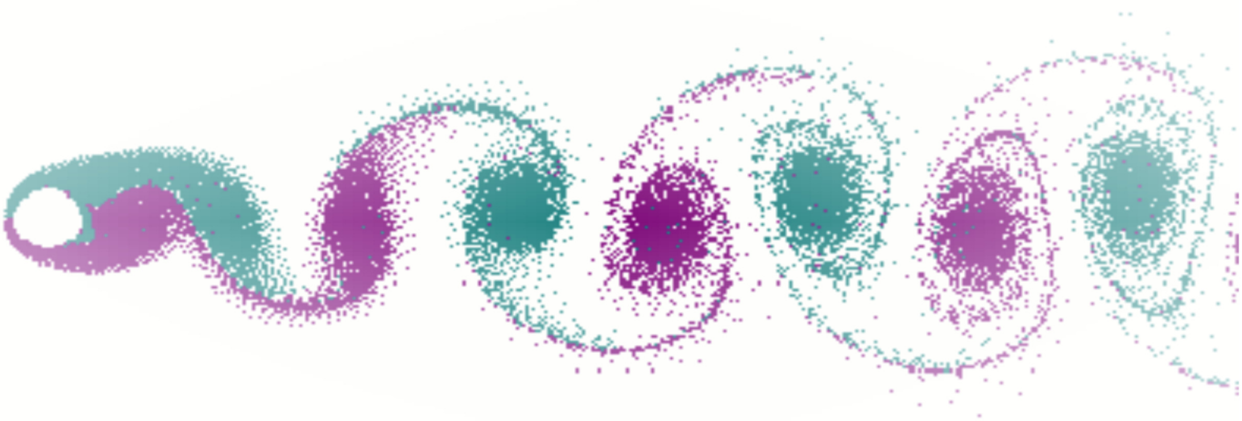
Imperial College London

ic fluids EPSRC Centre for Doctoral Training

CDT Fluids Student Symposium

Skempton Building 201, South Kensington Campus

Wednesday July 11th, 2018



Welcome to Imperial Fluids CDT

Thank you for joining us for the third annual CDT Fluids Student Symposium.

The aim of the symposium is to encourage students from different disciplines to share their work by providing an engaging one day conference to discover how diverse the field can be. As the Fluids CDT encourages collaboration between many departments at Imperial, we endeavour to showcase a wide range of applications of fluid dynamics with student presentations, poster sessions and guest speakers.

A special thanks goes to Matthew Piggott and Clodagh Li who helped us in the organisation of the day with insight and helpful suggestions.

We wish you a very productive and enjoyable day.

Sincerely,
CDT Fluids Symposium Organising Committee

Lecture Theatre 201, Skempton Building

08:30 - 09:00	Registration and Coffee
09:00 - 10:00	**Keynote I - Steve Tobias** **Order from Chaos: Direct Statistical Simulation in (Astrophysical, Geophysical and Engineering) Fluids**
	Session A - Magnetic and convective hydrodynamics
10:00 - 10:20	Paul Mannix *Weakly non-linear mode interactions in spherical Rayleigh-Bénard convection*
10:20 - 10:40	Francis Nwobu *Shock-induced vorticity production by entropy modes in magnetohydrodynamics*
10:40 - 11:00	Raquel Henriques Vaz *Unforced Navier-Stokes solutions with applications to convection and magnetohydrodynamics*
11:00 - 11:30	Morning Coffee Break
	Session B - Multiphase flows and stability theory
11:30 - 11:50	Rishabh Gvalani *Continuous and discontinuous transitions in a toy model for granular media*
11:50 - 12:10	Anxiong Liu *Population balance modelling and laser diagnostic validation of soot particle evolution in laminar ethylene diffusion flames*
12:10 - 12:30	Rishi Kumar *Linear stability of Poiseuille-Couette flow*
12:30 - 13:30	Lunch & Poster session
	Session C - Biological and environmental fluid dynamics
13:30 - 13:50	Simon Schoeller *Linking individual and collective dynamics of sperm in suspension*
13:50 - 14:10	Laura Sumner *Steady streaming as a method for drug delivery to the inner ear*
14:10 - 14:30	Sean Jamshidi *The role of potential vorticity in coastal outflows*
14:30 - 14:50	Afternoon Coffee Break
	Session D - Turbulent Flows
14:50 - 15:10	Owen Jordan *Direct simulation of turbulent plumes in a crossflow*
15:10 - 15:30	Adrian Leung *Finite of infinite predictability horizon?*
15:30 - 15:50	Vasilis Ioannou *Entrainment and mixing properties of plasma controlled turbulent jets using implicit large-eddy simulations*
16:00 - 17:00	**Keynote II - Martin Bees** **Modelling Suspensions of Cells: Swimming Behaviour to Bioconvection**
17:00 - 17:30	Closing Remarks and Prizes
17:30 - 18:30	Drinks Reception

Keynote Speakers

Steve Tobias

University of Leeds, Department of Applied Mathematics

Order from Chaos: Direct Statistical Simulation in (Astrophysical, Geophysical and Engineering) Fluids.

