EPFDC11 Timetable

Tuesday 7th July

11am Tea/Coffee break
11:30am Workshop with Dr Kat Phillips (part one)
1pm Lunch
2pm Workshop with Dr Kat Phillips (part two)
3:30pm Tea/Coffee break
4pm Dr Eric Hester
5:00pm *Mini-break*
5:10pm Contributed talks - Numerics
6:15pm Posters & drinks reception
7:30pm End of day

Wednesday 8th July

9am Contributed talks - Waves
11am Tea/Coffee break
11:30am Contributed talks - Aerodynamics
12:50pm Lunch, *food served at 1pm*
1:50pm Contributed talks - Instabilities and transition
3:30pm Tea/Coffee break
4:00pm Dr Michael Negus
5:00pm *Mini-break*
5:10pm Contributed talks - Phase change and boundary layer flows
5:50pm End of day, followed by conference dinner

Thursday 9th July

9:00am Contributed talks - Thin films and interfacial flows
11:00am Tea/Coffee break
11:30am Dr Anna Kalogirou
12:30pm Lunch, *food served at 1pm*
1:50pm Contributed talks - Microfluidics and applications
3:30pm Tea/Coffee break & end of day

Plenary talks

Dr Eric Hester (Bath) | Automating Differential Geometry For Fluid Mechanics

Physical laws are coordinate-invariant, but practical computations are not. Good coordinates and frames can vastly simplify calculations, speeding up numerical simulations as well as facilitating rigorous proofs. But this doesn't make them easy to use. Though the chain rule and vector calculus are sufficient in principle, problems involving nonstandard geometries, curvilinear coordinates, or moving interfaces can rapidly spiral in complexity. Coordinate expansions become a major source of errors and lost time. Computer algebra systems help, but functionality for general geometries isn't "out-of-the-box". Other higher-level packages like xAct are powerful, but are often designed for index calculations in general relativity rather than typical continuum-mechanics PDEs with boundaries and constraints. There is a missing middle of tools to automatically convert systems of PDEs on nonstandard geometries to their concrete component forms.
I will introduce a small Mathematica package, Tensors, that aims to close this gap. Given coordinate mappings, tensor fields, frames, and metrics on manifolds and their boundaries, the package automatically translates expressions composed of standard differential operators and geometric data (e.g. gradients, integrals, curvatures, normals etc.) into component form. Automating this conversion makes complex calculations faster, more reproducible, and easier to generalise. I will illustrate with several application from multiphase fluid dynamics, showing how the package streamlines geometric constructions needed for high-order solvers for various free-boundary problems.

Dr Michael Negus (Vanellus) | CFD in Academia, Industry and Startups

In this presentation, I will present my experiences transitioning from a PhD student in applied mathematics into founding a CFD startup. I will start by discussing my research background during my PhD, using analytical and numerical methods to model droplet impact at Oxford. I will then explain my transition into industry, specifically founding my startup, Vanellus, developing GPU-accelerated CFD tools. The talk will focus on what I've learned on the differences, and similarities, between how CFD is conducted in academia and industry; and how to foster greater collaboration and knowledge transfer between the two.

Dr Anna Kalogirou (Nottingham) | Following the flow across scales: Building a career in Fluid Dynamics

This talk will share insights from my journey as a researcher in fluid dynamics. My research addresses physically- and industrially-motivated problems found across a range of scales: from the micro-scale, e.g. the flow of thin liquid films, to the macro-scale, e.g. extreme waves in the ocean. These phenomena present rich mathematical challenges and have important real-world applications, from industrial processes to natural flows. My research follows a synergistic approach that combines mathematical modelling, asymptotic analysis, and numerical computations to tackle complex problems involving moving interfaces or free surfaces. I will discuss how this integrated methodology has shaped my research trajectory, highlight key results and applications, and reflect on strategies for building a sustainable and impactful research career. The presentation aims to provide both technical insights and practical guidance for early-career researchers interested in fluid dynamics and related areas.

Numerics

David Tudor (Bath) | Are thermodynamically derived phase-field models better?

Phase-field models can predict the evolution of a multi-phase system over time and feature a smooth interface between phases rather than a sharp transition. There exist several formulations of such models when applied to an ice-water system, but they can be grouped into two types: those which are thermodynamically derived, and those which are not. We compare these two types and investigate the impact of thermodynamic consistency by measuring the convergence of each model to their sharp interface limit using the spectral code Dedalus. Finally, we test whether the models agree in the asymptotic sharp limit and show which type performs better.

Zachary Candelaria (Edinburgh) | The Finite-Element-Lattice-Boltzmann Method for Two-Phase Flow Simulation

Here we present a novel computational framework for simulating dynamic wetting and contact-line motion by solving the discrete Boltzmann equations using the finite-element method. Unlike conventional lattice-based approaches, the proposed method combines kinetic-theory-based modeling with the geometric flexibility of the finite-element method, enabling accurate treatment of complex boundaries and unstructured meshes. Numerical experiments demonstrate the ability of the approach to reproduce key wetting phenomena, including droplet spreading and contact-line evolution.

