns travelling free-surface water waves, under the action of gravity, surface tension, and nontrivial fluid vorticity. Our focus is on nonlinear solutions for small surface tension, a regime inhabited by many surface waves in nature. Numerical and asymptotic solutions will be presented, for both the unknown fluid surface and the internal velocity field. Recent advancements have shown that significant analytical progress can be made in the small-surface-tension limit, through the application of beyond-all-order asymptotics. This yields asymptotic solutions for the high-frequency modes induced by capillary effects, and a solvability condition corresponding to the non-existence of solutions at certain parameter values.

Paper ID CA-4 **Programme ID** 46

High-Precision Modelling of Capillary Forces for Precision Microfabrication

Mokhtari Ahlem, Panter Ramon Jack and Atkin Chris

Recent advancements in the fluidic self-assembly (FSA) method, developed by researchers from Seoul National University and LG Electronics [1], alongside die-to-wafer self-assembly by CEA-Leti and Intel [2], underscore the significance of capillary assembly in semiconductor manufacturing and display technologies. These breakthroughs showcase the remarkable precision and scalability achieved by harnessing capillary forces to position 45 μm size components with high accuracy. As these technologies drive towards further miniaturization, understanding capillary forces between finite sized objects becomes crucial. While capillary forces between infinite plates is well-established research, dealing with finite sized objects at the microscale remains challenging. Therefore, our research investigates the behaviour of capillary bridges between finite-sized micro solids, analysing the influence of factors such as geometry, corner pinning, volume, and surface energy.

Our study relies on an advanced mesoscale 3D numerical model developed in-house. we constructed the free energy of a bi-fluidic system interacting with a solid surface, in which we simulate intricate fluid-fluid interfaces of any shape and structure with great precision. This level of accuracy is essential for our specific interest in precise microfabrication techniques. Thus, to overcome the challenge of replicating contact angles in diffuse interface models, we incorporate a quartic wetting boundary condition. This method ensures a deviation of less than 1 degree between the simulated and desired contact angles, addressing a common issue encountered in such models, where differences typically fall within 4.8 degrees [3].

In contrast to prior numerical studies that neglected liquid overflow on solid corners [4], our model addresses this effect. Earlier investigations highlighted that excessive liquid volume is detrimental [5] and suggested a 60-degree contact angle is optimal without underlying the fundamental reasons [6]. Our extensive parametric study reveals that overflow intensifies lateral capillary forces, enhancing alignment while reducing normal capillary forces and weakening the attachment.

In conclusion, our research reveals an optimal interval where both components of the capillary force are sufficiently strong. This interval is crucial for precise positioning of micro components

[1] D. Lee et al., ‘Fluidic self-assembly for MicroLED displays by controlled viscosity’, Nature, vol. 619, no. 7971, Art. no. 7971, Jul. 2023, doi: 10.1038/s41586-023-06167-5.

[2] E. Bourjot et al., 'Integration and Process Challenges Of Self Assembly Applied To Die-To-Wafer Hybrid Bonding', in 2023 IEEE 73rd Electronic Components and Technology Conference (ECTC), Orlando, FL, USA: IEEE, May 2023, pp. 1397–1402. doi: 10.1109/ECTC51909.2023.00239.
[3] 'Effect of wall free energy formulation on the wetting phenomenon: Conservative Allen–Cahn model | The Journal of Chemical Physics 2023 | AIP Publishing. Available: https://pubs.aip.org/aip/jcp/article/159/16/164701/2918009/Effect-of-wall-freeenergy-formulation-on-the
[4] object Object, 'Modeling capillary forces for large displacements. Available: https://core.ac.uk/reader/148010190.
[5] M. Mastrangeli, Q. Zhou, V. Sariola, and P. Lambert, 'Surface tension-driven self-alignment', Soft Matter, vol. 13, no. 2, pp. 304–327, 2017, doi: 10.1039/C6SM02078J.
[6] A. Bond et al., 'Collective Die-to-Wafer Self-Assembly for High Alignment Accuracy and High Throughput 3D Integration', in 2022 IEEE 72nd Electronic Components and Technology Conference (ECTC), May 2022, pp. 168–176. doi: 10.1109/ECTC51906.2022.00037.

Paper ID CA-5 **Programme ID** 63

Filling the Gap on Drop Dynamics

Kat Phillips

A droplet falling towards a liquid bath will only rebound in the presence of a persisting thin layer of air, which separates the two liquids from coalescing. Within this talk we will introduce a fully coupled dynamic model for the drop-air-bath system, through assuming quasi-potential flow in the deep liquid bath, and a lubrication approximation in the air layer. Through obtaining a thin film equation in the air layer, we deduce the pressure transfer between the drop and the free surface, and the bounds of the lubrication region where this pressure is felt. We demonstrate the axisymmetric impacts of solid and liquid spheres on deep liquid baths in both two and three dimensions, and demonstrate the potential for this lubrication-mediated model to be capture complex repeated bouncing behaviours through simulating Faraday pilot-wave dynamics, where drop-bath pairings are able to enter stable periodic behaviour.

Paper ID CA-6 **Programme ID** 69

Quantifying the Importance of Diffusion in Mixing Printed Micro-droplets

Yatin Darbar

This research investigates the mixing behaviour of inkjet-printed micro-droplets using high fidelity simulations performed with the Finite Volume Method toolbox OpenFOAM. This study is pertinent to numerous industries, involving the production of bio-compatible materials, textiles, and electronics, where effective mixing of printed droplets is crucial. Previous experimental studies have primarily focused on larger droplets, failing to capture the diffusion effects accurately at inkjet scales. Our innovative numerical approach employs the Volume of Fluid method and integrates a transport equation for a conserved scalar within the droplet, enabling a direct analysis of mixing processes in coalescing and impacting droplets.

By numerically modelling these processes, we can quantitatively evaluate mixing using a metric derived from the standard deviation of the conserved scalar. Our simulations, validated against experimental data, demonstrate that molecular diffusion contributes minimally to the homogenization of inkjet droplets (around 50µm in diameter) within practical time frames. The challenge of high computational costs at this scale is mitigated by performing simulations at an intermediate scale, providing a novel approach for bridging the gap between laboratory experiments and real-world applications. This approach allows for accurate representation of diffusion effects in practical scenarios.

Our findings indicate that critical parameters, such as scale and droplet separation, significantly influence mixing efficiency. These insights provide valuable guidance for optimizing inkjet printing processes, ultimately enhancing the performance and uniformity of printed materials, ensuring high-quality outcomes in inkjet-printed products.

Paper ID CA-7 **Programme ID** 83

Engineering using capillary forces: using simulations, experiments and theory to understand capillary bridges on complex geometries

Jack Panter

It is particularly challenging to control the 3D assembly of solid structures of sub-millimetre size. This is because long range Van der Waals or electrostatic forces become large relative to gravity and elasticity, so conventional tools for manipulating macroscale objects become ineffective. However, it has been proposed that capillary forces could be a powerful tool to perform sub-mm manipulation, as at such length scales, these forces become dominant [1]. This concept is beginning to show some success across a range of emerging innovations, such as the high-throughput fabrication of micoLED displays [2], in self-assembling smart-materials [3], and in nanowire alignment for sustainable touchscreens [4].

However, aside from the simplest geometries, there is little fundamental understanding of how capillary forces mediate interactions between solid structures. This is due to the complexity of the fluid-fluid interface morphology and the three-phase contact line shape, both of which are in general non-analytic and discontinuous functions of solid configuration.

To begin understanding the coupling between solid geometry, liquid morphology, and capillary forces, we develop a combined approach using simulations, experiments, and theory. In the simulations, we develop a novel phase-field method, capable of modelling fluids interacting with complex solid structures. In experiments, we use high-resolution 3D printing with precision load cells to measure capillary forces directly. Together, this shows how capillary bridge forces are controlled through solid design, making a step towards 'capillary engineering'.

[1] K. S. Kwok, et al., Adv. Mater. Interfaces, 7, 1901677, (2020), DOI: 10.1002/admi.201901677
[2] Lee, D., et al., Nature, 619(7971), 755–760. (2023), DOI: 10.1038/s41586-023-06167-5
[3] McCaskill, et al., Adv. Mater., 35(51) (2023), DOI: 10.1002/adma.202306344
[4] S. Kang et al., Nano Lett., 15(12), pp. 7933–7942, (2015), DOI: 10.1021/acs.nanolett.5b03019

Paper ID CA-8 **Programme ID** 94

Origins of the nanobubble zeta-potential

Duncan Dockar, Patrick Sullivan, Jacqueline Mifsud, Livio Gibelli and Matthew K. Borg

Bulk nanobubbles have promising applications in water treatment, enhancing efficiencies of Advanced Oxidation Processes (AOPs) and improving aeration, as well as in biomedical imaging, cancer treatment, improving ammonia fuel combustion efficiency, and microfluidic surface cleaning. However, there remains a lot of controversy surrounding their apparent long lifetimes reported in the literature, given there is no experimental method to directly confirm their existence, and there is currently no well-defined mechanism for their diffusive stability.

There is increasing evidence to show that bulk nanobubbles possess a zeta-potential around

10 mV 10 mV

, suggesting they are negatively charged. Following from theories of Electric Double Layers (EDLs), it has recently been proposed that there is an additional electrostatic stress on nanobubble surfaces, counteracting the Laplace pressure, which stabilises them against dissolution.

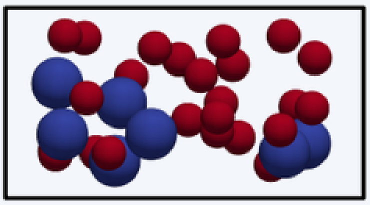
In this work, we present high-fidelity Molecular Dynamics (MD) simulations of bulk nanobubbles in a sodium iodide electrolyte solution. We find that ions do adsorb on the nanobubble surface, revealing an EDL-like distribution of ions surrounding the nanobubble, and by comparing to the Poisson–Boltzmann equation, we obtain a zeta-potential value consistent with experiments. However, we find that the adsorbed ions do not provide any additional electrostatic stress on the nanobubble surface, and the resulting internal gas pressure is still well predicted by the standard Laplace pressure equation, contradicting the proposed theories for bulk nanobubble stability. While the ion distribution resembles an EDL, the water molecules rearrange themselves to completely neutralise any net charge density in the bulk liquid region, although we suggest that the preferential adsorption of ions on the liquid-vapour interface could provide a potential mechanism for manipulating nanobubble oscillations and translation, through externally applied electric fields.

Numerical investigation of particle separation in dense heterogeneous suspensions of soft particles using inertial microfluidics

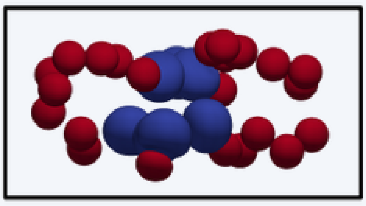
Benjamin Owen, Qi Zhou and Timm Kruger

Inertial microfluidics (Reynolds number of order (10–100)) emerged in the late 2000s and has contributed to wide applications in biomedical engineering [1]. One key application is label-free, high-throughput separation of target particles from heterogeneous suspensions (e.g. blood) for diagnosis purposes. Such separation relies on the migration of different particle types to their own focusing positions determined by the balance of inertial and drag forces within microchannel cross-sections. However, current inertial microfluidic devices are mostly limited to handling diluted samples (concentration 5%), due to a lack of understanding of the interplay of inertial particle migration and particle-particle interactions in dense suspensions, which are both experimentally and numerically challenging.

Using an in house 3D lattice-Boltzmann code with incorporated immersed-boundary and finite-element methods [2], we perform numerical simulations to elucidate the mechanisms for inertial particle motion in heterogeneous suspensions. A range of effects including flow inertia, sample concentration, mixture fraction and particle size/softness heterogeneity are investigated. We demonstrate that in homogeneous suspensions with increasing concentration, the hydrodynamic interactions between particles introduce substantial fluctuations to their motion and result in weakened focusing behaviour. For heterogeneous suspensions, the particle-particle interactions are more complex, affecting the lift velocities and collective motion of particles based on their absolute and relative size. Subject to appropriate particle confinement and mixture fraction, satisfactory focusing or enrichment of target particles can be achieved in dense suspensions (e.g. 20%) under high inertia conditions (Re=100). This work furthers our understanding of the inertial lift of particles in concentrated samples and provide evidence for the design and optimisation of microfluidic devices aimed at particle separation from dense heterogeneous suspensions.



(a) 5% concentration, t=0



(b) 5% concentration, t=end

Figure 1: Lateral locations of heterogeneous suspensions consisting of a 1:1 volume ratio of particles with confinement of 0.15 and 0.1 at 5% concentration and Re=50. Confinement is defined as the ratio between the particle diameter and the channel height. Left column shows the initial random distribution of particles while the right column shows the focused particles. Concentration is defined as the percentage of volume in the computational domain containing a particle.

[1] Zhou, J., Papautsky, I.: Size-dependent enrichment of leukocytes from undiluted whole blood using shear-induced diffusion. Lab. Chip 19:3416–3426, 2019.
[2] Owen, B. et al.: Lattice-Boltzmann modelling for inertial particle microfluidics applications - a tutorial review. Adv. Phys:X 8:1, 2023.

Paper ID CA-10 **Programme ID** 101

Impact of Surface Structure on Wetting Properties

Alexander Saal

Understanding the wetting of surfaces and the movement of drops is crucial in various applications, from microfluidics and drop dispensers to deicing, printing, and self-cleaning surfaces. The chemical properties and topological roughness of a surface determine the force required to move a drop, providing key insights for surface characterization. This understanding is essential for effective drop removal, whether on dirt-covered surfaces or in micro-scale applications and extends to biological contexts such as cell adhesion.

The shape and structure of a surface significantly influence its wetting properties. While most studies focus on sessile drops to determine wettability through contact angles, they often neglect the dynamic case. This includes measuring the forces acting on the drop, particularly on surfaces with complex structures such as overhanging pillar structures. In this study, we address this gap by calculating the forces based on surface geometry and comparing these calculations to direct measurements, thereby providing a more comprehensive description of such surfaces.

Our experiments involve moving a drop across different structured surfaces while directly measuring the contact angles and the forces acting on the drop. Our results demonstrate that the geometry of the micro-structure significantly affects the wetting properties. This research contributes to the design of surfaces with tailored wetting properties, potentially impacting a wide range of applications from industrial processes to biological systems.

Paper ID CF-1 **Programme ID** 109

Corner flows of viscoplastic fluids

Jesse Taylor-West

Many environmental and industrial fluids are viscoplastic; behaving like a solid at low stresses but flowing like a fluid at stresses above a threshold yield stress. The presence of a yield stress can often result in significant departures from Newtonian behaviour, exhibiting ‘plugged’ regions of unyielded fluid and the development of viscoplastic boundary layers (thin regions of highly sheared fluid) when the yield stress is large. In this talk I will present solutions for flows of viscoplastic fluids in corners and wedges, using asymptotics and numerics to demonstrate how the yield stress modulates the corresponding classical Newtonian similarity solutions. These flow configurations have application to transport and processing of viscoplastic fluids, and the quasi-analytical nature of the solutions can provide new bench-marks for computational codes.

Paper ID CF-2 **Programme ID** 42

Erosion of Sediment beds Using Impinging Jets: Application to Nuclear Waste Mixing

Ahmad Mohamadiyeh

Impinging jets serve as a pivotal mechanism for mixing and erosion employed across various industries, where one of the most relevant applications for this investigation is nuclear waste management. For example, within the facilities of Sellafield Ltd, impinging jets are used to erode and mix nuclear waste remotely in highly active nuclear waste storage tanks (HASTs). It is important to erode the nuclear particles and keep them in suspension inside the tanks to avoid the creation of highly tenacious sediment beds and radioactive hotspots. There are several factors that affect the intricate nature of erosion phenomena, including the properties of the fluid, the jet, and the properties of the bed. The effect of bed properties such as particle size distribution, density, particle shape, yield stress and cohesion on erosion is of particular interest given the wide range of nuclear particles available in the HASTs. Therefore, the critical aim of this study is to experimentally investigate the role of bed properties on erosion by impinging liquid jets. Initially, the erosion of spherical soda lime glass particles with different particle size distributions was studied as model cohesionless system, to determine the limits of size on analytical modelling using the critical erosion parameter (Ec). In addition, the erosion of fine barium sulphate and calcium carbonate suspensions were studied to represent cohesive nuclear waste simulants. The erosion clearance radius was tracked by a camera located underneath the tank and measured by using image analysis. Overall crater sizes and profiles were measured by implementing a combination of manual measurements and more advanced techniques, such as LiDAR and Ultrasonic Velocity Profiling (UVP). The UVP was mounted on a moving traverse and tracked profiles of both the static and dynamic erosion of opaque systems with a high load of suspended particles. Data was fitted to both cohesionless and cohesive erosion models to understand the effects of particle properties on scale independent predictions. In conclusion, studying these fundamentals of impinging jet erosion aids the nuclear industry in further understanding and optimizing their post operational clean out.

Paper ID CF-3 **Programme ID** 48

Multi-scale modelling of blood rheology in sickle cell disease

Freya C Bull

Sickle cell disease (SCD) is a haematological disorder, caused by a genetic mutation, in which mutant haemoglobin molecules can polymerise under low-oxygen conditions, altering the biophysical properties of the red blood cells. These cell-level differences then result in changes in the whole-blood rheology -- and those rheological properties are in turn linked to the pathophysiology of SCD.

Ongoing experimental work indicates that SCD blood exhibits increased frictional and viscous resistances to flow. Our work investigates the contribution of elevated red blood cell friction to the whole-blood rheology, utilising mathematical modelling and numerical simulation to develop descriptions of cell-cell interactions within blood flow.

Paper ID CF-4 **Programme ID** 52

Effective viscosity model for suspension of spherical particles in inelastic shear-thinning fluid matrices

Federico Peruzzini, Jonathan M. Dodds, Christopher Cunliffe, Henry C.-H. Ng and Robert J. Poole

There are several models available in the literature that account for the complex nature of suspensions with non-Newtonian matrices. Here we propose a viscosity model for non-Newtonian suspensions, consisting of non-Brownian spherical particles and a shear-thinning matrix.

Aqueous solutions of xanthan gum (XG) with and without calcium carbonate particles (CC) were tested. We prepared mixtures with tap water at XG wt% in the semidilute regime, and CC vol% between 5% and 30%. Two different mixing protocols were used, differing in whether or not they were diluted with the stock XG solution.

Base xanthan gum viscosity was collapsed into a master curve. The shear-thinning behaviour is described as a function of polymer concentration by scaling laws for the four Cross model parameters, zero-shear viscosity $\eta 0$, infinite viscosity $\eta\infty$, consistency index K and viscosity index m. The viscosity of protocol 1 mixtures is unexpectedly low, regardless of solids content. This behaviour was attributed to the mixing procedure and named the "dilution effect". Protocol 2 was developed to correct the mixing procedure. A predictive model was formulated based on the work of Peysson et al (2006) [1]. An equation for effective viscosity is derived through the Cross model and used to calculate the relative viscosity of the suspensions. The results show the relative viscosity to be roughly independent of the shear rate. A collapse of the average relative viscosities was achieved using the Pal (2020) [2] model for Newtonian suspensions.

We present a viscosity model of non-Newtonian suspensions with non-Brownian particles. The Cross equation and scaling laws can be used to calculate the viscosity of xanthan gum solutions in the semidilute regime. Self-dilution is highlighted as a critical factor in mixing. The developed model predicts viscosity over a broad range of shear rates, especially at shear rates where the power model is not applicable to the data.

[1] Peysson, Y., Aubry, T., Moan, M.: Phenomenological approach of the effective viscosity of hard sphere suspensions in shear-thinning media. Applied Rheology 16, 145–151 (2006) https://doi.org/10.1515/arh-2006-0010

[2] Pal, R.: New generalized viscosity model for non-colloidal suspensions and emulsions. Fluids 5, 150 (2020) https://doi.org/10.3390/fluids5030150

Paper ID CF-5 **Programme ID** 53

Effects of viscoelasticity and shear-thinning on the mixing performance of a T-channel geometry

R. J. Hill, M. Davoodi, C.P. Fonte and R. J. Poole

The T-channel is known to be one of the most efficient microfluidic mixing geometries. For a three dimensional T-channel consisting of two opposing planar inlet channels that join and turn through 90°, it is well known that, for a Newtonian fluid above a critical Reynolds number, we encounter an instability where the flow breaks symmetry but remains steady[1]. The same instability is also observed in the presence of shear-thinning[2] and viscoelasticity[3]. However, both properties influence the degree of inertia required for the instability to occur. For a Newtonian fluid, the mixing quality is significantly increased when the asymmetric engulfment regime is reached[4], compared to the earlier symmetric regimes. Hence, understanding the instability in complex fluids is of utmost importance regarding mixing quality.

In this work, we studied the effects of viscoelasticity and shear-thinning on the mixing efficiency of a T-channel geometry by simulating the flow of fluids described by three different viscoelastic constitutive models and a fluid described by a generalised Newtonian viscosity model, alongside the Newtonian benchmark. The models considered are Oldroyd-B, Oldroyd-A, Giesekus and Carreau. For the Giesekus and Carreau constitutive equations, the parameters are chosen to match the shear viscosity across the two models.

Irrespective of the fluid properties, all models considered undergo a change from a symmetric to an asymmetric flow via a supercritical bifurcation - which is likely dictated by the magnitude of the secondary 'Dean' flow. We show that whilst a T-channel geometry proves to be an efficient mixer regardless of rheology, both viscoelasticity and shear-thinning result in the degradation of the mixing performance.

1 N. Kockmann, et al., Proc. SPIE. 4982, 319 (2003).
2 R.J. Poole, et al., Chem. Eng. Sci. 104, 389 (2013).
3 R.J. Poole, et al., Procedia Eng. 79, 28 (2014).
4 S. Thomas, T.A. Ameel, Experiments in Fluids. 49, 1231 (2010).

Paper ID CF-6 **Programme ID** 82

Towards modelling gas migration through dense sediment suspensions: LBM-DEM-FSLBM

Isabel F. Latimer

Particle laden flows are ubiquitous to the world around us, commonly occurring in environmental and industrial settings such as ocean beds and industrial-scale chemical reactors. Advances in parallel computing capabilities have allowed for large-scale numerical modelling of multi-phase flows. Enhancing our understanding of the rheological behaviour of these multi-phase flows is important for practical and industrial applications. This research focuses on modelling industrial scenarios within Sellafield, such as ion exchange beds and acute gas release from nuclear waste slurries. A lattice Boltzmann approach using the opensource framework waLBerla (Erlangen, Germany) is chosen due to its ability to handle complex boundaries, incorporate microscopic interactions and the parallelizability of the algorithm. This presentation will detail the initial validation studies and current progress on the development of a coupled lattice Boltzmann - discrete element method - free surface (volume of fluid) simulation of multiphase sediment beds. The ultimate aim of this research is to develop a computational model to capture how the presence of gas influences the rheology of sediment beds particularly during gas migration and retention. Contextually, this is important when considering the release of hydrogen from nuclear waste slurries or methane from the sea bed. The main motivator for this research is the need to mitigate against periodic acute release of hydrogen from nuclear legacy waste sludge at Sellafield Ltd, UK to ensure safety during decommissioning of the facility.

Paper ID CF-7 **Programme ID** 90

A numerical study of stress distribution around a viscoelastic cut

Andrea Sendula, Sam Pegler and Oliver Harlen

Viscoelastic fluids are non-Newtonian materials that exhibit both viscous and elastic characteristic. This results in unique phenomena that is often not expected in fluid like substances. One of these is fracturing, which has been shown to occur in viscoelastic gels (Tabuteau et al., 2011). During pendant drop experiments, these gels rupture corresponding to brittle failure described by elastic fracture theory.

This project seeks to explore this behaviour through understanding the distribution of stresses around the point of fracture and the role viscoelasticity plays. To achieve this, we model the flow around a viscoelastic "cut" with respect to governing parameters, such as the radius of curvature of the cut, the Navier slip-length describing the behaviour on the surface, and the viscoelastic parameters. We explore the effect of these key values on the size and distribution of stresses along the cut surface.

The conclusions of this study will present the effects of both the viscous and elastic components acting upon the stresses along the surface, which will contribute to a deeper understanding of how viscoelastic fractures propagate and how they differ from elastic materials.

Paper ID CF-8 **Programme ID** 39

A minimal continuum model of clogging in spatio-temporally varying channels

Duncan Hewitt and Philip Pearce

Particle suspensions in confined geometries exhibit rich dynamics, including flowing, jamming, and clogging. It has been observed that jamming and clogging in particular are promoted by variations in channel geometry or fluid material properties - such variations are often present in industrial systems (e.g. local confinements) and biological systems (e.g. stiffening of red blood cells in deoxygenated conditions in sickle cell disease). The aim of this talk is to shed light on the macroscopic dynamics of particulate suspensions in these conditions. To this end, we present a continuum two-phase model of particle suspensions based on granular rheology that accounts for spatio-temporally varying material properties or channel geometries. The model comprises a continuous particle phase which advects with flow and has material properties dependent on the particle volume fraction, and a suspending fluid which flows through the particle phase obeying Darcy's law. We solve the system using a finite-volume method and simulate the evolution of an initially uniform particle density. We find that varying material properties and varying geometry can induce heterogeneity in particle volume fraction. We are able to show the emergence of high and low particle density regions in volume-driven flows. These results clarify how spatial variation in material and channel properties can contribute to clogging of particle suspensions.

Paper ID CS-2 **Programme ID** 26

Experimental Investigation of the Settling Behavior of Perforated Thin Disks

Jörg T Sommerau

Freely rising and settling bodies is a popular and important type of problem in fluid dynamics. A classic example is the settling behavior of thin solid disks which tend to undergo quite dierent modes of settling depending on disk diameter, density and fluid viscosity and which has been extensively studied and documented in the past. Less studied but being investigated more recently is the dynamics of annular and perforated disks which are relevant to various engineering applications, an example of which are the design and control of microfliers. To combine and study annular and perforated geometry for solid thin disks, a custom experimental setup is developed and used to conduct experiments. The setup consists of a large volume of water in which the perforated disks are dropped from a device specially developed for releasing perforated disks and two cameras in an orthogonal arrangement recording the descent which is reconstructed in three dimensions in the post-processing. Moreover, a model to generate the perforation patterns is developed and implemented and the disks are then produced from acrylic glass using laser cutting. The experiment is conducted in two stages, the first of which is a parameter space exploration in terms of void fraction and void fraction distribution and the second one is an investigation of the influence of permeability for a given constant void fraction. It is found that the void fraction increases descent velocity and dampens oscillations. Moreover, the void fraction distribution has an even stronger influence on the settling velocity while not significantly affecting oscillatory motion. For a given constant void fraction, varying the permeability by varying the hole to disk diameter ratio also influences settling speed and how oscillatory the descent is. Additionally, it is shown that there exists a diameter ratio which globally maximises descent velocity while simultaneously minimizing oscillatory motion.

Paper ID CS-3 **Programme ID** 56

Settling rates of non-spherical microplastic particles in a quiescent fluid using numerical simulation

Abhimanyu Gaur, Lee Mortimer, David Hodgson, Gareth Keevil, Jeff Peakall and Mike Fairweather

Millions of tons of microplastics have entered the ocean comprising various shapes, sizes, and densities. The dissemination of these polymer particles has highlighted concerns about their environmental effects and potential impacts on marine ecosystems. Settling rate is a key parameter in studies of the vertical distribution of microplastics in the ocean. To generate insight into the transport mechanism requires studies using both numerical computations, and physical experiments in a controlled environment. The complexity in understanding the settling rates of these microplastics is considered by choosing suitable density ratios, shapes and fluid properties. Nek5000 has been used in this study, which is an open-source spectral element based, computational fluid dynamics code. The settling rates of particles were studied using direct numerical simulations, based on Nek5000, together with an immersed boundary method to simulate the solid particles. These techniques can precisely capture the trajectories of particles and hydrodynamic effects on the flow past the particles. Disk-shaped particles with circular and bevelled edges have been investigated for a range of particle Reynolds numbers. The settling histories of disks are found to be sensitive to initial orientations, size, density ratio, shape of the disk edges and fluid properties. The disks with sharp edges show lower settling rates compared to their bevelled counterparts due to drag imposed by the sharp corners. Also, the settling dynamics are orientation dependent. Future goals are to conduct simulations for the settling rates of other non-spherical spheroidal particles. Also, full-scale experiments using a Shake-the-Box setup (i.e. 3D particle tracking velocimetry/Lagrangian particle tracking) will be used to explore hydrodynamics effects in the vicinity of the settling particles.

Paper ID CS-4 **Programme ID** 68

Physics Informed Kriging for Particle Tracking Velocimetry

John M. Lawson

The widespread adoption of particle tracking techniques has generated the need to reconstruct velocity fields on regular grids from particle tracks, which provide velocity (and acceleration) data at randomly distributed locations. Several data assimilation techniques have emerged, notably radial basis function interpolation (e.g. Vortex In Cell) and regression spline (e.g. FlowFit) based methods, which enforce mass and momentum conservation to improve the reconstruction accuracy. Outstanding problems in this area lie within uncertainty quantification and improving accuracy. Kriging interpolation is a well-established technique for the interpolation of irregularly sampled data, which has received relatively little attention from the PIV community. Physics Informed Kriging can provide rigorous estimates of uncertainty and leverages statistical information as well as known statistical symmetries (e.g. homogeneity, isotropy) to quantify and reduce reconstruction uncertainty, in addition to incorporating conservation laws, boundary conditions and other constraints into the solution.

In this work, we demonstrate how Physics Informed Kriging can be applied to reconstruct dense velocity fields from even very sparse particle tracking data. As an input, we provide velocity and acceleration data from particle tracks, and as an output we obtain dense reconstructions of the velocity field, with quantified uncertainty, satisfying conservation laws and boundary conditions. As a useful byproduct, we also obtain the mean velocity field, two-point correlation functions and proper orthogonal decomposition modes directly from the particle tracking data, which also satisfy boundary conditions, conservation laws and statistical symmetries. The approach is tested upon synthetic particle tracking data from homogeneous isotropic turbulence and turbulent channel flow for different seeding concentrations and measurement noise levels. The performance is compared to the Vortex In Cell method.

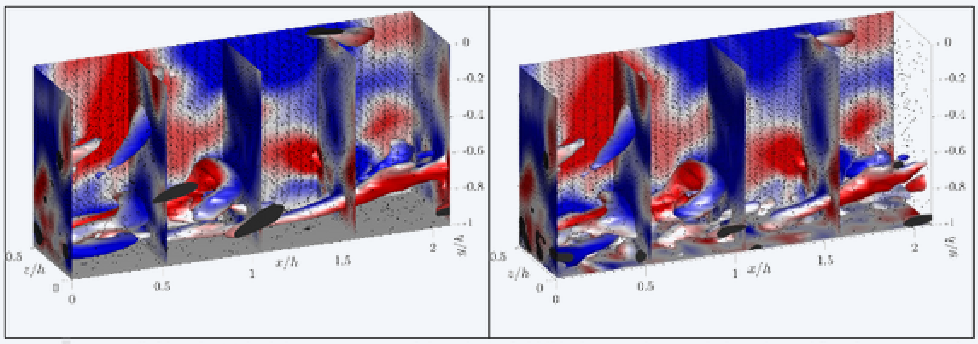


Figure 1: Kriging (left) and VIC+ (right) reconstruction of the velocity field (black vectors) in a turbulent channel flow from scattered velocity data (black dots). Isosurfaces of constant Q-criterion highlight hairpin vortices, coloured by wall-normal velocity component (red: up, blue: down). Near-wall features, poorly captured by VIC+, are coherently reconstructed using Physics Informed Kriging.

Paper ID EX-1 **Programme ID** 51

Flow-Induced Vibration of Porous Plates in the yaw direction

Cicolin, M., Carlson, D. and Ganapathisubramani, B.

The flow past a solid flat plate is characterised by the formation and shedding of vortices. This process is the onset of most fluid-structure interaction problems in engineering, as it creates an unsteady force that can cause structural fatigue and failure, fuel consumption, noise, and other effects. Introducing permeability to the model, making it porous, can delay the formation or suppress vortex shedding, mitigating some undesired effects, as it changes the vortical structure and its interaction with the body.

Recent studies with fixed porous plates have shown that the classical Von Karman vortex street past a flat plate can be extinguished if the porosity ratio \beta, defined here as the ratio between the open area of the body and the total area, is greater than 30%. Even for lower values, the porosity effect shifts the vortex shedding process downstream, weakening the unsteady forces, and inducing a fall in the mean force component too (Cicolin et al, 2024).

In this work, we add one degree of freedom to the plate, expanding the investigation towards a fluid-structure interaction context, as merely shifting the vortex shedding point may not be sufficient to prevent the vibration of a flexibly mounted structure.

We report preliminary results for a new experimental campaign, intended to evaluate the porosity effects at suppressing FIV in the yaw direction. The experiments were carried out in a water channel with several plates with constant width D, varying the porosity from 5% to 40%, whereas the solid plate (0%) is the reference case. Reynolds number based on D is within the moderate range, varying from 10,000 to 40,000. The results are presented using flow visualisation with optical techniques, amplitude of movement, and force measurements.