Nawal Alshahrani (Exeter) | Spectral Optimisation of Interpolation and Time Integration for Computational Fluid Dynamics

High-order numerical schemes are widely used in computational fluid dynamics to improve accuracy. However, a high formal order of accuracy does not necessarily guarantee an accurate representation of small-scale flow features, which are associated with the high-wavenumber region of the spectrum. This limitation motivates the use of spectral optimisation techniques that directly target the numerical behaviour across a range of scales. In this work, a spectral optimisation framework is developed for the construction of numerical schemes used in advection problems. The approach is applied to both interpolation operators and explicit time-integration schemes. In each case, numerical coefficients are determined by minimising a spectral error measure, with the aim of improving the representation of waves over a prescribed range of wavenumbers or frequencies. For interpolation, optimised schemes are compared with

standard methods, including cubic Lagrange interpolation, using spectral-error and L2-error diagnostics. Repeated application of the interpolation operator is used to mimic its role in semi-Lagrangian advection methods and to investigate the accumulation of error over time. For time integration, the same optimisation philosophy is applied to the construction of stability polynomials, and the resulting schemes are assessed through analyses of amplification, phase accuracy, and long-time behaviour. The results show that spectral optimisation can improve the spectral properties of both interpolation and time-integration schemes and reduce error over selected ranges of wavenumbers and frequencies. However, repeated application highlights the importance of stability and amplification control, demonstrating that improvements in spectral accuracy do not necessarily translate into improved long-term performance. Overall, the study illustrates the potential of spectral optimisation as a unified framework for the design of numerical schemes for advection problems, while emphasising the need to balance accuracy and stability in practical computational fluid dynamics applications.

Waves

James Bell (Nottingham) | Experimental study of the structure of drain vortices in rotating flows

Vortices in rotating systems are widely observed in both natural and industrial settings and are of particular relevance to geophysical flows such as mesoscale ocean eddies and atmospheric vortices. Such systems can often be described as having low Ekman and Rossby numbers, meaning the Coriolis term dominates over the viscous and non-linear effects respectively. An asymptotic solution for an axisymmetric drain vortex in a rotating system exists in literature for small Rossby and Ekman numbers. The overall flow structure predicted matches that found in previous experiments and it agrees quite well quantitatively with limited data from previous studies, which have few data points and do not completely match the conditions of the theory. The work presented here reports laboratory experiments run specifically to provide detailed comparison with these asymptotic solutions. Velocity fields are measured using particle image velocimetry in experiments that more closely match the theoretical conditions. Initial results show qualitative agreement, while measurable quantitative discrepancies in velocity measurements have been observed and are the subject of ongoing investigation. This work is part of a PhD project working towards the novel experiential study of a drain vortex in a two-layer stably stratified rotating fluid.

Nathan Boachie (Queen Mary) | Computational modelling of weak shock wave-gas bubble interactions in water

The interaction of a shock wave with a gas bubble in a liquid medium is central to engineering applications such as shock wave lithotripsy and cavitation damage. Numerical simulations and experimental observations have observed that the interaction of a shock wave with a bubble leads to the formation of a re-entrant jet for amplitudes above approximately 10 MPa. However, the interaction with waves of lower amplitude remains less well-studied. In this study, the response of a single bubble to low-amplitude, infinite-width shock waves is investigated through numerical simulations. We characterise the collapse across a range of shock strengths and observe a collapse regime in which the re-entrant jet grows during the bubble's rebound. We then explore the influence of surface tension and viscosity on the formation of the re-entrant jet for bubbles of different diameters. Finally, we examine the early stage movement of the bubble's interface using both instability models and a boundary-integral method to demonstrate that the early stage movement of the bubble's interface is dominated by the sustained acceleration provided by the infinite-width shockwave. Our simulations suggest that single bubbles impacted by shock waves of infinite width form a re-entrant jet at much lower amplitudes than previously reported.

Kelvin Xie (UCL) | Parametrisation of an intermittent gravity wave source in a simple model of the quasi-biennial oscillation

Parametrisations of unresolved gravity waves used in general circulation models can be made more computationally efficient by introducing a stochastic component to the forcing. An additional advantage of introducing stochasticity is that intermittency associated with the scheme could be tuned to resemble the intermittency of observed gravity wave sources, and could therefore act to improve the physical fidelity of the scheme. Here, it is argued that using stochastic differential equations (SDEs) to drive the stochastic component provides a natural general framework to develop such schemes.

Jinhui Gong (UCL) | The rotating stratified waves

The arbitrary finite amplitude rotating stratified waves for inviscid and incompressible fluids are considered. The stratification is assumed to be valid under the Boussinesq ap- proximation. The linear problems in two and three dimensions under constant buoyancy frequency are first constructed, including the external and internal modes, in the physical frame. In the nonlinear problem, the primary objects of study are two-dimensional waves of the external modes, which are solved numerically in a co-moving rectangle frame. Prior to this, the linear solutions in this frame are obtained analytically as the numerical initial guess, along with the corresponding internal-mode solutions. The results are shown under three specific types of stratification, no stratification, constant stratification, and a two-layer constant-density structure.

Daire O'Donovan (Dublin) | Optimising Wave-Driven Propulsion

Horizontal locomotion of a body on the fluid surface can be achieved by interacting with self-generated waves via a vertical bobbing motion. Mathematically, this can be interpreted as a pressure source acting on the surface. To study the conditions for maximal thrust in a chosen direction, an optimal control problem can be posed, where the pressure source is the control and the thrust force is the objective. A bound is then applied to the control to regularise the problem. The work is split into two cases given two different bounds, firstly the norm of the control function, and secondly the power, which is derived from the rate of change of the energy. To begin this work, the problem is reduced to a one dimensional set-up in the shallow limit. Given the assumption that the pressure source is periodic, an analytical approach can be taken with variational calculus. Numerical optimisation can be carried out to calculate the optimal pressure given a constraint and can be shown to match the analytical solution. Utilising an optimal solution in each case, achieving maximal efficiency is found to be dependent on emitting a wave purely in the direction opposite to movement. With the same bounds on the control function, the work is extended to a second spatial dimension. Analytical and numerical optimisation approaches are taken again to validate further numerical investigations for a two dimensional shallow water surface.