Paper ID EX-2 **Programme ID** 104

Drag of ammonite shell models

Bappa Mitra, Marc Desmulliez and Ignazio Maria Viola

Ammonoids are extinct spiral-shelled cephalopods (Fig. 1a,b), such as squids, and their fossils are used extensively as proxies for biostratigraphy and animal behaviour in response to past climate changes. Most are thought to have been good swimmers, but the geometric features associated with low flow resistance remain uncertain. In this work, we present the measurement of the drag coefficient of 36 different 3D printed ammonite shell models in a uniform and constant stream. These shells, prepared using morphometric variables as described in Figure 1c, have a maximum height of 60 mm. The tested Reynolds number ranges from 13757 to 18343. We found that the drag coefficients of circular aperture shells (S=1) increase with the increasing shell expansion ratio (W), and also with the increase of umbilical exposure (D). However, unlike for shells with circular apertures, the trend reverses for inflated (S=1.5) and over-inflated apertures (S=2) when umbilical exposure increases. While the dataset of circular aperture shells has been used in understanding the ammonioid behaviour (Ritterbush, 2016), this new dataset complementing the existing literature may contribute to providing additional insights into the theory of evolution,trade-offs, and the objectives pursued by the ammonoids.

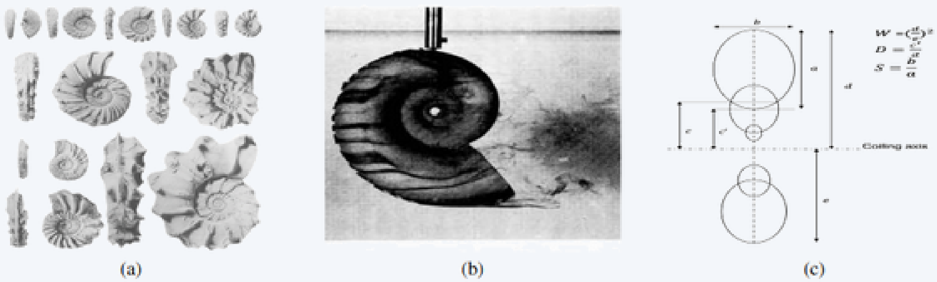


Figure 1: a) Morphological variations among intraspecific ammonite shells (Kennedy, 1989) ; b) Flow around an ammonite replica (Chamberlain, 1976); c) Morphometric variables for an ammonite shell

Chamberlain, J. A. (1976). Flow patterns and drag coefficients of cephalopod shells. Kennedy, W. (1989). Thoughts on the evolution and extinction of cretaceous ammonites. Proceedings of the Geologists' Association, 100(3), 251–279.

Ritterbush, K. A. (2016). Interpreting drag consequences of ammonoid shells by comparing studies in westermann morphospace. Swiss Journal of Palaeontology, 135, 125–138.

Paper ID EX-3 **Programme ID** 54

Internal flow fields of structured porous media subject to a grating flow

Elias J. G. Arcondoulis and Thomas Geyer

Grating flows over porous materials have gained attention in recent years, as porous materials are widely studied as an effective passive flow control technique to alleviate bluff body vortex shedding and aeroacoustic trailing edge noise. The porous material influences the development of the boundary layer over its surface yet how the characteristics of the porous material directly alter the boundary layer is still unclear. It has been shown via numerical simulation that the flow tangential to the surface should enter the porous media to facilitate communication between the upper and lower surfaces of the trailing edge. However, revealing this flow penetration via experiment poses a significant challenge. Structured porous media (such as lattice structures) have been used to replace typical randomised porous media, yielding similar performance; yet, they are customisable in their porous properties, such as porosity and permeability. Two structured porous grating flows were tested in a wind tunnel, using circular and square shaped pores, with a freestream velocity of 19.1 m/s over a block dimensions of length 150 mm and thickness 80 mm. A two-dimensional hot-wire anemometry probe was placed inside the structured porous media during wind tunnel experiments, to record the flow field subject to tangential flow. These experiments were repeated by rotating the probe by 90 degrees, to quantify flow fields in three dimensions. These recently conducted experiments reveal that the grating flow indeed induces flow penetration into the porous structure and is dependent on the geometry of the porous media. The influence of the porous structure on the turbulence developed within the porous structure and an estimate of the mass flux that enters the porous media is currently being quantified and will also be presented at the conference.

Paper ID EX-4 **Programme ID** 57

Wave interaction with a structured porous breakwater

Elias J. G. Arcondoulis, Richard Porter, Andrew Hogg and Rory Bingham

Climate change is causing more frequent and intense storms that are, and will continue, to erode and damage UK coastline infrastructure and impact coastal communities. Solid and rubble breakwaters are widely used to diminish wave energy and protect sensitive coastal areas yet these intrusive structures disrupt natural coastal flow process, such as longshore drift that lead to significant sand dredging costs, and impede on tidal and sediment passage in salt marshes. Porous breakwaters have been investigated as an alternative to solid breakwaters that allow the transmission of sediment, while also reducing the transmission of wave energy. A structured porous breakwater is designed and will be tested and validated in a 600 mm wide wave flume in the COAST lab at the University of Plymouth from June 25-27. Wave heights upstream and downstream of the breakwater will be recorded for a series of wavenumbers, and will be compared against an analytical formulation of the transmitted and reflected wave height as a function of wavenumber. The wave-interaction of the structured breakwater will also be compared against a solid breakwater with the same dimensions. Time-resolved particle image velocimetry will also be used to visualise and quantify the turbulent flow structures exiting the porous structure when subject to a current flow.

Paper ID EX-5 **Programme ID** 67

The effect of axial separation distance between coaxial counter-rotating rotors

Hulya Biler

As the drone delivery market expands, the demand for higher payload capacity drives the adoption of compact drone architectures, including co-axial propellers. While overlapping propellers offer significant advantages, they also pose challenges. Among these concerns is the noise generated by these vehicles, which affects public acceptance and drone adoption. To design efficient and quiet multi-rotor propulsion systems, a deeper understanding of aerodynamic noise sources is crucial. In this study, we experimentally explore the impact of axial separation distance on the interaction between coaxial counter-rotating propellers during hover conditions. Our measurements took place in a semi-anechoic room at the University of Southampton. Using a modular rig equipped with two two-bladed 16-inch diameter rotors, we adjusted the axial separation distance. Throughout the experiments, we maintained a constant total thrust of 16 N and zero total torque. Our investigation focused on how separation distance affects the aerodynamic response of the system. We employed force and phase-locked (both upstream and downstream rotors) high-resolution low-speed Particle Image Velocimetry (PIV) measurements. Notably, we found that axial separation significantly influences wake behavior. The final presentation will include a detailed analysis of tip vortex evolution. Leveraging the Γ_2 criteria, we will pinpoint the tip vortex positions and examine their meandering across various separation distances. Additionally, we will thoroughly analyze the strength and velocity profiles of these tip vortices. Finally, we will compare rotor wakes with the Kocurek-Tangler wake model to quantify the effect of the added pressure gradient due to the second rotor. This research contributes valuable insights for the development of quieter and more efficient multi-rotor drone propulsion systems, ultimately enhancing their acceptance and integration into various applications.

Paper ID EX-6 **Programme ID** 71

PIV-PLIF experiments of pollutant dispersion over a scaled urban model

Tomos Rich and Dr Christina Vanderwel

Air pollution is a significant problem within urban environments, with considerable impact on population health. Accurate modelling of air pollution requires an understanding of the processes that transport scalar concentrations in urban environments. To this end, an investigation into scalar dispersion over a 3D-printed model of the city of Southampton was carried out.

This model was made using Ordnance Survey data of 1km2 of Southampton's city centre and was printed at a scale of 1:1000. This model was created from a section of the city that includes some of the port area, and has been designed to simulate the predominant wind direction of an onshore south-south-westerly.

Experimental measurements were made in the University of Southampton's Recirculating Water Tunnel. All experiments were carried out with an incoming flow depth of 0.65m and a freestream flow speed of 0.6 m/s. Upstream flow conditioning was used to simulate an atmospheric boundary layer with a depth at the test section of 0.3 m. Using the maximum building height of 45 mm as the length-scale, the Reynolds number was 27,000.

To experimentally simulate an airborne pollutant release at the docks, a scalar ground-level point source was added 50 mm upstream of the city model. With this experimental setup simultaneous particle image velocimetry (PIV) and planar laser induced fluorescence (PLIF) measurements were taken. The PLIF data was processed using a custom Matlab code that has been published on Github.

This dataset has allowed us to document how the mean plume spreading rate and direction are influenced by the city model. We found that the scalar plume is being deflected away from the mean flow direction by the alignment of street canyons in the model. The relative strengths of the vertical advective and turbulent fluxes have been compared and it has been found that both advective and turbulent processes are significant in the vertical ventilation of the scalar at most measurement slices.

Paper ID EX-7 **Programme ID** 75

Practical experiences from testing a renewable energy turbine prototype in a tidal estuary

Ian Masters, Ali Esmaeili, Iestyn Evans, Deepak George, David Glasby, Jose Horrillo-Caraballo, Thomas Lake, Michael Togneri and Alison J Williams

Reflections on the design, build and operation of a 3 meter diameter tidal turbine in a tidal estuary in Pembrokeshire, Wales, UK in January 2024. The device is 3 meters in diameter and is positioned below a floating barge. The test site at Warrior Way on the Cleddau river is part of the Marine Energy Test Area (META), a project run by Marine Energy Wales. The turbine is open source, including technical reports and SolidWorks drawings [1]. It is based on a design concept that has been previously reported [2].
Six separate records of velocity were taken, four seabed mounted ADCP units, one ADCP downwards pointing on the barge and an ADV, which was recording at 10Hz and synchronized with the turbine performance data. The primary performance measurement was four pairs of strain gauges measuring blade root bending moments in thrust and torque directions. These were then used to obtain the usual power and thrust coefficients.
Mechanical and operational issues of the deployment are described. Our turbine is designed to operate in remote communities and has a purely mechanical system offshore. The power take off is a water pump and this was a source of issues throughout the test. Test filling of the system revealed that the sliding seal in the front of the turbine had failed since the previous test 12 months earlier. We have also learnt that this system is difficult to align and keep the pump shaft straight through the whole rotation.
Overall, a successful test was conducted with several hours of operational results.

[1] https://zenodo.org/doi/10.5281/zenodo.8082024
[2] https://doi.org/10.1680/jener.21.00101

Paper ID EX-8 **Programme ID** 84

Measurements of air mixing and stratification at full building scale

Filipa Adzic, Oliver Wild and Liora Malki-Epshtein

Airflow properties such as turbulent mixing and buoyancy are difficult to dynamically scale and can be missed in reduced-size models. Controlled Active Ventilation Environment (CAVE) laboratory provides a unique opportunity to test a variety of indoor air and fluid mechanics applications at full building scale. Due to the laboratory floor area of 205 m2 and a height of 9.54 m, dynamic scalability of acting forces is not a constraint and producing experimental data sets of different ventilation scenarios is useful for theoretical fluid mechanics, computational fluid dynamic validation and various engineering applications A variety of ventilation scenarios can be achieved in the laboratory through 24 supplies and 8 extracts, placed at low and high levels (1.75 and 9.2 m respectively). A set of stratified and wellmixed ventilation scenarios were tested in the laboratory with measurements taken before, during and after the ventilation scenarios were operated. Measurements taken include air temperatures, CO2 concentrations and relative humidity at over 90 points, surface temperatures of walls, ceiling and floor at 40 and air velocities at 29 points. These measurements were used to obtain the non-dimensional Stratification, Richardson, Rayleigh and Raynolds numbers for a set of operated ventilation scenarios.

Paper ID EX-9 **Programme ID** 91

An experimental study on Moving Surface Boundary-Layer Control systems

Jan W Modrzynski

Moving Surface Boundary-Layer control system (MSBC) in the form of a leading-edge rotating cylinder was proved to be exceptionally effective in delaying the stall and maximizing the Cl/Cd ratio. This study explores the effects of rotating leading edges on a Joukowsky 15% symmetric aerofoil. Various rotating leading-edge cross-section designs were employed to control the flow injection rate, which enabled a better understanding of the underlying mechanisms of performance improvement. Particle image velocimetry, steady surface pressure and force measurements were conducted to quantify the effect of the flow control technique. The experiments were performed for speed ratios Ω of 1, 2 and 4, i.e. the ratio of cylinder linear velocity to the free stream velocity, at free stream velocities of 5 m/s and 10 m/s in the subsonic closed-circuit wind tunnel at Swansea University. Performing PIV experiments with two cameras allowed for a broad set of data with both high and medium spatial resolutions, which, together with surface and wake pressure measurements, enabled a detailed performance evaluation. The best cases showed stall angle improvements by over 30 degrees compared to the unmodified case, accompanied by a significant increase in the maximum Cl. Scooped designs improved stall angle and Cl at higher speed ratios. However, the stall angle and respective Cl at Ω=0 were lowered due to the sharp edges and deep grooves. Most designs at Ω=1 performed similarly to the baseline aerofoil, only showing improved results at higher speed ratios.

Paper ID EX-10 **Programme ID** 21

Advancing Spray Analysis: Combining PDA and High-Speed Velocimetry

Zuhaib Nissar, Oyuna Rybdylova, Steven Begg and Guillaume de Sercey

Understanding local droplet size and velocity distributions is crucial for enhancing spray-based technologies. Phase Doppler Anemometry (PDA) measures droplet sizes and velocities simultaneously, yet its application is typically limited to the far-field, dilute region of a spray, where droplets are smaller and more spherical compared to those in the breakup region. To complement PDA, a new method involving high-speed shadowgraphy has been developed. This method integrates droplet and ligament detection with a multi-image stitching algorithm to analyse the breakup region, determining local droplet size and velocity distributions in a steady-state, continuous, flat fan water spray. By collating non-overlapping images, a composite spray image covering the entire spray region was created. To remove artefacts along the borders, the data along the 'stitches' was reconstructed using four grids, each shifted vertically and/or horizontally by half the height and/or width of the field of view. Altogether, 1,200 composite images were generated for each grid at a frame capture rate of 84 kHz. Droplet sizes from 10 µm and greater were identified in the images, with PDA extending this range down to 1-2 µm. Particle Image Velocimetry was then applied to the temporally resolved composite images to measure droplet velocities. The droplet characteristics at flow rates of 4, 4.5, and 5 kg/h were analysed and compared using both the new method and PDA. The findings showed good qualitative agreement between the two methods, demonstrating consistent trends, although quantitative discrepancies were noted. Additionally, velocity measurements showed up to a 20% difference between the two methods. These differences may stem from the exclusion of non-spherical droplets or droplets approaching the probe volume at an angle in PDA. In imaging, lower resolution and droplets outside the depth of field, which move slowly, may lead to bias towards lower velocities during cross-correlation.

Paper ID EX-11 **Programme ID** 22

Design and Validation of a Low-Deadtime Stopped-Flow Device for High-Resolution Reaction Monitoring

Mostafa Soroor, Nik Kapur, Arwen Tyler and Nick Terrill

Stopped-flow devices are extensively employed for monitoring dynamic reactions with high temporal resolution. This research focuses on designing a “beamline-friendly” stopped-flow device that maintains transparency to the beam and minimizes sample usage while achieving very low deadtime. Additional constraints included material compatibility with the chemicals used within the device and transparency to the beam used to illuminate the sample. CFD, using COMSOL Multiphysics, was utilized to analyse 24 geometries of a vortex T-mixer using Design of Experiment, at fixed flow rates to identify the optimal flow geometry with giving the highest mixing index (maximum 0.96) and shortest deadtime. Additional simulations were carried out to ensure optimal heater and sensor placement within the flow cell. Utilizing these results, the device was constructed, incorporating a control board managing the feed pumps and heating of the cell, and including TTL triggering capability to ensure synchronization with data collection of the beamline. Experiments to verify the simulation findings and examine device capabilities were conducted, including fluorescence (reduction of DCIP with Ascorbic Acid) and KBr-water mixing tests, giving reasonable deadtimes of 14 ms and 4 ms at flow rates of 35 ml/min and 10 ml/min, respectively. Good agreement was seen between experimentally determined values and theoretical predictions. This stopped-flow device was used to facilitate studies on dynamic reaction processes in biological systems, notably contributing to the analysis of AdhE spirosome length in enterohaemorrhagic Escherichia coli. The adaptation of the vortex T-mixer into a stopped-flow device enables the use of low-volume samples (25 µl) with open-source instrumentation suitable for any synchrotron facility. These developments contribute to the field of dynamic reaction monitoring, offering more precise and efficient techniques that are adaptable to various research environments.

Paper ID FD-1 **Programme ID** 14

A Homotopy-Based Modeling Approach for Poiseuille Flow in Fully-Filled Sewer Pipes with Egg-Shaped and Horseshoe-Shaped Cross-Sections

André Lopes

This work presents a novel analytical solution for laminar flow through fully-filled pipes with egg-shaped and horseshoe-shaped geometries, employing a homotopy-based approach. Alongside the circular section, these profiles are commonly employed in sewer systems and have predominantly been studied in the literature using experimental, empirical, or numerical methods. Our results are validated by comparing the predicted volumetric flow rate and the product of the friction factor and Reynolds number with those obtained using the finite element method.

Paper ID FD-2 **Programme ID** 16

A CFD Performance Study of Novel Vortex Bladeless Wind Turbines

Reece Luetchford, Sonia Melendi-Espina and Jack Panter

Vortex Bladeless Wind Turbines (BWTs) are an innovative upcoming renewable technology that exploits the vortex-induced vibration phenomenon to extract energy from the wind. The concept design consists of a simple two-segment mast and base structure containing a vibration-based energy harvesting system, which inherently allows for a wide operating range of wind speeds in comparison to traditional horizontal axis wind turbines. As a result, this supports a more efficient functionality at both low and high wind extremes and significantly reduces vulnerability to the environment. Despite the recent advancements in BWT research, there is still progress to be made on understanding the aerodynamic performance and feasibility of the concept before commercial production is to be considered. This study explores the performance of BWTs using ANSYS fluent, given its advanced CFD modelling capabilities, to capture vortex shedding and gather lift force data at wind speeds of 1- 20 m/s. Transient 3D CFD performance models provided the necessary data to analyse power output, from which the annual energy output was projected considering wind speed distributions at a given location. Findings show promising performance results overall, indicating that a 5 m tall BWT model displays potential as a feasible renewable energy source. With performance models projecting an annual energy output of 327.0 kWh at Cp efficiencies ranging from 2-13%. In comparison to solar PV, a technology of similar scale and cost, results suggest that BWTs offer a largely substandard energy performance. However, paired with the proposed benefits of less required maintenance and low environmental vulnerability, BWTs offer potential as an alternative method of small-scale renewable energy generation.

Paper ID FD-3 **Programme ID** 27

Study on Close-Coupled Gas Atomisation (CCGA) Process : CFD modelling via discrete-phase model (DPM)

Reece Luetchford, Sonia Melendi-Espina and Jack Panter

Over the past decade, powder metallurgy has gained immense popularity within the Additive Layer Manufacturing (ALM) industry owing to its extensive applications, and its current demand is all-time high. Gas atomisation process is the most viable method to produce such metal powders. However, due to the complicated and unpredictable nature of the process, due to the rapid interaction between gas and the melt, the background physics of the process is hardly understood inspite of the use of high-speed cameras. This randomness and instability in the process leads to increased particle size and particle-size distribution (PSD), resulting in a highly inefficient process. Consequently, a huge amount of energy and capital is lost, where the yield is $< 35\%$, and the rest of the out-of-specification powders are sent for re-melt, and twice the energy is used to recompress the inert gas as used for the process.
In this research, a CFD analysis of the gas atomisation process is carried out to understand the instabilities present between the deep sub-ambient pressure formed in front of the melt nozzle tip known as aspiration pressure and the melt flow rate since it significantly influences the particle size and PSD. User-Defined Functions (UDFs) were employed to couple the aspiration pressure to the melt flow rate, to understand the extent of instability in the process. The gas atomisation process is modelled as a two-phase flow with primary phase as Argon, and the secondary phase is modelled as inert constant diameter DPM particles, indicating the melt. This research confirms that at elevated melt temperatures, increased activity of melt instability is observed, reflecting the randomness in the physical process. Additionally, this research identified a new form of gas flow-field fluctuation that was not observed at lower melt temperatures.

[1] MOTAMAN, S., MULLIS, A. M., COCHRANE, R. F., MCCARTHY, I. N. & BORMAN, D. J. 2013. Numerical and experimental modelling of back stream flow during close-coupled gas atomization. Computers & Fluids, 88, 1-10.

Paper ID FD-4 **Programme ID** 81

Steady Euler flows in a channel with piecewise constant vorticity

Karsten Matthies, Jonathan Sewell and Miles H. Wheeler

We consider a two-dimensional, two-layer, incompressible, steady flow, with vorticity which is constant in each layer, in an infinite channel with rigid walls. The velocity is continuous across the interface, there is no surface tension or difference in density between the two layers, and the flow is inviscid. Unlike in previous studies, we consider solutions which are localised perturbations rather than periodic or quasi-periodic perturbations of a background shear flow. We rigorously construct a curve of exact solutions and give the leading order terms in an asymptotic expansion. We also give a thorough qualitative description of the fluid particle paths, which can include stagnation points, critical layers, and streamlines which meet the boundary.
We also present numerics for the periodic case, which allows us to move beyond the small-amplitude regime.

Paper ID FD-5 **Programme ID** 113

Integrating Mean-Line and CFD Approaches for the Redesign of a Single-Stage Micro Gas Turbine Stator-Rotor System

Qiming Yu and Robert Howell

The objective of this study was to aerodynamically analyse and redesign the Wren100 micro gas turbine (MGT) stator-rotor blade flow passages supplied by Turbine Power Solutions Ltd., aiming for higher thrust and isentropic efficiency using a combined approach of mean-line analysis and CFD. The MGT stage features one nozzle guide vane (NGV) and one rotor disk with blade heights of less than 20mm. Given the small blade sizes, the flow in MGTs typically exhibits low Reynolds numbers, indicating a significant presence of transitional flow within the blade passages. After reverse-engineering the blade components, 3D discrete models were generated and subjected to CFD simulations using ANSYS based on boundary conditions from Wren100 engine tests. With the Wren100 MGT mid-span profile, the RANS 4-equation transitional SST and LES WALE models were both validated based on a custom-built low-speed wind tunnel linear cascade rig, confirming their adequacy for analysing the MGT stator-rotor system. In this research, design enhancements based on mean-line predictions and parametric studies provided critical insights into the complex flow dynamics within microturbomachinery. Under 120,000RPM, it was found that the secondary loss contributed the most to the total blade loss.

For the Wren100 stator, modifications such as reducing the number of vanes, halving trailing edge thickness, and increasing the aspect ratio by 11% led to a thrust increase from 24.25N to 29.95N (approximately 23.5% improvement) and an improvement in rotor isentropic efficiency from 80.1% to 81.1%. Further enhancements, including halving the rotor tip clearance, increased isentropic efficiency to 83.4% while maintaining higher thrust levels. The redesigned MGT components were manufactured using SLA and are ready for future testing. These findings underscore the potential of combined mean-line and CFD modelling to optimise the performance and efficiency of micro gas turbines.

Paper ID FD-6 **Programme ID** 59

Unsteady rotating detonation structures and effects of wall temperatures

Zhaoxin Ren

Three-dimensional (3D) rotating detonation simulations solving the compressible Navier-Stokes equations with reduced chemistry are performed in a linear channel with radial premixed injection, mimicking rotating detonation combustor (RDC) configurations. Gaseous pre-evaporated kerosene and air are used as fuel, with air as the oxidizer. The RDC's wall temperature is varied to understand the effects of cooling on the detonation structure. Results show that 3D detonation waves propagate with highly inhomogeneous blast dynamics and transient 3D cell structures. Intersections of many transverse waves lead to extreme thermodynamic states and highly overdriven wave velocities. The lower the wall temperature, the more pronounced the unsteady wave structures with irregular cell sizes during detonation propagation. The conclusions reached in the current work could help in the design of cooling systems for RDCs.

Paper ID FD-7 **Programme ID** 79

High-fidelity simulation of cryogenic hydrogen jets for zero-carbon energy applications

Jac Clarke

Amidst global efforts towards achieving net-zero carbon emissions, cryogenic hydrogen as an energy source has emerged as a significant area of interest. This research aims to leverage computational fluid dynamics (CFD) to explore the complex multiphase, turbulent, and compressible flow phenomena inherent in the formation of pressurised liquid/gas hydrogen jets. Through the development of an in-house CFD code, deeper insights into these phenomena can be gained, with the ultimate goal of advancing our understanding and facilitating the safe transition to hydrogen systems.

Paper ID FD-8 **Programme ID** 87

Hydrodynamic forces on accelerating bluff bodies

Nicholas J Copsey

In this study our aim is to quantify the forces acting on an accelerating bluff body.
Previous work has shown added mass alone does not make up the gap between quasisteady forces and the measured forces for an accelerating bluff body. This suggests there is an additional acceleration dependent force.
A flat plate accelerating perpendicular to its surface is simulated using a boundary data immersion method with an iLES solver in a 3D domain. These simulations explore and quantify the acceleration dependent forces for a wide range of accelerations and Reynolds numbers. We explore a range of Reynolds numbers spanning 2 orders of magnitude and non-dimensional accelerations spanning 4 orders of magnitude.
Our results characterize the relationships which define the scaling of these acceleration dependent forces. The goal is to use these scaling definitions to model the forces on any accelerating bluff body, supplementing quasi-steady analysis techniques for a range of applications for more accurate results. Our initial motivation for this work was to model the forces on a swimmer's arm for any stroke without the need to use computationally expensive CFD models.

Paper ID FD-9 Programme ID 92

Transpiration Cooling Strategies and their Impact on Hypersonic Boundary Layer Flow

Raahil Sanjay Nayak, Adriano Cerminara and Jonathan Potts

Hypersonic flows generate high levels of heating on the vehicle surface due to viscous forces, causing high thermal stress in the vehicle structure, a highly undesirable phenomenon due to the possibility of structural failure. Various film cooling methods like transpiration and effusion have been explored both numerically and experimentally to address this issue, particularly for hypersonic flows. High cooling efficiency is usually achieved near the porous region, depending on the blowing ratio. However, recent studies indicate that cooling effectiveness significantly decreases in the turbulent boundary layer downstream of the injection site due to turbulent convective effects due to increasing turbulent kinetic energy. The present study employs Direct Numerical Simulations (DNS) of the flow governing equations using the SBLI code for transpiration cooling in Mach 6.1-7.7 flows in cases of both, induced boundary-layer transition and natural laminar flow over a flat plate. A systematic parametric study examines the sensitivity of near-wall coolant distribution, overall cooling efficiency of the mechanism and its effect on the transition mechanism in the freestream flow to blowing and suction ratios at the wall, as well as the distance between the porous injection and suction sections. This evaluation spans laminar and transitional flows to compare cooling efficiency across different flow configurations.

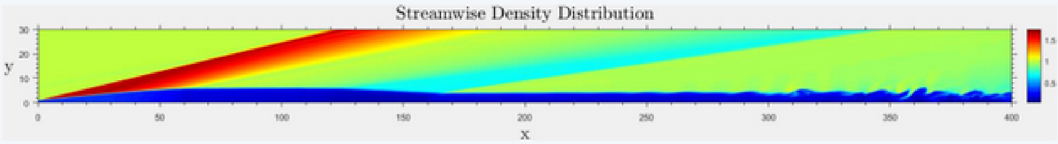


Figure 1 Streamwise Density Distribution in a flow with blowing section (x=20 - x=60) and suction section (x=120 - x=160)

Paper ID FD-10 **Programme ID** 112

Exhaust Jet - Wake Vortex Interactions Downstream of High-Lift Geometries

Scott Bennie, and Marco Fossati

The continued growth of commercial air traffic has led to airport capacity limitations worldwide. Such restrictions arising from the enforcement of mandated aircraft minimum safe separation regulations are directly influenced by our ability to categorise the relative risk downstream trailing vortex systems pose to follower aircraft. With an improved understanding of wake vortex phenomena, solutions to airport capacity problems may be achieved through the informed alteration of mandatory spacing restrictions. To deliver this, new wake vortex simulation strategies are required that provide accurate representations of the trailing vortex system whilst remaining computationally affordable. Presented in the following work, the Actuator-Line-Method (ALM) is extended to allow for the inclusion of aircraft exhaust jets within the simulation environment downstream of high-lift geometries. Through the ALM, observation of the exhaust-wake interactions becomes economically viable, allowing new insights to be gained regarding vortex-jet behaviour patterns. In particular the accelerated decay process such wake-jet systems promote with the formation of secondary vorticity surrounding the inner vortex pairs. The results of the present work forming the basis of cohesive arguments regarding the quantification of risk surrounding wake vortex phenomena.

Modelling Aerodynamic Drag of a Very Low Earth Orbit 1U CubeSat with a Boltzmann-BGK Approach

Joseff Parke Sturrock, Ben Evans and Zoran Jelic

The aerodynamic drag of a 1U CubeSat was investigated utilising an existing Boltzmann-BGK solver [1-2]. Very low Earth orbits (VLEO) are desirable regions for satellite operations for numerous reasons. Benefits include significantly improved pixel resolution of Earth observation instruments, exponentially reduced link budgets, end-of-life passive de-orbiting and a reduced orbital insertion altitude, effectively increasing launch vehicle payload mass – typically measured in $/kg to a higher altitude, stable low Earth orbit [3]. The main engineering challenges of these regions include significant atmospheric drag. This results in rapid orbital decay, leading to very short satellite lifespan. Additionally, the number density of corrosive monoatomic oxygen is orders of magnitude higher than at higher altitude, stable orbits [4]. Compensating drag with active propulsion eliminates orbital decay, allowing a stable orbit for as long as thrust is maintained.
All atmospheric parameters for the solver were sourced from NASA's NRLMSIS 2.0 atmospheric model [5]. Altitudes investigated range from 50 km to 500 km. Periods of solar minima and maxima, seasonal variances and local day/night cases were modelled. Drag coefficients were evaluated and compared with corresponding Knudsen numbers. In the near continuum regime, of Kn below 0.001 (altitudes below 50 km), classical macroscopic phenomena such as bow shocks were clearly visible. In the transitional regime, above Kn 0.001 and below Kn 1 (altitudes 50-100km), the aerodynamic shape of the off-axes cases produced slightly less drag than the on-axes case. In this Knudsen regime, a large fluctuation of drag coefficient values was observed in both geometries. In both configurations, a higher drag coefficient was produced in the Kn $<$ 1 regime. Peak drag coefficients occur for both geometries at around Kn $=$ 1. Between this region, and Kn $=$ 10, CD values drop sharply for both. The on-axes cases settle to a consistent CD of around 1.24, while the off-axes cases settle to a CD of 1.60 (using a consistent reference length), above 115 km altitude. At higher Knudsen numbers CD was found to be independent of Kn.

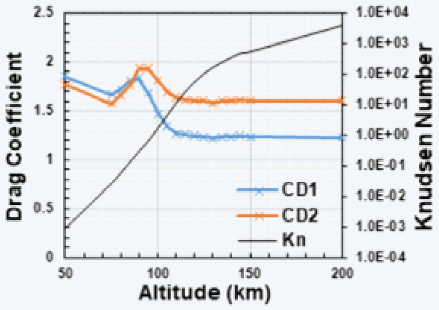


Figure 1 – Drag Coefficient vs Altitude (CD1/CD2) represents on-axes and off-axes cases, respectively.

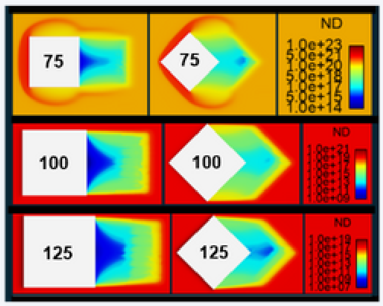


Figure 2 – Examples of flow fields around a 1U CubeSat – both on and off-axes cases studied. Altitudes of study within geometry (in kilometres).

[1] B.J. Evans, K. Morgan, O. Hassan, A discontinuous finite element solution of the Boltzmann kinetic equation in collisionless and BGK forms for macroscopic gas flows, Applied Mathematical Modelling, ***32*** *(2011) pp. 996-1015.*
[2] B.J. Evans, S.P. Walton, Aerodynamic optimisation of a hypersonic re-entry vehicle based on solution of the Boltzmann–BGK equation and evolutionary optimisation, Applied Mathematical Modelling, ***52*** *(2017) pp. 215-240.*
[3] P.C.E. Roberts, 1st Symposium of very low Earth orbit missions and technologies, CEAS Space Journal, ***14*** *(2022) pp. 605–608.*
[4] B.A. Banks, K.K. De Groh, S.K. Rutledge, Consequences of atomic oxygen interaction with silicone and silicone contamination on surfaces in low Earth orbit, NASA Technical Memorandum--1999-209179, (2019).
[5] J. T. Emmert, D. P. Drob, J. M. Picone, D. E. Siskind, M. Jones Jr., M. G. Mlynczak, NRLMSIS 2.0: a whole-atmosphere empirical model of temperature and neutral species densities, Earth and Space Science, ***8*** *(2021).*

Paper ID FD-12 **Programme ID** 121

AI Mesh-Informed Techniques for Optimising the Design Process

Callum Lock, Oubay Hassan, Rubén Sevilla and Jason Jones

This presentation introduces an approach for generating near-optimal meshes for computational simulations, ensuring adequate capturing of the solution and delivering reliable results efficiently. Traditional mesh refinement techniques require manual intervention by engineers, who adjust the mesh spacing based on observed solutions or employ mesh adaptivity. These methods are time-consuming, and the mesh quality heavily depends on the expertise of an engineer, often resulting in meshes with either an excessive number of elements or insufficient refinement around critical features.