Gianni Cassoni (Roma Tre) | Curvature-Induced Modal Coupling in Microgravity Sloshing

Curved equilibrium menisci can change the sloshing spectrum of partially filled cylindrical tanks in microgravity even in the linear regime. A semi-analytical boundary-operator formulation shows that the classical flat-interface problem is recovered as a limit, while curved menisci introduce coupling between radial modes and shift the low-order frequencies. For low Bond number, curvature becomes a leading-order effect: concave menisci soften the fundamental mode, convex menisci stiffen it, and both alter the spatial structure of the modes. The resulting frequency shifts and mode mixing are important for spacecraft applications because they modify the slosh signatures used in reduced-order models for guidance, navigation, and control.

Aerodynamics

Hibah Saddal (Birmingham) | Gust Response of Morphing Wing Sections

Small unmanned aerial vehicles (sUAVs) are susceptible to unsteady environments, particularly when the scale of flow disturbances approaches that of the vehicles' airspeed [1]. In such scenarios, gust encounters can induce substantial flow separation over aerodynamic surfaces, resulting in degraded aerodynamic performance and reduced flight stability. To address this, avian-inspired wings with variable flexibility, motivated by the adaptive morphing mechanisms observed in bird flight have emerged as a promising approach for improving aerodynamic efficiency across a broad range of operating conditions [2]. This study presents a numerical investigation into the response of flexible wing-sections subjected to translational gust profiles characterised by varying accelerations. A two-way coupled fluid-structure interaction framework is employed using a partitioned strong coupling strategy. The fluid flow is governed by the incompressible Navier-Stokes equations and solved in OpenFOAM using the finite volume method. Mesh deformation is managed through a quadratic inverse-distance diffusion-based dynamic meshing technique. The structural response is resolved in CalculiX using the finite element method. The data exchange between the fluid and structural solvers is achieved through preCICE, with radial basis function interpolation ensuring accurate data exchange across the mesh interfaces. A parallel implicit coupling scheme, accelerated using the Interface Quasi-Newton Inverse Least Squares (IQN-ILS) scheme is employed. The study examines the influence of wing geometry, bending rigidity, and trailing-edge flexible segment size on aerodynamic performance and the gust response. Results are compared with those obtained for comparatively rigid wing configurations. The findings aim to inform the development of next-generation bio-inspired sUAVs capable of improved aerodynamic performance and stability in gusty flight environments.

[1] Jones, A. R. (2020). Gust encounters of rigid wings: Taming the parameter space. Physical Review Fluids, 5(11), 110513.
[2] Yudin, D., Floryan, D., & Van Buren, T. (2023). Propulsive performance of oscillating plates with time-periodic flexibility. Journal of Fluid Mechanics, 959, A31.

Shuhan Guo (Southampton) | Flow evolution over an accelerating aerofoil

This study numerically investigates the unsteady aerodynamic response and flow evolution of a 15% thick symmetric Joukowski aerofoil under fifth-order polynomial acceleration at a Reynolds number of 100,000 and a fixed angle of attack of 6°. By varying the acceleration duration Tac, this work isolates the effects of kinematic timescales on transient flow separation behaviour. Unsteady aerodynamic lifts are modelled via Wagner's theory and Duhamel's integral to accommodate gradual acceleration conditions. The results show that aerodynamic lifts are initially dominated by added-mass contributions, which rapidly decay and transition to circulatory-dominated loading alongside viscous-induced oscillatory behaviour throughout the transient stage. A linear correlation with a consistent proportionality coefficient of 0.6 is identified between Tac and both the onset and termination of the transition regime. The transition process is further decomposed into two successive instability stages. The 2D stage is governed by the topological evolution of the separation zone, including periodic breathing of the primary separation bubble and discrete generation, accumulation and shedding of secondary bubble structures. The onset of 3D spanwise instabilities proves highly sensitive to acceleration magnitude. Fast acceleration induces dual-source instability originating simultaneously within primary and trailing-edge secondary bubbles, while slow acceleration yields a single transition driven by intrinsic spanwise deformation of mature secondary structures. This work establishes a unified mathematical relationship between acceleration kinematics, separation topology and unsteady force evolution, providing new physical understanding of the nonlinear transient mechanisms of accelerating aerofoil systems at moderate Reynolds numbers.

Arjun Shergill (Birmingham) | Development of a GPU-Accelerated Flow and Fluid-Structure Interaction Solver for Simulating Bio-Inspired Flight

Insect flight has been the focus of extensive research due to its exceptional manoeuvrability at low Reynolds numbers, enabled by employing unsteady vortex dynamics and specialised wing morphology. Insect wings are also capable of significant camber and twist deformation, which are influenced by their shape and size. While previous studies have explored wing kinematics and overall wing shape, and their impact on the flow field, comparatively little attention has been given to the role of wing deformation. Recent work has involved the application of a novel in-house Immersed Boundary Method (IBM)/overset flow solver to study both individual Butterflies and Butterfly groups using GPU acceleration on Baskerville. Butterflies are of particular interest because they operate at unusually low flapping frequencies with large wing sizes, and can change their aspect ratio and wing sweep angle simultaneously of these wings. As a result, their wings are likely to experience significant deformation during flight, highlighting the need for detailed investigation into the effects of structural flexibility on the surrounding flow field and aerodynamic forces. The present study focuses on coupling CalculiX, a three dimensional structural finite element program, with the in-house flow solver, enabling future investigations into bio-inspired Fluid-Structure Interaction (FSI) effects associated with wing flexibility. Preliminary simulations have been conducted on a perpendicular flexible beam to demonstrate the coupling approach.