To address these challenges, anisotropic meshes – which allow for element size adjustments in orthogonal directions, are utilised to enhance the computational efficiency. By elongating elements strategically in specific directions, the element count can be reduced, providing a more efficient means of representing the solution. However, this increases the complexity to the generation of near-optimal meshes.

The presentation outlines a proposed method to replace the traditional, time-consuming mesh generation process with an automated approach using artificial intelligence (AI). The AI learns from the spacing function over a background mesh, automating the refinement process and reducing dependency on manual adjustments.[1]

We apply this method to various three-dimensional compressible flow cases, demonstrating its potential in terms of computational efficiency and reduced carbon emissions, with variations in both flight conditions and geometries. Figure 1 illustrates the target and AI-produced meshes for an unseen aircraft geometry at a transonic condition, showcasing the AI's ability to generate near-optimal anisotropic meshes with appropriate refinement. The results highlight the capability of AI to produce high-quality meshes that significantly enhance the efficiency of computational simulations, offering a promising solution to the challenges of traditional mesh generation techniques.

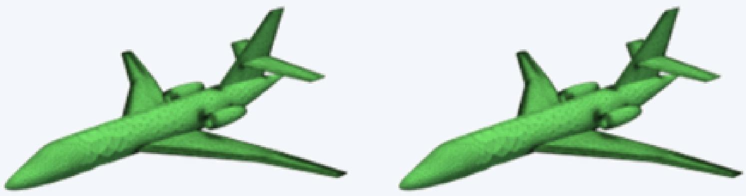


Figure 1 : Falcon: Target (left) and predicted (right) anisotropic meshes for a transonic flow condition

[1] C. Lock, O. Hassan, R. Sevilla, and J. Jones. Predicting the near-optimal mesh spacing for a simulation using machine learning, International Meshing Roundtable 2023

Paper ID FD-13 **Programme ID** 17

Body force modelling of surface Dielectric Barrier Discharge Plasma

Chigozie Okwudiri Eleghasim

The study presents the results of numerical investigation of the body force model derived from dielectric barrier discharge plasma actuator used for aerodynamic flow control. The investigation utilized multibody system of quiescent air, plasma, a dielectric, and combined models of electro-hydrodynamic force previously reported, upgraded, and implemented as a source term in the fluid flow solvers of OpenFoam software. The model's performance is assessed through numerical flow computations and empirical flow measurements reported in the literature. The numerical results show good agreement with the reported experimental results.

Paper ID FD-14 **Programme ID** 19

Electroosmotic Flow past an Array of Poly-Electrolyte Coated Solid Cylindrical Particles: A Particle-in-Cell Approach

Amit Kumar Saini and Ashish Tiwari

The present study is an attempt to deal with hydrodynamic aspects of mixed electroosmotic and pressure-driven flow through a membrane composed of a swarm of poly-electrolyte-coated solid cylindrical particles. The unit cell model approach is utilized to analyze the electrokinetic and hydrodynamic characteristics of the multiparticle system. The electroosmotic flow is generated under the influence of an externally applied electric field, and a pressure gradient is assumed in the axial direction of the cylinder. The coating of the poly-electrolyte layer, which is considered as a porous layer, has variable permeability characteristics. The electrolyte fluid contains charged ions, which can present both inside and outside of the poly-electrolyte layer (PEL). Hence, PEL acts as a semi-permeable porous layer. The PEL is denoted as a fixed charged layer (FCL) owing to an extra number density of immobilized charged ions. In order to derive the electric potential distribution in the membrane, the Debye Huckle approximation is assumed to linearize the Poisson-Botlzmann equation, which is further used in hydrodynamic governing equations to investigate the electrokinetic effects in the membrane. The flow domain is divided into two subdomains: the FCL region, where the flow is governed using the Brinkmann-Forchheimer equation, and the clear fluid region, where the flow is governed using the Stokes equation. The effect of electroosmotic parameters such as electric double layer (EDL) thickness, thickness ratio parameter, zeta potential, etc., and the media parameters such as viscosity ratio, particle volume fraction, Stress-jump parameter, Forchheimer number, variable permeability parameter, etc. is analyzed on the flow profile as well as hydrodynamic quantities of the membrane such as hydrodynamic permeability and the Kozeny constant.

Paper ID FD-15 **Programme ID** 31

Cooling next generation aircraft with PCM: Modelling of novel concept heat exchanger designs with CFD

J. Y. Frank, D. Borman, E. Greiciunas, A. Khan and J. Summers

The growing thermal demand in aircraft design poses significant challenges for developing thermal management systems (TMS) in next-generation aircraft. By incorporating a Phase Change Materials (PCM) Heat Exchanger into the Thermal Management System (TMS), current heat exchanger geometries could continue to be employed in future aircraft applications. PCMs absorb and release heat at almost constant temperature and could be used to absorb the energy at key stages of the flight, such that a reduction in the size of the main heat exchanger could be achieved.

While PCMs have been successfully applied in various cooling application including refrigeration, there is a lack of literature dedicated to the integration of PCMs into heat exchanger geometries for aerospace applications. The loading profile for aerospace favours a reduction of the melting time and the mass of the system in exchange for a higher allowable pressure drop so that more compact PCM heat exchanger geometries are needed. This can be achieved by increasing the surface area between the PCM and the heat transfer fluid. For aerospace applications, a plate-fin heat exchanger could be used as a base geometry, with layers of PCM enveloping the fluid layer.

Computational Fluid Dynamics (CFD) emerges as a powerful tool for accurately modelling the phase change within PCM heat exchangers and predicting the heat transfer performance. In this work, a CFD model will be used to determine the effect of Reynolds number on the uniformity of melting in a stripped fin compact heat exchanger geometry. Following these results, the distribution of fins will be varied over the length of the heat exchanger to obtain more uniform melting over the length.

Paper ID FI-1 **Programme ID** 15

Thermo-viscous fingering instability in cooling and spreading flows

Shailesh Naire

Molten fluid flows that cool as they spread are important in a wide variety of contexts, e.g., lava domes in geophysical flows and coolant in nuclear reactors. The interplay between the flow and cooling can also give rise to a variety of intriguing flow features and fingering instabilities. Motivated by the above, we consider theoretically a model system of a molten viscous drop extruding from a source and spreading over an inclined plane that is covered initially with a thin liquid precursor film. Lubrication theory is employed to model the one-dimensional spreading flow using coupled nonlinear evolution equations for the film thickness and temperature. The coupling between flow and cooling is via a constitutive relationship for the temperature-dependent viscosity. This model is parameterized by the heat transfer coefficients at both the drop-air and drop-substrate interfaces, the Péclet number, the viscosity-temperature coupling parameter and the substrate inclination angle. A systematic exploration of the parameter space reveals a variety of solutions illustrating the dynamics of a spreading flow undergoing cooling. These solutions are compared to a simpler model that results due to a further approximation of the temperature equation in the limit of small Péclet number. The stability of the one-dimensional solutions to small-amplitude variations in the thickness and temperature in the transverse direction is also investigated using linear stability and transient growth analysis, and numerical simulations. The existence of a thermo-viscous fingering instability is revealed. Two-dimensional numerical simulations confirm the stability analysis elucidating the underlying thermo-viscous mechanism.

Paper ID FI-2 **Programme ID** 118

Rotating Rayleigh-Benard Convection with Fixed Flux Thermal Boundary Conditions

Rhiannon Nicholls

The study examines how fixed temperature and fixed heat flux boundary conditions affect rapidly rotating linear Rayleigh-Bénard convection, considering stress-free mechanical boundary conditions. It is found that the solutions for fixed temperature and fixed flux are nearly the same in the fluid's interior. However, a double boundary layer structure is needed to meet the fixed flux condition. This structure includes an Ekman layer near the boundaries and a thermal boundary layer, which is purely diffusive and adjusts the temperature to meet the thermal boundary condition. The Ekman layer's thickness scales with the Taylor number as Ta^(-1/4), while the thermal boundary layer scales as Ta^(-1/6). To capture both layers' effects, their ratio is used as a small parameter in an asymptotic analysis: ϵ=Ta^(−1/12). The asymptotic solutions for these boundary layers are derived by analyzing the differences between the fixed temperature and fixed flux solutions. These asymptotic solutions are then combined with the fixed temperature solutions to obtain solutions for fixed flux boundary conditions.

Paper ID FI-3 **Programme ID** 125

Over-reflexion of gravity waves by vortices in a rotating ocean

C. Nolan and E. Benilov

We investigate a phenomenon where a gravity wave passing through a vortex is amplified by the interaction with the critical levels of the vortex. It is shown that even though some vortices do over-reflect some waves, the reflexion coefficient R is relatively small – namely, $R < 1.1$. This result is in stark contrast with the case of parallel currents, where R can even be infinite (a phenomenon, usually referred to as hyper-reflexion).

Paper ID FI-4 **Programme ID** 123

Over-reflexion of gravity waves by vortices in a rotating ocean

Miles Morgan

Fluid-driven grain flows comprise a particular facet of granular physics that is common in nature and industry yet poorly understood. These complex flows involve hydrodynamic forces on particles in addition to frictional grain-grain interaction. This work finds that when driven by fluid, grain flow in a silo can exhibit instabilities not present in classical dry silos. These can include viscous fingers, and dilute wormhole-like channels that rapidly approach the outlet, acting as a bypass of the wider silo packing. The onset of these wormholes is found to coincide with an imbalance of grain flow around the grain-fluid interface at the top of the granular bed.

Paper ID FI-5 **Programme ID** 29

Nowcasting of Atmospheric Convection using a Simplified Model

Kasia Nowakowska

Classical Rayleigh-Bénard convection has been extensively studied in the field of fluid dynamics; however, its application to understanding atmospheric convection has largely progressed independently. A key difference is the additional moisture in Earth's atmosphere. This leads to the condensation of water vapour which provides an additional buoyancy source to drive convection. The complex system means short term prediction or nowcasting which describes the current state of the weather and provides forecasts for the next few hours, typically within the 0-6 hour time range, has heavily relied on observational cell tracking. These methods predict the movement of convective storms effectively whilst prediction of the initiation and decay of cells has proven more difficult.

In this study, we explore the fundamentals of short-term prediction by employing a simplified model for moist convection known as the Rainy-Bénard model, which extends classical Rayleigh-Bénard convection. Our main objectives are to investigate secondary convective initiation driven by gravity waves and determine the predictive capability of a simple model. Forecasting is done using Echo State Networks (ESN), a specialised type of recurrent neural network.

Results indicate that plume initiation in the model is associated with gravity wave convergence, low convective inhibition, high potential energy for convective behaviour, and high kinetic energy. Initial predictions using ESNs on global parameters demonstrate good skill in predicting the timing of convective initiation.

By exploring prediction using a simplified model, we aim to deepen our understanding of gravity wave initiation and enhance our ability to make accurate predictions, with the potential to inform real-world nowcasting tools.

Paper ID FI-6 **Programme ID** 122

Predicting Bubble Fragmentation in Superfluids

Ryan Doran

In classical fluids, the Weber number is a dimensionless parameter that characterises the flow of a multi-phase fluid. The superfluid analogy of a classical multi-phase fluid can be realised in a system of two or more immiscible Bose-Einstein condensates. These superfluid mixtures have been shown to display a wider variety of exotic dynamics than their single component counterparts. Here we systematically study the dynamics of a binary immiscible Bose-Einstein Condensate in two dimensions, where a small bubble of the second component is used to "stir" the first component. We begin by rigorously mapping out the critical velocity for vortex shedding as a function of the size of the bubble, in analogy to the critical velocity of a laser spoon. Observing that the dynamics of the system depend on the initial size and velocity of the bubble, we then show that a dimensionless parameter with the same form as the Weber number accurately predicts the resulting bubble fragmentation.

Paper ID FI-7 **Programme ID** 80

Dynamics of rotating convection in Earth's outer core

Jo J Kershaw

An understanding of the flow dynamics of the electrically conducting liquid in Earth's outer core is a first step to unravelling the mechanisms behind the generation of our planet's magnetic field. Simplifying assumptions are necessary to numerically model extreme temperatures and pressures while accounting for complex non-linear dynamics and a huge range of temporal and spatial scales.
This project focuses on investigating the dynamics of non-magnetic rotating convection in different spatial regions of the spherical shell geometry. This should allow a more comprehensive exploration of the parameter space and facilitate comparison with experimental results.
Temperature and velocity data from existing simulations conducted in a spherical shell were interpolated onto cylindrical grids and data from the upper tangent cylinder (TC), the cylinder circumscribing the inner core, was extracted and evaluated to determine the heat transport and kinetic properties (represented by the Nusselt and Reynolds numbers) in this distinctive region.
At a fixed rotation rate, various constant temperature and flux conditions were imposed on the lateral boundary of a cylindrical simulation in Nek5000, their values informed by the analysis. These were combined with no-slip or no-stress kinetic conditions and fixed-temperature or fixed flux conditions on the horizontal boundaries. Scalings of the Nusselt and Rayleigh number were compared using these basic models.
The scalings vary significantly depending on the boundary conditions chosen. The critical Rayleigh number for convection onset also varies with the lateral boundary condition chosen, with local temperature gradients the determining factor. Future work will investigate kinetic conditions tailored to Earth's outer core.

Paper ID FI-8 **Programme ID** 120

Non-linear evolution of zonostrophic instability with varying magnetic field strength.

Azza Algatheem

The non-linear evolution of zonostrophic instability has a dynamic process that transfers magnetic fields within the solar interior. Studies have shown that a weak magnetic field can modify the flow onto the lower tachocline (Tobias, Diamond and Hughes (2007) , suppressing of zonostrophic instability initially, while a sufficiently strong field can enhance this instability. We consider a 2D Kolmogorov flow with a sinusoidal velocity field u=(0, sinx) for a magnetic field aligned with possible jet formation. The study uses a spectral code of the Dedalus package. Our nonlinear simulations are linked to some linear results presented from our comprehensive linear study (Al Gatheem, Gilbert, and Hillier (2023). In the longer simulation run, the study aims to probe into some fundamental processes at large-scale structures and generate a possible inverse cascade.

Paper ID OP-1 **Programme ID** 45

Numerically augmented experimental analysis of inclined backward facing steps

Morgan Taylor, Uttam Cadambi Padmanaban, Sean Symon, John Lawson and Bharathram Ganapathisubramani

The flow around backward-facing steps has been extensively studied due to its close resemblance to the rear of an automotive body. By understanding the shear layer detachment and reattachment phenomena, advancements can be made in terms of fuel efficiency and handling stability. A two- dimensional experimental investigation has been performed on a backward facing step with a constant height but varied inclination angles from 11 to 90 degrees using particle image velocimetry. The instantaneous variability in reattachment length and modal decomposition is compared for each angle to elucidate mechanism changes with inclination. The experimental data is augmented with discrete adjoint optimisation (data assimilation) to ensure compliance with the Naiver Stokes equations throughout the flow field. Lastly, resolvent analysis is applied to the data to yield further insight into frequency information and mechanisms of instability and the variation across inclinations. This work stages an investigation into deep inclined cavity flows that is to be performed at Southampton's newly developed matched index of refraction facility, where the impact of the downstream edge of the cavity and how it interacts with the separated shear layer is studied.

Paper ID OP-2 **Programme ID** 37

A Design Optimisation Framework for Lavel Nozzles in Uniform Supersonic Chemical Reactors

Luke Driver

The Interstellar media is a low temperature, low density (100K) vacuum, comprised of gas and dust. The ISM contains upwards of 450 species which are involved in roughly 6200 gas phase reactions. Some reactions have been suggested to be pathways to complex organic molecules (COM's), potential precursors to life. Finding the reaction rate of these reactions is the key to understanding the chemical evolution of the universe. Experimentally, low temperatures are obtained by expanding an inert bath gas (Nitrogen, Argon, Helium). through a Laval nozzle, generating a supersonic, low temperature jet where reactions are performed. Nozzles are designed using the method of characteristics (MOC), which is a way of generating a conventional nozzle geometry for a set of design conditions that produces a shock free exit flow. This assumes irrotational, inviscid flow, and hence the solution produced by this method is not accurate and provides no insight into the length of the jet or if a better nozzle geometry exists for a particular flow condition. The length of the jet and shock magnitude is critical as it determines the kinetic measurement error and the speed of the reactions can be performed; hence chemists would like to design based on these variables to improve their kinetic studies. There currently exists no CFD or design framework for the CRESU method, and this project aims to develop a fully automated (black box) framework that allows chemists to perform CFD on existing nozzles to obtain high fidelity data to improve current experimental techniques, and to enhance these designs through optimisation via Kriging and Cokriging surrogate modelling.

Paper ID TU-1 **Programme ID** 10

Characterisation of realistic rough walls in compressible turbulent boundary layers

D. D. Wangsawijaya, R. Baidya, S. Scharnowski, B. Ganapathisubramani and C. J. Kahler

Compressible turbulent boundary layers (TBLs) developing over rough walls comprise of phenomena related to high-speed flight vehicles, where the roughness is likely to be induced during operations due to thermal expansion, ablation, dust impact, water and ice droplets. In incompressible flows, drag penalty due to surface roughness is characterised by the log-law deficit corresponding to the Hama roughness function. Whether this deficit corresponds to that of compressible rough wall TBLs and which transformations should be applied to account for compressibility effects remains the subject of further investigation. This requires high-fidelity data over various rough surfaces across a range of Mach numbers. In this study, we investigate the boundary layer over two different rough surfaces, subjected to a wide range of freestream Mach numbers (M) and Reynolds numbers (Re).

Measurements are performed inside the trisonic wind tunnel (TWM) at the University of the Bundeswehr Munich. The test section comprises of a metal baseplate with removable inserts, upon which the test surfaces are assembled: a smooth wall and two rough walls made of P60- and P24-grit sandpaper sheets. Two sets of particle image velocimetry (PIV) measurements are conducted; the first is a set of cross-stream stereoscopic PIV and the second is in the streamwise-wall-normal plane. Tests are conducted in the following flow regimes: subsonic (M = 0.3), transonic (M = 0.8), and supersonic (M = 2 and 2.9). To ensure a wider range of Re at a constant M, the stagnation pressure of the tunnel is varied between 2.5 and 4.5 bar at M = 2. The range of friction Re (based on the smooth wall boundary-layer thickness of the test facility) covered in this study is between 1800 to 6400. A full characterisation of rough wall TBLs in a compressible flow regime, as well as the exploration of the appropriate incompressible flow analogy for logarithmic velocity deficit will follow.

Paper ID TU-2 **Programme ID** 47

On the evolution of the initially turbulent plane mixing layer: a numerical study

W. A. McMullan, J. Mifsud and M. Angelino

The turbulent mixing layer is a canonical flow type of interest to academic researchers, and is of relevance to many practical engineering configurations. Eighty years of academic research has shown the mixing layer to be an inherently complex flow configuration; a wide range of growth rates for the flow has been reported, with inconsistent results observed for single-stream, and two-stream shear layers. The flow displays a hypersensitivity to its initial conditions, with the distance required for self-similarity to be attained appearing to be a function of the state of the high-speed side boundary layer.

We report on the Large Eddy Simulation of a high Reynolds number incompressible plane turbulent mixing layer, originating from a turbulent high-speed side boundary layer, and a laminar low-speed side boundary layer. The Reynolds number of the high-speed side boundary layer is 1350, and the maximum local Reynolds number of the mixing layer is ~200,000, based on the visual thickness of the layer, and the velocity difference across it. A recycling and rescaling method is used to produce the time-dependent inflow condition for the turbulent boundary layer, and it is shown that this method is highly effective. Validation of the simulation methodology is performed through assessment of grid resolution, subgrid-scale model effects, and the influence of the spaneise domain extent.

We present evidence for three regimes of growth in the flow, each of which is governed by a distinct physical process. We demonstrate the physical characteristics of the flow which are required in order for the flow to attain a fully-developed state, and relate these to the properties of the upstream high-speed side boundary layer. An appropriate means of non-dimensionalisation of flow development lengths is proposed, which is shown to account for the apparent discrepancies in the development of self-similar initially turbulent shear layers observed between experimental facilities.

Paper ID TU-3 **Programme ID** 102

Wall function modifications in OpenFOAM for heterogeneous roughness modeling

Mridu Sai Charan Arukalava Seshasayee, Yabin Liu, Bharathram Ganapathisubramani, Michael P. Schultz, Paul Hamblett and Ignazio Maria Viola

The increase in frictional resistance on ship hulls can be attributed to non-uniform surface roughness, potentially through the settlement of bio-fouling or mechanical damage, which increases drag and fuel consumption. This roughness is not always distributed uniformly over the hull, and currently, there are no established methodologies using a Reynolds-averaged Navier-Stokes modelling approach to account for this non-homogeneous roughness distribution. In this study, we develop and validate with experiments modified wall functions within the computational fluid dynamics software OpenFOAM, in order to accurately simulate the effects of both uniform and heterogeneous surface roughness on boundary layer growth. These modified functions are informed by roughness parameters measured experimentally in a laboratory. Initial validation efforts focused on a wall with uniform roughness. The results demonstrate improved accuracy in predicting velocity profiles and shear stress distributions when compared to state-of-the-art wall functions. Following successful validation with uniform roughness, the study outlines plans to extend the model to abrupt streamwise changes in surface roughness. This extension aims to investigate the transient change of the boundary layer from that of the upstream surface to that of the downstream surface, and the associated growth of the internal boundary layer, providing insights into the flow dynamics and stress distributions. This fundamental research project will allow practitioners to accurately predict the resistance of ship hulls with heterogeneous roughness distribution using Reynolds-averaged Navier-Stokes simulations and will enable optimal planning of hull cleaning and potentially the targeted application of fouling control coatings.

Paper ID TU-4 **Programme ID** 105

End-to-end optimization of compressible turbulent flow over permeable interface via differentiable fluid dynamics

Wenkang Wang and Xu Chu

This research aims to advance the understanding and optimization of hypersonic flow over porous media using the automatic differentiation (AD) capability of differentiable fluid dynamics. The objective is to demonstrate the potential of AD-based optimization for end-to-end optimization of hypersonic coupled flow. This approach leverages AD to efficiently handle high-dimensional optimization problems, providing a flexible alternative to traditional methods. AD-based optimization has great potential for tackling notoriously challenging problems due to the high cost of highdimensional optimization. This study will provide valuable insights and guidelines for the design and application of porous media in Thermal Protection Systems (TPS) in aerospace technology. The research addresses significant challenges in multi-objective optimization and fluid mechanics, leveraging state-of-the-art differentiable fluid dynamics.

Paper ID TU-5 **Programme ID** 119

The impact of turbulent patches on transition to triadic resonance in internal wave beams

Laura Irvine

Internal gravity waves provide an important mechanism of energy transfer and mixing in the oceans. One such mechanism is the Triadic Resonance Instability, a weakly nonlinear instability leading to the creation of two ‘child’ waves of lower frequency, for which the wavenumbers and frequencies sum to those of the parent wave. For narrow beams, as may be generated by topographic interactions, for example, there is evidence of an amplitude threshold below which TRI does not occur. The behaviour around this threshold, however, appears complex, and it has been shown that disrupting the wave field can lead to the triggering of a persistent triadic response even below the linear stability threshold. Using synthetic schlieren and shadowgraph visualisations of laboratory experiments in a linearly stratified tank, we investigate the influence of several turbulent three-dimensional structures (vortex rings horizontally, vortex rings vertically, and bubbles) on a sub-threshold internal wavebeam. Our results show two types of ‘triadic response’ (so termed due to uncertainty about the underlying mechanism) can be triggered: transient and sustained. A transient triadic response shows the development of a wave triad at sub-threshold wavebeam amplitudes after a triggering event (vortex ring or bubble), and then the decay of this response until only the primary wave remains. A sustained triadic response appears identical to TRI, but occurs below the amplitude threshold and remains from the triggering event until the end of the experiment. We investigate the features of the triggering mechanisms (timing, location, size, velocity) and suggest that interaction time between the turbulent patch and the internal wave is the primary factor in ability of disturbance to trigger triadic response. The discovery of other mechanisms for triggering this transition to triadic states suggests that internal waves may be responsible for more mixing than previously thought.

Wall-modelled LES for BFS with injection of upstream turbulence

J. Dobrzycki, H. Xia, D. Butcher and I. Langella

Flow separation, recirculation and reattachment are complex concepts in turbulent flows and are commonly exhibited in external and internal flows, such as combustion chambers. One particular example, namely the "backward facing step" (BFS) combines these features and serves as a benchmark case for the CFD community to validate turbulence models and numerical methods. Both RANS and LES studies can be widely found in literature with some lower Reynolds number cases performed using DNS. In this work, a Wall-Modelled LES (WMLES) approach is employed to reduce grid sizes hence computational costs. Specific attention is paid to the introduction of upstream turbulence as it is integral to the accuracy of results [1] using two robust methods. In the first method (Fig 1a), the inlet boundary is split into near-wall and bulk zones, where synthetic eddies are introduced to each zone with turbulence intensity and length scale [2] without any profile for simplicity. In the second method (Fig 1b), ("transverse") synthetic eddies are introduced vertically from a slot on the upstream wall boundary. Both a lower and moderate Reynolds number BFS cases are studied (with a lower Re case shown below). Favourable comparisons with reference DNS and experimental data [3] are obtained.
Fig 1c compares the mean velocity profiles at an upstream reference location and three downstream locations, where the current WMLES agrees well with reference data. Fig 1d plots the skin friction with WMLES being on par with the DNS where the reattachment length has been accurately predicted. The potential benefit of the present approach is that it can be incorporated in wall bounded flows with or without a wall intersecting inlet. Our results also show that this can be done for relatively lower costs, given the use of considerably fewer grid points (WMLES 4.6m; LES 7m; DNS 9m). They also show our approach is robust and directly applicable to mainstream commercial and open source CFD codes.

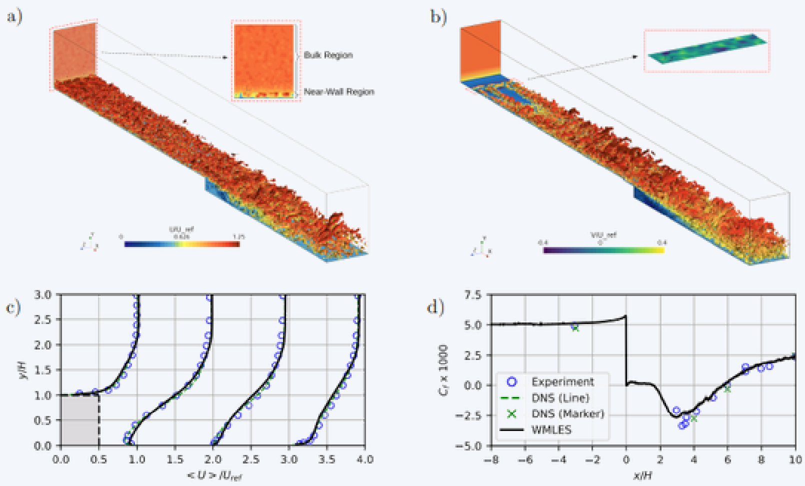


Figure 1: Results for Jovic & Driver case [3]: a) Injection of turbulence at inlet; b) Injection of turbulence on the wall; c) U velocity profiles at $x/H = -3.12, 4, 6, 10$; d) C_f comparison (step at $x/H = 0$)

[1] Aider, J.L., Danet, A. and Lesieur, M., 2007. Journal of Turbulence, 8(51).
[2] Jarrin, N., Benhamadouche, S., Laurence, D. and Prosser, R., 2006. Inter. J. Heat and Fluid Flow, 27(4), pp.585-593
[3] Jovic, S. and Driver, D.M., 1994. (No. NASA-TM-108807).

Paper ID TU-7 **Programme ID** 32

Non-linearities induced by deterministic forcing in the low-frequency dynamics of transitional SBLI

M. Mauriello, P. Sharma, N. Sandham and L. Larcheveque

One-period direct numerical simulations (DNS) are performed in a M = 1.5 transitional shock reflection with separation. The aim is to investigate any unsteadiness and the mechanism underlying its origin. The work of Sansica et al. (2016) suggested that the appearance of the low-frequency unsteadiness in transitional shock-wave/boundary layer interactions (SBLI) is due to the breakdown of the deterministic turbulence, in his work stimulated with a pair of monochromatic oblique unstable modes. To clarify the mechanisms, in this work the incoming laminar boundary layer at the inlet is forced with two different and opposite arrangements of oblique unstable modes selected from an a priori stability analysis. Each arrangement is given by the combination of two unstable waves moving at frequencies such that their difference falls in the low-frequency range corresponding to the Strouhal number of 0.04. The deterministic forcing allows the introduction of nonlinearities, and high-order statistical tools are leveraged to identify the occurrence of quadratic couplings. The results show that the low-frequency unsteadiness and transition to turbulence are decoupled problems. The unstable modes of the boundary layer interact non-linearly: high-frequency modes cascade non-linearly towards higher frequencies, initiating the turbulent cascade process, and towards lower frequencies. The low-frequency quadratic coupling with the flow characteristics at the separation point is responsible for the unsteadiness. It is shown that the trace of the unsteadiness in the wavenumber space is 2D.

[1] SANSICA, A., SANDHAM, N. D. & HU, Z. 2016 Instability and low-frequency unsteadiness in a shock-induced laminar separation bubble. Journal of Fluid Mechanics 798, 5-26

Paper ID TU-8 **Programme ID** 12

Boundary layer flows over rough surfaces

Jason Ferguson

The study of boundary layer flows involving Newtonian fluids has been a subject of interest for researchers in fluid dynamics for more than a century. Amongst a large variety of studies, the investigation of surface roughness is of particular interest to us. Studies have shown that surface roughness can delay the onset of instability for the rough rotating disc boundary layer see the work of (Garrett, 2016).
We consider high Reynolds number boundary layer flows over a rough moving surface. We restrict our attention to the study of boundary layer flows of fluids induced by a continuous semi-infinite impermeable moving plate. Various models for roughness are considered and we analyse how such surfaces affect the boundary layer. The purpose of this research is to investigate the linear stability of rough surfaces and to determine if surface roughness can delay the onset of instability for this flow configuration.
We formulate the problem by appropriately transforming the Navier-Stokes equations to account for the roughness and solve the governing equations utilising the Keller-box scheme. We examine the basic flow profiles for a range of amplitudes, and we observe spatial periodicity. We proceed by analysing the linear stability of the flow by considering the mean flow profiles and conducting a normal mode analysis. We solve the resulting eigenvalue problem numerically and present our findings. To verify our numerical results two approaches are considered. For the first approach we conduct a large Reynolds number asymptotic analysis which is a technique that has been carried out for a variety of flow configurations. The second approach involves freezing the basic flow in space and analysing the stability of the individual flow solutions, a technique which is often utilised for time period flows, and we follow the work of (Morgan, Control of stationary convective instabilities in the rotating disk boundary layer via time-periodic modulation, 2021)

Paper ID TU-9 **Programme ID** 25

Frequency-domain reduced-order modelling for nonlinear dynamics in turbulent flows

Xiaodong Li and Davide Lasagna

Gradient computation of turbulent flow quantities is of high importance in optimization and flow control. However, reliable and efficient methods to obtain these gradients are lacking due to the chaotic nature of turbulence. In previous work, Unstable Periodic Orbits (UPOs) were used to bound the exponential growth of adjoint solutions. Although finding UPOs is feasible in low-dimensional dynamic systems or low-Re turbulent flows, the computing cost becomes expensive for high-Re turbulent flows since UPOs proliferate dramatically with increasing their time periods and become increasingly unstable in turbulent flows.

Rather than finding UPOs in a full-state space, we propose to utilize a reduced-order model (ROM) to circumvent such impediment while the benefit of periodicity constraints is maintained by considering the ROM in the frequency domain. To this end, the spatiotemporal basis functions of the low-order space are extracted using Spectral Proper Orthogonal Decomposition. By virtue of using SPOD modes, the Navier-Stokes equations are converted into a low-order nonlinear algebraic system via Galerkin projection. Due to the truncation of SPOD modes, the amplitude coefficients projected from flow data violate momentum conservation. Therefore, the ROM's amplitude coefficients are tuned with gradient-based optimization to conserve the momentum, with minimizing the residuals in the low-order space.

The proposed approach is validated in a 2D lid-driven cavity at Re=20,000, which exhibits chaotic dynamics. Different frequency-domain ROMs were constructed using the truncation of frequencies and SPOD modes. Periodic solutions were obtained by initializing the optimization problem with the values projected from different data blocks. Numerical results show that the periodic solutions of frequency-domain ROMs can well predict the timeaveraged turbulent kinetic energy and dynamical flow features.

Paper ID TU-10 **Programme ID** 33

Flow Characteristics of High Reynolds Number Turbulent Boundary Layers Over Heterogeneous Ridges

T. Medjnoun, M. Nilsson-Takeuchi and B. Ganapathisubramani

Boundary layer flows over heterogeneous surfaces are crucial for various engineering applications, especially in natural and urban environments where predicting frictional drag is challenging. This research focuses on turbulent boundary layers over smooth spanwise-ridge heterogeneous surfaces at high Reynolds numbers.