William Skipwith (Manchester) | Instabilities and their mitigation in high Reynolds number boundary-layer flows

Boundary layer receptivity is the process by which external (or artificial) disturbances excite instabilities within the boundary layer. These instabilities are known as Tollmien-Schlichting (T-S) waves and are one of the more common routes to transition to turbulence. Here we study T-S waves that arise from the interaction between a free-stream acoustic disturbance and some steady surface roughness/injection/heating. The nonlinear receptivity problem derived is similar to that of Ruban (Fluid Dyn., vol. 19, 1984, pp. 709-717) on the generation of T-S waves by sound, but with compressible effects included. The linearised problem is solved both analytically and numerically similar to that of Brennan et al. (J. Fluid Mech., vol. 909, 2021, A16) on T-S wave cancellation via heating elements. In particular, the analytical solution gives rise to a heating profile, dependent on the surface roughness/injection, that leads to no pressure perturbation meaning no T-S waves produced. Numerical results show the predicted stable, neutral, and unstable frequencies of acoustic noise as well as demonstrating heating cancellation using an approximation of the exact analytical profile. Results for weakly-nonlinear and fully nonlinear amplitudes of acoustic noise are also shown.

Instabilities and transition

Adam Hanlon (Belfast) | Mixing of the Richtmyer-Meshkov Instability under anisotropic strain

Mean flow strain has a significant impact on the evolution of the three dimensional multi mode Richtmyer Meshkov instability. Previous studies have analysed the effects of the application of strain either in the axial (perpendicular to the mixing layer plane) or transverse directions separately. Results indicated agreement with the linear theory at early time, and simulation data was used to propose modifications to a buoyancy drag model and a URANS model at late times, which model the observed behaviour well. This research differs by applying mean compressive strain rates in two and all orthogonal directions, with the goal to provide fundamental insight on how the mixing properties evolve under conditions similar to cylindrical and spherical compression. Each compared to a layer with no mean strain. Result indicate that axial compression dominates integral width predictions, but with important distinctions in flow physics between the other compression cases.

Rumesh Sudhaharan (Warwick) | On thin ice: Computational modelling of an unstable interface

Understanding the dynamics of ice formation is important in many real life applications. An example is ice accretion on aeroplane wings, where liquid droplets below the melting point freeze to form ice crystals on the wings. Modelling ice growth in such conditions presents a challenge, as we have a Stefan-type problem, with a moving boundary and an unstable ice-water interface. The ice crystals form dendritic shapes due to an instability at this interface, called the Mullins-Sekerka instability. We attempt to mathematically model and build an associated numerical simulation to study this instability by solving the two-dimensional heat equation using the Finite Element Method. We use an interface tracking method to investigate the shape of the interface and study its evolution with time. We verify our numerical results to ensure robustness and mesh independence, as well as validate against analytical solutions derived based on a linear stability analysis procedure.

Dimitrios Michail Kyriakou (Coventry) | Linear stability of compressible boundary layers around a rotating cone

In this work, the stability of compressible boundary layer around a rotating cone is investigated using local linear stability theory. Both temporal and spatial formulations are considered in order to examine the eigen-spectrum, modal growth rates, neutral stability characteristics and critical Reynolds numbers of the flow. The study focuses on the influence of the Mach number, the cone half-angle, the rotation rate, and the wall temperature on the onset of instability. The boundary-layer base flow is first computed in conical coordinates using a Crank-Nicolson-type scheme, after which the eigenvalue problem is solved using a fourth-order finite difference scheme. The results show that the effect of each parameter is strongly mode-dependent, and not uniform across the spectrum. Individual unstable branches are therefore tracked as the parameters are varied, allowing the corresponding changes in growth rates and critical Reynolds numbers to be compared directly.

Carlos Martinez Otaduy (Catalunya) | Symmetry breakings and novel Routes to Chaos in a Varicose Channel

An analysis of the flow in a 2-D channel with symmetric wavy walls is presented. Previous studies have dealt with the transition to chaos via time integration, but missed studying the bifurcations and symmetry breaking of the system. In the present work, linear stability analysis has been performed on the basic, symmetric, flow, resulting in bifurcations of different nature, which give rise to different families of solutions. Stable invariant sets have been converged via time-integration; Newton-Krylov methods have also been utilized to converge both stable and unstable solutions, and to continue them using pseudo-arclength techniques. The system traverses a variety of states, from the breaking of the symmetry in favor of an asymmetric stationary flow, a later periodic state, which evolves into a quasiperiodic one and finally reaching a chaotic state, which reconnects the two branches that arose from the symmetry-breaking. The linear stability of all these states has been thoroughly studied using both types of invariants, eigenvalues and Floquet multipliers, providing a complete picture of the system evolution. An interesting Hopf, period-doubling, Neimark-Sacker sequence has been found, as a variation of the more common Ruelle-Takens-Newhouse route to turbulence.