Experiments were conducted in the Boundary Layer Wind Tunnel at the University of Southampton using a Floating-Element Drag Balance (FEDB) to measure frictional drag, stereoscopic Particle Image Velocimetry (sPIV) for flow topology assessment, and Hot-Wire Anemometry (HWA) for spectral analysis. Results indicate that the skin-friction coefficient depends on the Reynolds number for ridge spacings $S/\delta \sim O(1)$ (where S is the spanwise spacing and δ is the boundary layer thickness). For $S/\delta < 1$, drag appears invariant to Reynolds number, suggesting a possible existence of a heterogeneous aerodynamic roughness height equivalent to the homogeneous roughness height ks. Secondary motions induced by surface heterogeneity significantly influence flow characteristics, correlating more with relative roughness height h/δ than roughness Reynolds number $h+$. These motions redistribute energy among large scales, inhibiting very-large-scale motions (VLSMs) above ridges while allowing VLSMs in valleys for $S/\delta \sim O(1)$.

These findings highlight the limitations of conventional models assuming homogeneous roughness and emphasize the need for refined models that account for surface heterogeneity. Improved predictive tools are essential for engineering applications requiring accurate drag and flow behaviour predictions. These results underline the necessity for further research on both strip- and ridge-type surfaces at high Reynolds numbers to develop more reliable predictive tools for real-world applications.

Paper ID TU-11 **Programme ID** 34

Wall pressure fluctuations beneath a turbulent boundary layer subjected to mean pressure gradients at high Reynolds numbers.

Prateek Jaiswal, Thomas Preskett, Marco Virgilio and Bharathram Ganapathisubramani

Wall pressure fluctuations beneath a turbulent boundary layer with and without mean pressure gradients were measured in the boundary layer wind tunnel at the University of Southampton at very high Reynolds number ($Re\tau \geqslant 2000$). The fluctuating pressure was measured using an array of sub-miniature and quarter-inch microphones. Additionally, flow velocity profiles and fluctuations within the turbulent boundary layer were obtained using hot wires and particle image velocimetry. Adverse and favorable pressure gradients were achieved by placing a rotatable NACA 0012 airfoil with a chord length of 1250 mm above the wall. The one-point spectra and the two-point velocity and pressure correlations were analysed and compared with exisiting empirical spectral models for wall pressure. Furthermore, scaling of wall pressure fluctuations with mean and turbulent quantities is performed in order to show suitable parameters that dictate the wall-pressure spectra. The effects of mean pressure gradients on the wall-pressure spectra and very-large scale motions are discussed using synchronized velocity-pressure and velocity field measurements.

Paper ID TU-12 **Programme ID** 35

High Reynolds number smooth wall turbulent boundary layers with streamwise pressure gradients

Thomas Preskett, Marco Virgilio, Prateek Jaiswal and Bharath Ganapathisubramani

The effects of pressure gradient history on smooth-wall turbulent boundary layers is not well understood primarily due to lack of data. This study aims to address this gap by generating data in a turbulent boundary layer at high Reynolds numbers subjected to a series of pressure gradient histories (favourable to adverse and vice-versa). Measurements are carried out over the smooth bottom wall of a 12m long wind tunnel with the pressure gradient history imposed on by using an airfoil mounted in the freestream. By changing the angle of attack of the freestream airfoil, different combinations of favourable/adverse pressure gradients (and vice-versa) are imposed. Particle Image Velocimetry (PIV) captures the detailed structure of the boundary layer from one chord upstream to one chord downstream of the aerofoil. This is followed by measurements of skin-friction using oil-film interferometry. Mean and turbulence profiles as well as two-point correlations of the velocities at various streamwise locations will be compared to isolate the effects of different pressure gradient histories on the boundary layer characteristics. This analysis will elucidate the influence of flow history on boundary layer development and its subsequent impact on downstream flow structures.

Paper ID TU-13 **Programme ID** 40

Mitigating Delays in Turbulence Onset: SST-IDDES with Synthetic Eddy Method for Separated Flows

Aan Yudianto, Alistair Revell, Saleh Rezaeiravesh, Ajay B Harish and Mark Quinn

Addressing the Grey Area problem in hybrid simulations between Reynolds Averaged Navier Stokes (RANS) and Large Eddy Simulation (LES) is critical for accurate prediction of separated flow over smooth-body geometries. This study evaluates the capability of seamless hybrid RANS-LES simulation to predict mild and smooth-body separation and examines the application of the Synthetic Eddy Method (SEM) for mitigating delays in turbulence onset in separation of the flow over a rounded ramp. A precursor simulation is generated as a prescribed inlet condition for the main simulation. The k-ω Shear Stress Transport (SST) Improved Delayed Detached Eddy Simulation (IDDES) was utilised as the seamless hybrid RANS-LES method. A mesh independency study is conducted to determine the optimal mesh resolution. Results indicate that the standard SST-IDDES overpredicts the reattachment point due to delays in the onset of resolved turbulence, leading to a larger recirculating region. An improved version of Synthetic Eddy Method (SEM) has been implemented as an inlet boundary condition, following a turbulence length scale sensitivity analysis that converges at a length scale value of 25% of the boundary layer thickness. Comparisons of skin friction coefficient and pressure coefficient between SSTIDDES and SEM-IDDES simulations showed good agreement with reference data. The reattachment point prediction improved which shows only a 1.63% deviation from the reference data. Flow visualisation, including Q-criterion, instantaneous velocity, vorticity, Detached Eddy Simulation field, and Reynolds stress comparison, are also presented to analyse the effects of different mesh resolutions for both SST-IDDES and SEM-IDDES.

Paper ID TU-14 **Programme ID** 41

Transition to turbulence in supersonic flow over a Gaussian bump

Max C Walker

This study utilises direct numerical simulation to investigate the influence of Gaussian bumps in a supersonic flow field. Bumps constitute a canonical shock-wave/boundary-layer interaction (SBLI) configuration, but have not received the same level of attention as shock reflection and ramp configurations. At sufficiently high flow velocities, the bump exhibits two separation regions at the feet of the bump which amplify disturbances through recirculation. A relevant flight condition of Mach 6 at 28km altitude is studied where the bump sits downstream of a sharp wedge. The impact of varying the wedge angle on flow physics and turbulence onset was investigated using simulations of two wedge configurations. A constant cooled wall temperature of 600K was maintained throughout. Disturbances that are amplified through the upstream separation are damped as the flow accelerates over the bump and experience partial re-laminarisation. They are subsequently amplified through the downstream recirculation, but with a different preferred wave number. These mechanisms are sufficient to cause a transition to turbulence downstream, demonstrated through turbulent kinetic energy (TKE) visualisations. A higher magnitude of reverse flow in the separation bubble is seen to trigger an earlier transition to turbulence. The development of disturbances persists even in the absence of upstream forcing, giving confidence of the existence of absolute instability within the separation bubbles.

Paper ID TU-15 **Programme ID** 44

Round jet impingement confined in a cylindrical cavity: a parametric study using Large Eddy Simulation

Jacqueline Mifsud, James Jewkes and Andrew McMullan

Impinging jets are synonymous with efficient transport of heated fluid to (or from) the surface of an object. However, a round jet impinging into a cylindrical cavity, a configuration typically encountered in mould cooling, has received limited attention to date, especially for the flow and geometrical properties considered in this study. In particular, we study Reynolds number and diameter ratios that are representative of mould cooling applications.

This talk will review a parametric study of the interaction of a turbulent round jet impinging within a cylindrical cavity at a Reynolds number of 4000. We conduct wall-resolved Large Eddy Simulation (LES) to generate highly-resolved, transient, three-dimensional computational predictions of the flow field. The simulations are conducted using LES in the finite-volume framework of OpenFOAM, and use a custom recycling-rescaling method for turbulent inflow generation that has been previously validated for an incompressible turbulent pipe flow. We leverage this numerical methodology to explore the effects of varying degrees of confinement on flow features. The simulation methodology is further validated through the assessment of grid resolution and sub-grid scale model effects.

We review the flow structures and circulation patterns observed when varying: a) the distance between the nozzle-exit and the impingement surface from 0.5 to 5 jet diameters, and, b) the ratio of the cavity diameter to the jet diameter. The model is shown to predict key flow features that are indicative of the heat transfer in this configuration. Overall, the proposed OpenFOAM LES methodology can be applied to predict turbulent flows in various internal configurations, with modifiable material and geometric parameters.

Paper ID PO-1 **Programme ID** 9

Enhancing Indoor Air Quality in Historic Educational Buildings: A Computational Fluid Dynamics Approach to Retrofitting HVAC Systems in K-12 Schools

James Afful

In our world today, there is a growing focus on making sure that students have a healthy and comfortable learning environment, which includes maintaining thermal comfort and high-quality indoor air, essential for maximum student productivity. Several K-12 schools, built primarily in the 1990s, are now facing the challenge of upgrading their old HVAC systems, and it is ideal to have a best practices policy manual for the retrofit. In our study, we introduce a straightforward computational method that schools can use to analyze and improve the indoor air quality and thermal comfort their classrooms. Temperature and air velocity data was collected from classrooms of six schools which have representative archetypes of the schools in the Des Moines Public Schools District, to fine-tune our computational fluid dynamics (CFD) model. After validating our model, we applied it to compare the current HVAC setups in various classrooms with what's recommended by current standards with the goal of identifying how these systems could be better designed or adjusted to improve air quality and make students more comfortable. Our findings are meant to be a practical guide for schools looking to upgrade their HVAC systems. By following our recommendations, schools can make informed decisions that benefit student well-being, enhancing learning outcomes and improving energy efficiency.

Paper ID PO-2 **Programme ID** 103

On Constructing Lagrangians for dissipative systems

Josiah-Shem Davis

When physical systems convert energy to heat in a moving mechanical system these systems are said to be dissipative dynamical systems.
In the 1930s it was shown to be impossible to derive equations of motion for a constant mass Newtonian fluid which included dissipative terms, from a variational principle - that is, using Euler-Lagrange equations containing only integer-order derivatives of coordinates [1].
However, since Fred Riewe in 1995 it has been shown that with the addition of non-integer order "fractional" derivatives it is possible to derive equations of motion for dissipative dynamical systems from a variational principle [2,3].
The following presentation is an exploration of a novel functional-derivative-based approach to the stationary action principle, wherein by making use of fractional derivatives, equations are derived describing a general system of dissipative linear dynamical equations.
Application is then made in the description of the flow of a coarse-grained, and linearised version of hydrodynamics involving dissipation.

[1] Bauer, Paul S. (1931). Proceedings of the National Academy of Sciences, 17(5), 311-314.
[2] Riewe, Fred (1995). Physical Review E, 53(2), 1890-1899.
[3] Lazo, Matheus J. and Krumreich, Cesar E. (2014). Journal of Mathematical Physics, 55(12), 122902.

Paper ID PO-3 **Programme ID** 107

Investigating the Breakup Dynamics of Liquid Gallium Jets

Josh Parkin

Whilst jet breakup has been a key area of research for nearly a hundred years, there is limited work on the breakup dynamics of liquid metal jets, particularly at velocities where gravity is significant. Our study conducts computational and experimental work to investigate the breakup length of liquid gallium jets as a function of driving velocity. The results are of key interest for achieving nuclear fusion by firing a projectile at a fuel confined by liquid lithium jets.

Paper ID PO-4 **Programme ID** 108

Multi-Objective Optimisation of an Aerofoil using DNN driven CFD and MCS algorithm framework

Asif Mushtaq Ahmed Ansari

The optimisation of aerofoil shapes is crucial for enhancing the aerodynamic performance of an aircraft. Traditional CFD-based optimisation is computationally expensive and time-consuming. Recent advancements in deep learning (DL) have opened new avenues for accelerating CFD simulations while maintaining accuracy. This research introduces a novel framework for the multi-objective optimisation of aerofoil designs, integrating deep neural networks (DNNs) with computational fluid dynamics (CFD) to predict the flow fields around aerofoils and the Modified Cuckoo Search (MCS) algorithm for multiobjective optimisation. The framework begins by generating a diverse dataset of aerofoil shapes using the PARSEC parameterisation and simulating their flow fields using FLITE2D CFD solver. This dataset trains a DNN model, implemented using PyTorch, to predict key flow variables such as density, velocity components, and total energy for different airfoil geometries and Reynolds numbers. The MCS algorithm is then employed to iteratively explore the design space, employing the DNN model to quickly evaluate candidate solutions. This hybrid methodology is validated against traditional CFD methods, revealing significant computational time savings while maintaining high accuracy in aerodynamic performance predictions. The outcomes of this research offer significant potential for streamlining aerofoil design processes in the aerospace industry, enabling faster and reliable optimisation for civil aviation, military, and UAV applications.

Paper ID PO-5 **Programme ID** 111

Hybrid particle-phase field hydrodynamic theory of dilute colloidal sedimentation and floatation close to a liquid-gas interface

Alexandra Hardy, Abdallah Daddi-Moussa-Ider and Elsen Tjhung

We present a two-phase field model and a hybrid particle-phase field model to simulate dilute colloidal sedimentation and flotation close to a liquid-gas interface. Both models are coupled to the incompressible Stokes equation, which is solved using a mixture of sine and complex exponential Fourier transform to take into account of no-slip boundary conditions. Both models are shown to be thermodynamically consistent through explicit coarse-graining from lattice theory and mapping to a Fokker-Planck equation. We also derived the coupling parameter, which couples the dynamics of the colloidal particles to that of the fluid and we show that the particles remain confined inside the liquid phase as long as the coupling parameter is large enough compared to the temperature scale. Finally, we solved the interfacial profile analytically using a perturbative approach, which shows an excellent agreement with both two phase field and hybrid particle-phase field simulations.

Paper ID PO-6 **Programme ID** 115

A Data-driven Model Based on Gated Recurrent Unit Neural Network for the Prediction of Fluid Flow a Past Triangular Cylinder

Ganesh Sahadeo Meshram, Suman Chakraborty and Partha P. Chakrabarti

A predictive model for the fluid flow around a triangular cylinder is created using a Gated Recurrent Unit neural network (GRU-NN) that is driven by data. The lattice Boltzmann model is employed to simulate the spatiotemporal fluid flow data obtained by solving the two-dimensional unsteady NavierStokes equations. The GRU-NN model is utilized to predict the temporal drag coefficient as well as the streamwise and crossflow velocities of the flow passing the triangular cylinder. Furthermore, the LSTM-NN was employed to compare the forecast results with those obtained from the GRU-NN. The models are subsequently adjusted to provide an optimum model with specified model hyperparameters. The model performance and accuracy were evaluated using various metrics parameters, including R^2 score, mean squared error, and mean absolute error. The various performance plots are shown to evaluate the model. The projected velocity fields, drag, and lift forces are being compared to the results obtained from numerical simulations.

Paper ID PO-7 **Programme ID** 13

Guided Oscillations in Partially Ionised Solar Chromosphere Driven by a Spectrum of Waves

A. Alharbi

Partial ionised solar plasma has received recently more attention as several processes in the solar atmosphere could not be explained by assuming a fully ionised state. In this study, we investigate the numerically temporal evolution of waves driven by a spectrum of waves in a thin magnetic stratified flux tube. The plasma dynamics is studied considering a two-fluid (charged particles and neutrals) framework, for a frequency regime that is comparable with the collisional frequencies between ions and neutral particles. The evolutionary equations for the two species have been derived analytically, and analytical solutions are obtained considering an initial value problem. Our results show that the charged population oscillates in such a way that an observer would sense a wave propagating with the frequency of the driver, and this signal is followed by a decaying wake that oscillates with the cutoff frequency. Thanks to collisions between ions and neutrals, the wave associated with the neutral species is very rapidly decaying.

Paper ID PO-8 **Programme ID** 18

A Systematic Review: How are Pathogens Distributed in Respiratory Emissions

Danny A P Blundell

Pathogen-laden aerosols are the mechanism of choice for a myriad of airborne transmissible diseases. Much has been done to build a picture of the physics, biology, generation and transport of these bioaerosols, in the hopes of mitigating infection risk to those susceptible. However, the initial distribution of the pathogens across the varying sizes of respiratory aerosols remains unclear, due to the demanding experimental challenges of measuring particle size segregation and those inherent when working with microorganisms. This problem is further magnified when considering the role played by evaporation in the air and the implications aerosol and droplet sizes have on the transport of the pathogens to the susceptible people. This systematic review aims to extract and collate data from the current literature to answer the question "What is the concentration of pathogens throughout the range of aerosols sizes produced in respiratory emission". The review is employing a systematic methodology, as laid out by PRISMA, in order to identify and examine the experimental methods capable of measuring aerosol size segregation and the quantity of pathogens/microorganisms present across the different sizes released during respiratory activities. This analysis of the literature will ultimately collate measured data to develop a better quantitative distribution of initial aerosol sizes and pathogen concentrations. The review also aims to employ bibliometric software in its analysis to help identify research gaps and connections between those working on this problem.

Paper ID PO-9 **Programme ID** 28

Impact of Pressure Gradient History on Wall Shear Stress Using the Momentum Integral Equation: Validation of Existing Methods

Marco Virgilio

This study investigates the impact of pressure gradients on wall shear stress using the momentum integral equation for boundary layer flows, which is crucial for evaluating the performance and energy efficiency of fluid systems and aerospace applications. By numerically solving this equation, it is possible to predict skin friction through integral parameters without needing to measure flow properties close to the wall. For laminar flows, Thwaites' method offers an approximate expression for the growth of momentum thickness, accounting for pressure gradients. In turbulent boundary layers, strong pressure gradients can lead to history effects, causing different boundary layers with identical Reynolds numbers to respond differently based on upstream pressure gradients. These non-equilibrium effects complicate the accurate solution of the momentum integral equation. To address this, Agrawal et al. extended Thwaites' method to turbulent flows, though their approach is limited by the constants derived from specific pressure gradient data. This work validates Agrawal's code for boundary layers under various pressure gradient intensities and history effects. The pressure gradients are generated by varying the angle of attack of a NACA 0012 wing and its proximity to the wall. Integral quantities such as momentum and displacement thicknesses are measured using Particle Image Velocimetry (PIV), while Oil Film Interferometry assesses the friction coefficient distribution on the wall. Additionally, a modified version of the Thwaites method for turbulent flows in non-equilibrium states is presented.

Paper ID PO-10 **Programme ID** 76

Predicting Friction in Total Hip Replacement Bearings: A Multiscale Approach

Robin Furze

The tribological performance of total hip replacements (THRs) significantly influences their longevity and success and, consequently, are subject to extensive preclinical tribological tests, which are time-consuming and difficult to predict the outcome of in silico. One such test, ASTM F3143-20 [1], prescribes the measurement of frictional torque in a THR when subject to a reciprocating motion in one degree of freedom coupled with a dynamic loading profile.

THRs predominantly operate in the mixed lubrication regime, where accurate friction prediction is challenging due to the complex interactions between solid and fluid domains across disparate scales. Deterministic models have been developed to predict friction of THRs under ASTM F3143-20 operating conditions, dividing the bearing into contact and lubricated regions [2]. Such models do not capture the microscale fluid and solid contact phenomena resulting from asperity interactions and, as a result, do not accurately replicate in vitro friction per ASTM F3143-20.

This project examines whether a new computational model utilising heterogeneous multiscale methods [3] can overcome the limitations of current models for predicting friction in the mixed lubrication regime under ASTM F3143-20 operating conditions. The methods provide a framework for coupling macro and microscale models, which can capture complex microscale fluid-structure interaction phenomena not possible with conventional approaches utilising the Reynolds equation.

[1] ASTM F3143-20. (2020). Determination of Frictional Torque and Friction Factor for Hip Replacement Bearings. ASTM International.

[2] Wang, F., Brockett, C. L., Williams, S., et al. (2008). Lubrication and Friction Prediction in Metal-on-Metal Hip Implants. Phys. Med. Biol., 53(5), 1277.

[3] Gao, L., & Hewson, R. (2012). A Multiscale Framework for EHL and Micro-EHL. Tribol. Trans., 55(6), 713-722.

Paper ID PO-11 **Programme ID** 77

Numerical modelling of the diffusive solidification of mushy layers with application to sea ice

Arthur Scott

A mushy layer is a two-phase, reactive porous medium, an example of which is sea ice (Feltham et al., Geophys. Res. Lett., 2006). Studies on mushy layer dynamics to date have typically utilised a continuum approach in which the mushy layer is modelled as a single phase whose physical properties depend on both the liquid and solid fractions. Here we present a new approach for modelling the growth of mushy layers using a combination of lattice Boltzmann and enthalpy methods. This approach allows us to resolve mushy layer dynamics explicitly on the small dendritic scale, and hence provides a more detailed picture of mushy layer evolution than the continuum approach does. We extensively validate results from our two-dimensional simulations of mushy layer growth in the absence of fluid flow with the observationally consistent continuum theory of Worster (J. Fluid Mech., 1986). We then explore the dynamics by varying key parameters — such as the temperature and concentration ratios and the ratio of thermal and salt diffusivities — to understand their effects on mushy layer growth and morphology. Preliminary results from our study suggest that our approach is well suited for exploring the interactions between shear and buoyancy-driven fluid flows and mushy layers.

Paper ID PO-12 **Programme ID** 78

Blind Estimation of the Arterial Input Function in DCE-MRI

Jake L Cray

Blind deconvolution in DCE-MRI can increase the accuracy of estimated perfusion parameters. However, there are many challenges in acquiring good quality validation data, such as backflow and partial volume effects. As an attempt to counteract these, a study was carried out where breast DCE-MRI images are captured using an additional back coil in order to provide high quality images of the descending aorta. Pairing this with a bookend correction (Cron 1999), allows for a confident measurement of the contrast agent concentration within the descending aorta (AIF), alongside high temporal resolution images of the breast tumours. Blind deconvolution is a technique that aims to estimate the AIF, from DCE-MRI images, without the use of arterial voxels. This poster describes the use of said data in validating a blind deconvolution tool, PerfLab, developed by Radovan Jiřík. The effect of various pharmacokinetic models on the blind estimation process will be compared.

Paper ID PO-13 **Programme ID** 85

A multiscale multiphysics simulation framework for predicting heat exchanger performance in hydrogen powered aircraft

Girindra Ramgobin

Heat exchangers are crucial components in a thermal management system, particularly in hydrogen-powered aircraft utilising fuel cells. In the dry wings of such aircraft, heat exchangers also have the added functionality of being load bearing. Simulating the thermal and mechanical performance of such components traditionally requires significant computational resources due to the extensive domain size and intricate geometries. Additionally, there is a lack of tools which is able to simultaneously handle these two disciplines and perform a comprehensive multiphysics analysis.

A conventional cross flow heat exchanger is made up of a pattern of geometries, repeated across multiple layers. This feature can be harnessed such that the information from one simulation of this pattern can be passed on to the neighbouring cells and used to predict the overall thermal performance of the heat exchanger. Established analytical methods, such as the effectiveness-NTU or Logarithmic Mean Temperature Difference methods rely heavily on empirical data but fail to accurately predict internal thermal distributions within the heat exchanger core. This study builds upon previous work by Ciuffini et al. [1] and Starace et al. [2], which demonstrated that multiscale methods can achieve high accuracy at a fraction of the computational cost.

A novel framework is introduced that integrates OpenFOAM and Python to create a multiscale, coupled thermofluid-mechanical model. This framework solves a conjugate heat transfer model using a partitioned approach and uses an upscaling algorithm to iteratively reconstruct the macroscale thermal distribution across a layer of the heat exchanger by adjusting the boundary conditions in each cell. The thermal strain generated from this temperature distribution is coupled with a solid mechanics model of the heat exchanger subjected to a point load. This model can be driving tool for heat exchanger optimisation studies.

[1] A. Ciuffini, A. Scattina, F. Carena, M. Roberti, G. Toscano Rivalta, E. Chiavazzo, M. Fasano, and P. Asinari, "Multiscale computational fluid dynamics methodology for predicting thermal performance of compact heat exchangers," Journal of Heat Transfer, vol. 138, no. 7, p. 071801, Jul. 2016. [Online]. Available: https://asmedigitalcollection.asme.org/heattransfer/article/doi/10.1115/1.4032980/375424/Multiscale-Computational-Fluid-Dynamics

[2] G. Starace, M. Fiorentino, M. Longo, and E. Carluccio, "A hybrid method for the cross flow compact heat exchangers design," Applied Thermal Engineering, vol. 111, p. 1129–1142, Jan. 2017. [Online]. Available: https://linkinghub.elsevier.com/retrieve/pii/S1359431116321755

Paper ID PO-14 **Programme ID** 88

Spatiotemporal Evolution of the Flow Instabilities in Stagnation Regions: The Effect of Flow Rate and Porous Geometry

Negar Razaghia and Mohaddeseh Mousavi Nezhad

Turbulent flow in porous media is a complex phenomenon often associated with high Reynolds numbers and inhomogeneous flow at stagnation zones. Stagnation zones cause high stretching and large tensile stresses, which play a crucial role in the formation and dynamics of vortices, inducing intense chaotic spatiotemporal fluctuations in the flow field. Abrupt changes in the geometry of porous media can result in flow separation and formation of recirculation zones, stemming from the competition between surface forces on solid boundaries and inertial forces in the turbulent flow, influencing larger-scale flow field. The onset, strength, scale, and dynamics of flow recirculation patterns are affected by the inlet flow conditions and the geometry of the porous medium.

This study investigates the occurrence and spatiotemporal evolution of vortices in stagnation regions during fluid flow in porous media with high Reynolds numbers. The Lattice Boltzmann Method (LBM) is used as a numerical method to simulate the physics of inhomogeneous flow in a homogeneous porous medium, which consists of solid obstacles in a rectangular domain. The flow field and vortical features were analysed in various porous structure configurations with different obstacle shapes, sizes, and spacings subjected to Reynolds numbers, encompassing flow regimes from laminar to turbulent.

The results demonstrate that the vortical flow features are strongly affected by the interplay between inlet flow boundary conditions and the geometry. Formation of vortices and flow instabilities in the flow field at elevated Reynolds numbers results in an additional pressure drop and reduces the apparent permeability, highlighting the significance of here-simulated flow features on the transport and mixing characteristics. We show that either a symmetric or an asymmetric vortex rotation in the pore-scale can be formed beyond a critical value of inlet flow velocity, depending on the configuration of porous structures.

Paper ID PO-15 **Programme ID** 127

Interaction of Sedimenting Semi-Flexible Fibres in Stokes Flow

Nasrollah Hajaliakbari, David Head and Oliver Harlen

In some processes for material or composite making such as papermaking, cylindrical fluctuating semiflexible fibres sediment in the flow due to gravitational forces imposed on them. The speed of this process depends on the competition among the viscous, elastic and Brownian forces and hydrodynamic interactions between fibres. A precise numerical tool that can evaluate the deformation and motion of these fibres would be useful in the design of novel fibre materials.

In this research study, Slender Body Theory (SBT) with the Rotger-Prager-Yamakawa (RPY) singularity representation have been implemented to simulate the fluid-structure interaction of immersed semi-flexible fibres under gravitational forces within the flow.

For rigid fibres, the model results have been quantitatively validated against the known solution of Jeffrey Orbits and one flexible fibre sedimentation. For a pair of semi-flexible fibres, it is found that the fibres undergo different shapes, configurations and the sedimentation velocity depends on the bending stiffness, length, diameter of the filament, strength of gravitational forces, the distance between individuals in a pair and persistence length. In the poster, we will present the latest results investigating the roles of fibre configuration, elasticity, hydrodynamic interaction and Brownian motion.

Paper ID PO-16 **Programme ID** 49

Initial Sizing of Conventionally Configured Hydrogen-Powered Commercial Aircraft

Abdullah Mejbil

Commercial aircraft are highly e>icient transport which play a significant role in the world economy. The aviation industry has boosted economic activity but has also led to increased emissions. Technological advancements have reduced emissions, yet this falls short of net-zero goals, and this drives the need for sustainable aviation. Hydrogen-powered aircraft have the potential to significantly reduce climate impact. Optimised conventional aircraft designs have evolved through years of experimentation and tool development, but frameworks for sizing hydrogen-powered aircraft designs do not exist. Liquid hydrogen requires four times the storage volume of kerosene to hold the same amount of energy, current tools must be adapted to consider this and changes to on board systems within aircraft sizing. This study aims to develop a toolkit for the initial sizing of hydrogen powered conventionally configured commercial aircraft using the OpenConcept software package. Key developments are delivered for modelling of propulsion, aerodynamics, and weights for a hydrogen powered concept in the conventional configuration. Thrust-specific energy consumption is maintained for the same mission specification as used to drive a kerosene fuelled aircraft using scaling arguments. Sizing of storage tanks and systems for liquid hydrogen are also included within the toolkit. Outcomes provide new insights into the feasibility and sizing of hydrogen concepts through robust calculation, moving beyond current understanding of the future of sustainable aviation conceptual design.

Programme ID	Paper ID	Author
AC-1	93	Muhammad Nuramirul Hijjaz
AC-2	110	Saikat Datta
AE-1	8	Henry Lizcano
AE-2	55	Burak Turhan
AE-3	61	Shanshan Xiao
AE-4	114	Bryn M.F. Jones
AE-5	72	Charles A Proe
AI-1	20	Usamah Adia
AI-2	24	Martina Formichetti
AI-3	38	Miranda J S Horne
AI-4	65	Harshinee Goordoyal
AI-5	66	Christian M Toma
AI-6	116	Agustina Felipe
AI-7	50	Jose Florido
AI-8	58	Ankan Banerjee
AI-9	98	Seun Coker
AN-1	36	Amanda S M Smyth
AN-2	43	Jack Keeler
AN-3	62	Christian A Jones
AN-4	64	Aaron D'Cruz
AN-5	74	Henry W Writer
AN-6	86	D. Wall
AN-7	100	Josiah-Shem Davis
AN-8	128	Ciara A Higham
AN-9	95	Ibrahim Abubakar Masud
AN-10	70	Phil Trinh
BI-1	73	Gregory Holba
BI-2	89	Isla Henderson
BI-3	97	Cristina Teleanu
BI-4	99	Emily J Butler
BI-5	117	Ourania Giannopoulou

Programme ID	Paper ID	Author
CA-1	23	Meg Richards
CA-2	106	Nick Creasy
CA-3	30	Josh Shelton
CA-4	46	Ahlem Mokhtari
CA-5	63	Kat Phillips
CA-6	69	Yatin Darbar
CA-7	83	Jack R Panter
CA-8	94	Duncan Dockar
CA-9	96	Benjamin Owen
CA-10	101	Alexander Saal
CF-1	109	Jesse Taylor-West
CF-2	42	Ahmad Mohamadiyeh
CF-3	48	Freya C Bull
CF-4	52	Federico Peruzzini
CF-5	53	Rebecca J Hill
CF-6	82	Isabel F Latimer
CF-7	90	Andrea Sendula
CF-8	39	Anushka Herale
CS-1	7	Mohamed Ashar Sultan Mohamed Yousuf
CS-2	26	Jörg T Sommerau
CS-3	56	Abhimanyu Gaur
CS-4	68	John M Lawson
EX-1	51	Murilo Cicolin
EX-2	104	Bappa Mitra
EX-3	54	Elias Arcondoulis
EX-4	57	Elias Arcondoulis
EX-5	67	Hulya Biler
EX-6	71	Tomos Rich
EX-7	75	Ian Masters
EX-8	84	Filipa Adzic
EX-9	91	Jan W Modrzynski
EX-10	21	Zuhaib Nissar
EX-11	22	Mostafa Soroor

Programme ID	Paper ID	Author
FD-1	14	André Lopes
FD-2	16	Reece D Luetchford
FD-3	27	Jo Samuel
FD-4	81	Jonathan Sewell
FD-5	113	Qiming Yu
FD-6	59	Zhaoxin Ren
FD-7	79	Jac Clarke
FD-8	87	Nicholas J Copsey
FD-9	92	Raahil Sanjay Nayak
FD-10	112	Scott Bennie
FD-11	124	Joseff Parke Sturrock
FD-12	121	Callum D Lock
FD-13	17	Chigozie Okwudiri Eleghasim
FD-14	19	AMIT KUMAR SAINI
FD-15	31	Julie Y Frank
FI-1	15	Shailesh Naire
FI-2	118	Rhiannon Nicholls
FI-3	125	Conor James Nolan
FI-4	123	Miles Morgan
FI-5	29	Kasia Nowakowska
FI-6	122	Ryan Doran
FI-7	80	Jo J Kershaw
FI-8	120	Azza Al gatheem
OP-1	45	Morgan T Taylor
OP-2	37	Luke Driver
TU-1	10	Dea D Wangsawijaya
TU-2	47	Andrew McMullan
TU-3	102	Mridu Sai Charan Arukalava Seshasayee
TU-4	105	Xu Chu
TU-5	119	Laura Irvine
TU-6	126	Jan Dobrzycki
TU-7	32	Mariadebora Mauriello
TU-8	12	jason Ferguson

Programme ID	Paper ID	Author
TU-9	25	Xiaodong Li
TU-10	33	Takfarinas Medjnoun
TU-11	34	Prateek Jaiswal
TU-12	35	Thomas D Preskett
TU-13	40	Aan Yudianto
TU-14	41	Max C Walker
TU-15	44	Jacqueline M Mifsud
PO-1	9	James Afful
PO-2	103	Josiah-Shem Davis
PO-3	107	Josh Parkin
PO-4	108	Asif Mushtaq Ahmed Ansari
PO-5	111	Alexandra J Hardy
PO-6	115	Ganesh Sahadeo Meshram
PO-7	13	Abdulaziz H Alharbi
PO-8	18	Danny A P Blundell
PO-9	28	Marco Virgilio
PO-10	76	Robin Furze
PO-11	77	Arthur J Scott
PO-12	78	Jake L Cray
PO-13	85	Girindra Ramgobin
PO-14	88	Negar Razaghi
PO-15	127	Nasrollah Hajaliakbari
PO-16	49	Abdullah Mejbil