Nestor Garcia-Cabrera (Dublin) | Statistical behaviour of triad phases in 3D shell models of turbulence

Shell models of turbulence have been used for many years to study the fundamental mechanisms of turbulence [1,2]. They are especially useful in the analysis of energy transfer and intermittency in fully developed hydrodynamic turbulence as they allow us to study different scales with ease. We examine the behavior of the Sabra shell model, described by the system of equations for the complex velocity variables $(u_n)_{n=1}^{N}$:

$$\begin{aligned}\frac{du_n}{dt} =& i(ak_{n+1}u_{n+2}u_{n+1} + bk_n un + 1u_{n-1} - ck_{n-1}u_{n-1}u_{n-2}) \\ & - \nu k_n^2 u_n + f_n\,, \quad n = 1, \cdots, N,\end{aligned}$$

in the context of three-dimensional hydrodynamic turbulence (the constants

a, b, c are appropriately chosen and the constant forcing term f_n is nonzero for the first shell $n = 1$ only)[3]. We focus particularly on the dynamics of triad phases, $\theta_n := \phi_{n+1} - \phi_n - \phi_{n-1}$, where ϕ_n is the complex phase in the polar decomposition of the complex velocity variable, $u_n = a_n e^{i\phi n}$. We demonstrate through numerical simulations that the triad phases tend to cluster around particular values (as in the Burgers equation [4]), reflecting the fact that energy cascades from small values of n to larger values of n. Using a new simplified model for the dynamics of the triad phases, we apply stochastic methods to find the triad phases' evolution in the inertial range, thus characterizing the triad phases' probability density functions across all scales.

Building upon the equation of motion for the Sabra shell model, we derive an equation of motion for the triad phases. The equation for θ_n contains nonlinear terms from neighboring triad phases at $m \neq n$, making its analysis difficult. After a series of justified assumptions, we model these neighboring triad terms as a single noise term $\xi_n(t)$. This reduces the equation for the θ_n into a stochastic differential equation of the form $d\theta_n/dt = F_n(\theta_n) + \xi_n(t)$, where F_n is a deterministic function. This choice leads to a stochastic modelling challenge, where the behavior of the noise term plays a crucial role in determining an appropriate method for obtaining a probability density function. Through the assumption of Gaussianity of the noise term we develop an extension to the Ornstein-Uhlenbeck noise models by adjusting the autocorrelation function of the noise, which is now described as

$$< \xi_n(t)\xi_n(s) > \quad \propto \quad \exp\left(- \left(\frac{|t-s|}{\tau_n} \right)^{p_n} \right) , \quad \tau_n > 0 \, , \, p_n > 0 \, , \tag{1}$$

where the parameters τ_n and p_n can be obtained numerically by a fitting procedure. From this, incorporating approximations similar to the ones used by J.M. Sancho et al. [5], we obtain analytical formulae for the probability density functions of the triad phases that closely match those obtained independently through numerical simulations.

In summary, we have developed and validated a working hypothesis regarding the separation of the rate of change of the triad phases into "deterministic" terms plus "noise" terms, which results in a better understanding of the dynamical mechanisms that shape the triad phases' preferential values and the corresponding energy cascade directions. The presence of autocorrelation in the noise suggests that the system has strong memory components. Preliminary findings further indicate that the characteristic time, τ_n, is inversely proportional to the shell number n. This decreasing trend implies that noise becomes increasingly random at higher shell numbers, potentially explaining the increased clustering around the system's stable fixed points.

The study of triad phases is crucial, as they play a key role in governing energy transfer between shells, directly influencing the cascade mechanism in turbulent systems. By accurately modelling triad phases, we can achieve precise predictions of energy flux, providing deeper insights into turbulence and its underlying dynamics.

Phase change and boundary layer flows

Jatinder Pannu (Keele) | The spreading and cooling of molten fluid flows

Understanding the dynamics of molten fluids as they spread and cool is important in numerous situations. A few non-exhaustive examples include the evolution of lava domes, the spreading of molten glass, or radioactive isotopes (corium) in the event of a nuclear meltdown. For these examples at these scales, the rate at which new fluid/material enters the system, the temperature of the fluid itself, and gravitational effects, all play a key role. Investigating flow scenarios where the spreading lengths are orders of magnitude larger than fluid thickness, we are able to employ a long wavelength approximation to simplify and reduce the governing equations to a set of coupled equations for fluid thickness (/free surface) and temperature. The coupling of the equations is through a temperature-dependent viscosity. We obtain solutions of the coupled system numerically, and the results are discussed for a systematic variation of parameters.

TD Dang (Warwick) | Three-dimensional modelling of ice melting in uniform shear flow

The erosion of a slender three-dimensional lump of ice lying on a solid cooled surface within an incident water boundary layer is investigated. The physical model used is based on features for large Reynolds number flow. Two-way coupling occurs between the evolving ice shape and the water flow near the surface, involving a local three-dimensional interactive boundary layer formulation and a latent heat-transfer representation in the movement of the melting ice boundary (Here the cooled patch of surface is of fixed size and remains cold even after the ice has melted). Analysis is applied to determine the incident shear and heat transfer factors and especially to predict shape effects, with particular emphasis first on initially circular or elliptical planforms of varying aspect ratio and, second, on general very thin planforms. The times taken to complete erosion are determined along with the evolution of the ice shapes, the induced surface shear stress and pressure, the subsequent generation of upstream influence, and the effects of incident heat transfer. Persistence of the ice coverage is also found with surface undercooling applied, leading to a limiting shape of ice. An explicit solution is derived for the case of a thin initial ice lump aligned with the incident water flow.

Thin films and interfacial flows

Merete Seyfried (Darmstadt) | Electrolyte-filled Nanochannels: Increasing the Thermoelectric Response

While energy demands are rising, low-grade waste heat is in abundant supply. This waste heat provides a temperature gradient and could be converted into electric energy using electrolyte filled nanochannels. In such nanochannels, a combination of electrokinetic and confinement effects induces charge transport and thus an electric current. However, for a significant effect, configurations with a strong thermoelectric response are needed. We show that using weak electrolytes – instead of strong electrolytes – as working fluids in nanoconfinement is expected to lead to higher thermovoltages. While strong electrolytes fully dissociate into anions and cations, weak electrolytes only dissociate partially. The degree of this partial dissociation is temperature dependent. This increases the thermoelectric response of nanoconfined weak electrolytes compared to their fully dissociated counterparts. In this talk, we will discuss the physical mechanism behind thermoelectricity in nanochannels and present our analytical and numerical research on confined weak electrolytes. With this, we strive towards better understanding and therefore towards enhancing charge transport in electrolyte-filled nanochannels for thermoelectricity. This project has received funding from the European Union's Horizon 2020 research and innovation programme under Grant Agreement No. 964251.

Jishnu Goswami (Manchester) | Curling of Elastic ribbons in low *Re*

Curling is a common feature of rapid movements in biological systems. It is a mechanism that involves converting the stored elastic energy into kinetic energy, which governs processes such as explosive seed dispersal in plants. Similarly, curling governs the rupture dynamics of the membranes of red blood cells infected by malaria, which occurs on a timescale of 100 milliseconds. In this case, membrane curling occurs in a low-Reynolds-number environment, and fluid viscosity prevents rapid curling, thereby affecting the subsequent release of parasites. In this study, we are developing an analogous experimental process that involves the curling of naturally curved ribbons submerged in a highly viscous liquid. The pre-curved ribbon is pulled taut within the liquid, and once released, it starts to form curls by subtending an acute angle with the vertical, with a distinctly bigger outer curl followed by multiple tightly formed inner curls. Our results show that the ribbon moves through the liquid with a uniform velocity, which increases with a decrease in the natural radius of curvature but is independent of the ribbon width and a curling angle that is constrained by the geometry of the ribbon. We have derived a scaling relationship for the curling velocity by balancing the elastic and viscous forces on the ribbon, which elucidates the effect of various parameters on the process. We observe two different regimes of curling when the ribbon moves near a boundary, whereby the effective drag changes from a Stokes drag to a form mediated by a thin lubricating layer between the curling roll and the boundary. Together, these results establish a simplified physical picture of the process, which is controlled by the mechanics of the curled ribbons involving elasto-viscous interactions relevant in low Re environments.

Owen Nicholls (Nottingham) | Solidification of Rotating Sessile Water Droplet

This project explores the solidification of sessile and impacting water droplets and subsequent behaviour of a flowing water surface layer. Motivated by experimental observations, to simulate the accelerating boundary layer and rotational effects experienced as ice forms on a wind turbine blade, we consider a rotating free-surface contact problem. The freezing process of a sessile drop experiencing rotational forces is captured from images of the final accretion shape. A constant flux of water is introduced to flow over the surface, mimicking runback flow created by airborne water droplets impacting with a turbine blade. Mathematical modelling of this problem is two-fold, the shape of the solidified drop is found through boundary integral methods, incorporating the effects of centrifugal and Coriolis forces arising from the rotation and verified by experimental measurements. Following this initial solidification process, the flow of a thin fluid film over the curved substrate is modelled using thin film theory. Linear stability analysis is employed, investigating how rotational effects influence both the stability of the substrate and the free surface during this thin film flow. This two stage approach allows the roles of geometry, rotation and surface curvature, modified as the ice accretes, to be clearly separated and analysed.

Florian Stoll (Darmstadt) | Repeated Non-Coalescence of Droplets in Electric Fields

When two droplets come into contact, typically surface tension drives the coalescence into a single larger drop. The application of a sufficiently strong electric field can suppress this coalescence by deforming the droplets into conical shapes, causing them to recoil upon contact rather than merge. While previous studies have examined this phenomenon for isolated, single-contact events, sustained repetition of this recoiling behavior has not been established. Here we demonstrate that replacing pure water with a water-glycerol mixture enables hundreds of consecutive recoil events. In pure water droplets, the interface appears to destabilize following the initial contact, giving rise to capillary waves that disrupt subsequent recoil events. In water-glycerol mixtures, the elevated viscosity dampens these capillary waves and dissipates the interfacial energy sufficiently between contacts, allowing the process to repeat. These findings indicate that droplet viscosity plays a critical role in stabilizing the recoiling behavior and suggest that tuning the droplet viscosity is a viable strategy for achieving sustained, controlled droplet non-coalescence.

Henry Thomas Sharp (Strathclyde) | The effect of gravity on a pinned sessile droplet undergoing one-sided evaporation

The evaporation of a sessile droplet is a widely studied phenomenon due to its occurrence in a variety of industrial applications, including inkjet printing. In many of these applications, droplets contain suspended particles and the morphology of the deposit of particles left behind on the substrate is of considerable interest. In the case of inkjet printing for example, a uniform deposition of particles is typically required to produce a high-quality image [1]. When the droplet is sufficiently large that its characteristic length scale is greater than the capillary length, the effect of gravity becomes significant and the free surface profile becomes pancake-shaped. This occurs, for example, when the volume of a water droplet is much greater than 1 μL [2]. When an evaporating droplet is surrounded entirely by vapour, or when the vapour moves rapidly away from the surface of the droplet, the liquid and gas phases decouple, and the rate of evaporation is entirely dependent on the liquid phase [3, 4]. In this case, the associated thermodynamic disequilibrium at the free surface drives evaporation and is described by the so-called "one-sided" model of evaporation [3, 4]. This situation is in contrast with the widely-studied case of diffusion-limited evaporation in which evaporation is driven by the transport of vapour in the gas phase [5]. A mathematical model is formulated and analysed that predicts the evolution of, and the flow within a thin, pinned, sessile droplet undergoing one-sided evaporation with gravity-driven shape change. It is shown that when the effect of gravity is strong compared to that of capillarity, the behaviour of the droplet is different in an outer region away from the contact line and inner region close to the contact line. In the outer region, the free surface is approximately flat and the local evaporative mass flux is approximately spatially uniform. In the inner region, the dominant contribution to the local evaporative mass flux comes from the contribution near the contact line, where the local evaporative mass flux is constant. Moreover, in the limit of large initial volume, the total evaporative mass flux tends to infinity. Despite this, in the limit of large initial volume, the lifetime of the droplet tends to the lifetime in the corresponding case of a thin-film undergoing one-sided evaporation [3].

[1] Kuang, M., Wang, L., and Song, Y., "Controllable printing droplets for high-resolution patterns," Advanced Materials, Vol. 26, No. 40, 2014, pp. 6950–6958.

[2] Larson, R. G., "Transport and deposition patterns in drying sessile droplets," AIChE Journal , Vol. 60, No. 5, 2014, pp. 1538–1571.

[3] Burelbach, J. P., Bankoff, S. G., and Davis, S. H., "Nonlinear stability of evaporating/condensing liquid films," Journal of Fluid Mechanics, Vol. 195, 1988, pp. 463–494.

[4] Murisic, N. and Kondic, L., "On evaporation of sessile drops with moving

contact lines," Journal of Fluid Mechanics, Vol. 679, 2011, pp. 219–246.
[5] Wilson, S. K. and D'Ambrosio, H.-M., "Evaporation of sessile droplets," Annual Review of Fluid Mechanics, Vol. 55, 2023, pp. 481–509.

Jingce Tao (Nottingham) | Surfactant-laden film flow over an inclined corrugated wall

This study investigates the coupled effects of capillarity and Marangoni stresses in a gravity-driven liquid film flowing down an inclined corrugated wall in the presence of insoluble surfactant. The flow is considered in the zero Reynolds number limit. Using an asymptotic long-wave framework, a coupled system consisting of a film thickness evolution equation and a surfactant advection-diffusion equation is derived. The resulting model is solved numerically to obtain steady states. The results demonstrate that surfactant-induced Marangoni stresses reduce the flow velocity while preserving the capillary ridge structure. Furthermore, the topography-induced deformation of the film leads to a non-uniform distribution of surfactant concentration along the free surface.

Microfluidics and applications

Jason Boateng (Leicester) | Computational Study of Spontaneous Coronary Artery Dissection

Spontaneous coronary artery dissection is a hard-to-predict and potentially fatal disease that occurs in the human cardiovascular system. Key aspects of the condition, such as the mechanism underlying the formation and propagation of the "false lumen" hematoma in blood vessels, remain unsatisfactorily understood, due to limitations in medical imaging. In order to gain understanding of this phenomenon, a series of computational fluid dynamics (CFD) and fluid structure interaction (FSI) simulations of a dissected blood vessel are conducted using OpenFOAM and related libraries. Hypotheses concerning the development of the condition are investigated, such as the "outside-in mechanism" conjecture.

Lewis Melvin (Manchester) | Microfluidic Fuses: Flow-Induced Choking in Soft Cylindrical Channels

Fluid-structure interactions are commonly used in the development of microfluidic analogues of microelectronic devices. Examples that exist already in the literature include microfluidic diodes, transistors, and capacitors. In this talk, we theoretically and experimentally explore a microfluidic fuse, a device which prevents fluid flow above a determined critical flow rate. These devices consist of a cylindrical channel formed inside a cylindrical elastomer, confined on all sides by a rigid mold. The softness of the elastomer permits shape-morphing of the channel in response to fluid forcing. Beyond the critical flow rate, flow-induced deformation causes the channel diameter to increase nearer the flow inlet, displacing elastomer towards the flow outlet, where the channel constricts. If the magnitude of this deformation is sufficient, the channel will constrict to the point that fluid flow is inhibited (the flow is "choked"). For this proof-of-concept device, we demonstrate the robustness of this choking mechanism, explore the tunability of the critical choking flux, and compare with theoretical predictions.

Claire Denham (Edinburgh) | Behaviour of heterogeneous blood suspensions as a disease model in microchannel constrictions

Blood disorders such as iron deficiency anaemia (IDA) and spherocytosis affect millions globally. These blood disorders have a small population of stiffer and smaller red blood cells (RBCs) which may behave differently to healthy RBCs. Blood vessels may constrict due to healthy physiology, for controlling bleeding or blood pressure, and pathology, such as plaque build-up creating a stenosis. The behaviour of blood disorder suspensions in straight and constricted microchannels remains unclear. We hypothesise that the presence of these stiffer cells changes the spatial distribution, in turn changing the wall shear stress (WSS). Using immersed-boundary-lattice-Boltzmann simulations, we explore the effect of the IDA and spherocytes on the haemodynamics of a straight channel and constricted microchannels. We investigate the change in distribution of RBCs and WSS as a result of the stiffer cells and the constrictions. As the degree of constriction increases, the lateral and angular migration of the RBCs increases leading to RBC mixing. The cell free layer decreases immediately upstream of the constriction leading to an increase in WSS fluctuations in that region. Our simulations show that the presence of stiffer cells create individual WSS footprints, changing the distribution of WSS. However, as the degree of constriction increases these footprints are less present in the WSS pattern. The stiffer RBCs and constricted microchannels lead to significant changes in the WSS. The behaviour of blood disorders has been determined for different microvessel constrictions which may be of note for better understanding these conditions and the following endothelial cells response.

Ebughni Okoria Nangi (Cullum) | Fluid Structure Interaction Analysis of an Irregular Subsea Jumper Pipe under Gas Liquid viscous Slug Flow Loading: Dynamic Response, Stress and Fatigue

The co-flow of gas and liquid in pipes may be unstable at certain flow conditions due to the fluctuations of void fraction, phase density, pressure and momentum flux. These flow fluctuations are more prominent under intermittent flow regime. Irregular subsea jumper pipes operating in deepwater oil and gas fields operating under this intermittent flow regime may experience sustained gas liquid slug flow that imposes impulsive, fluctuating loading at each bend. Despite the critical importance of fatigue integrity at these locations where remotely operated vehicle inspection intervals of three to five years leave significant crack propagation undetected existing structural analyses of subsea jumpers have relied exclusively on static equivalent representations of the internal flow loading: single-amplitude pressure values, spectral density functions applied independently to each structural mode, or peak momentum change forces at bends computed from time-averaged CFD results. None of these approaches preserve the transient character of individual slug events, the spatial correlation of pressure loads across multiple members, or the three-phase flow evolution from gas intrusion through slug development to quasi-periodic slug flow. This paper presents direct coupling, time-domain fluid structure interaction (FSI) framework for an irregular five-bend subsea jumper in which the complete CFD derived force and pressure time history is applied to the structural model at every timestep without approximation or modal truncation. The two-phase flow across the jumper is computed using SST K-ω, with VOF model used for the interface tracking. 56 monitor data sampled at 2000Hz over 35s is saved for every time step during the CFD run, for direct application to the structural model. The structural model represents a five-bend irregular M-shaped jumper made of API 5L X65 steel, with an internal diameter of 67mm, external diameter of 73mm and a wall thickness of 73mm and elbow radius of R=3D-201mm, bending stiffness EI = $8.097 \times 10^4 N \cdot m^2$, The geometry comprises three vertical risers (3.0 m, 4.0 m, 4.0 m) and three horizontal spans (5.0 m, 7.0 m, 2.5 m) with a total pipe arc length of 25.50 m, defined in the X-Y plane. A Euler-Bernoulli beam finite element model is discretized into 330 elements at 30 elements per segment, yielding 331 nodes, 662 degrees of freedom, and 656 free degrees of freedom. Mixed boundary conditions apply fully fixed constraints (zero displacement and rotation) at both terminal flanges and simply supported constraints (zero transverse displacement, rotation free) at the midpoints of Bends 3 and 4 at arclengths of 12.79 m and 18.11 m from the inlet respectively, physically representing seabed mudmat supports at those elbow locations. The 30-element discretization reduces the minimum element length at bend cross-sections to 10.5 mm (0.14D), providing adequate spatial resolution for accurate curvature and bending stress recovery across the full 24.58 m pipe arc. Transient structural dynamics are solved using the Newmark-β constant-average-acceleration scheme (β = 1/4, γ = 1/2), which is unconditionally stable and second-order accurate. The method is applied directly to all 656 free degrees of freedom without modal truncation. The effective stiffness matrix is factorised once by LU decomposition, and 35000 triangular back-substitutions recover displacement, velocity, and acceleration histories at every node and timestep. Rayleigh proportional damping (α_R = 0.9440 S^{-1},β_R = 2.234×10^{-4}s) is calibrated to ζ = 2% damping at modes 1 and 3, representing the frequency dependent damping of structural-steel subsea piping. Sixteen natural frequencies and mass-normalised mode shapes are extracted by generalised eigenvalue method, and the effective modal mass fractions confirm that the 90% cumulative mass-participation criterion is satisfied within this mode set. Five structural response fields are derived at every node and time step.

Zexuan Chen (Dublin) | Experimental and Computational Fluid Dynamics Study of Gas–Liquid Mass Transfer in Hydrogen-to-Biomethane Processes

Green hydrogen and biomethane are expected to play key roles in achieving net-zero carbon. However, hydrogen faces several challenges in storage and transport. Therefore, hydrogen is converted into methane, a gas fully compatible with existing infrastructure. Hydrogen-to-biomethane processes employ microorganisms to convert H_2 and CO_2 into CH_4. This study aims to overcome the low gas–liquid mass transfer bottlenecks in this process, through experiments and Computational Fluid Dynamics (CFD) simulations. The solubility of H_2 in water is low compared to that of methane and carbon dioxide, and this limits the process efficiency. In this study, the hydrogen mass transfer coefficient will be determined in a laboratory-scale reactor using a conventional sparger–mixer system and compared against other gas delivery options: slug flow formation, static mixers, and bubbleless gas transfer. Dissolved hydrogen concentrations will be directly measured by an inline H2 probe, delivering high frequency data allowing accurate calculation of gas mass transfer rates. The design and operational parameters of various gas delivery methods will be assessed with the aid of Computational Fluid Dynamics modelling. For example, for the conventional sparger–mixer which equipped with blade impellers is simulated using Ansys Fluent. The optimization strategies involve varying the impeller positions, rotational speeds, and gas inlet flow rates to evaluate the volumetric mass transfer coefficient (kla). For the static mixer, the optimization strategies involve varying the inlet velocity, inlet gas ratio, and number of mixing elements. kla consists of two components: the mass transfer coefficient kl, which is a function of the turbulent dissipation rate which can be obtained from Fluent. The gas–liquid interfacial area a depends on the gas volume fraction and the bubble diameter which can also be obtained from Fluent. Therefore, by combining CFD simulation results with physical experiments, ways to improve the efficiency of gas delivery can be identified.