The eleven year solar activity cycle is a remarkable example of regular behaviour emerging from an extremely turbulent system. The jets on Jupiter sit unmoving on a sea of turbulent eddies. Astrophysical phenomena often display organisation on spatial and temporal scales much larger than the turbulent processes that drive them. An outstanding problem of astrophysics (and indeed other branches of nonlinear physics) is how to describe the statistics of systematic behaviour emerging from the underlying chaos, given that Direct Numerical Simulation of these objects is simply impossible. Here I shall introduce these fascinating phenomena and describe how methods from non-equilibrium statistical mechanics and many body quantum systems may be developed to give some insight into their behaviour. I shall also speculate on possibilities of using this approach in engineering fluids.

Martin Bees

University of York, Department of Mathematics

Modelling Suspensions of Cells: Swimming Behaviour to Bioconvection

Suspensions of swimming microorganisms behave differently to standard fluids, with the cells providing an internal source of energy that can drive flows. The microorganisms, such as algae and bacteria, typically attempt to swim in directions that improve their environment (for example, against gravity, biased by shear, or towards nutrients or light). This biased swimming behaviour together with differences in physical properties can initiate large scale flows, sometimes resulting in cells accumulating in counter-intuitive regions. For instance, up-swimming cells often find themselves at the bottom and in a flow disperse differently to nutrients. Here, I shall describe experiments that aim to disentangle some of the mechanisms and explain how to scale up from the hydrodynamics of individuals to models of dilute suspensions. I shall introduce applications from planktonic segregation, dispersion of swimming cells in shear and the non-intuitive transport of swimming cells in oscillatory flows due to helical resonant alignment.

List of Talks

Session A: Magnetic and convective hydrodynamics

Weakly Nonlinear Mode Interactions in Spherical Rayleigh-Bénard Convection

Paul Mannix
Imperial College London, Department of Mathematics

In an annular spherical domain with separation d, the onset of convective motion in the form of roll (mode) patterns occurs at a critical Rayleigh number Ra. Solving the linear stability problem, it is shown that degenerate points (Ra_c, d_c) exist where two sets of rolls simultaneously become unstable. Considering the weakly nonlinear behaviour in the neighbourhood of such points the preferred solution is established. It is demonstrated that four distinct coupled amplitude equations govern the nonlinear evolution of these interactions, the choice of which can be predicted from the inherent symmetry of the interacting modes. Direct numerical simulation is used to validate and contextualise all results.

Shock-Induced Vorticity Production by Entropy Modes in MHD

F. Nwobu, N. Alferez, E.Touber
Imperial College London, Department of Mechanical Engineering

Shock-wave turbulence interactions (SWTIs) in ionised gases permeated by a mean magnetic field have recently seen growing interest [1]. In particular, the influence of the mean magnetic field's strength and orientation on energy redistribution across the shock [2]. In hydrodynamics, turbulent kinetic energy (TKE) is amplified and its length scale attenuated across a shock, effects which dictate post-shock properties such as drag and heat transfer and would also affect turbulent magnetic field growth in MHD. Understanding the interaction in MHD carries importance within astrophysics where SWTIs are thought to affect magnetic field evolution in supernova remanants [3] and hypersonic vehicle design in which a magnetic field can be applied to control an incoming ionised air flow, improving re-entry and off-design performance [4]. For weak turbulent fluctuations, linear theory has been shown to be good predictor of TKE redistribution in hydrodynamic SWTIs. However, its MHD counterpart has seen relatively little study, notably, linear/nonlinear contributions to the production of vorticity (a key indicator of TKE) by upstream density fluctuations interacting with a shock and how the mean magnetic field affects the isotropy of this interaction. In Linear Interaction Analysis (LIA) weak hydrodynamic turbulent fluctuations can be decomposed into a (Fourier) series of plane waves comprised of vortical, acoustic and entropy modes. A shock redistributes energy from an incident wave to all other eigen-modes. The refraction properties for each respective incident wave can be computed individually and superimposed to reconstruct the downstream flow field. A similar wave decomposition is possible in MHD for which the interaction can be decoupled into in-plane magneto-acoustic/entropy and out-of-plane Alfven mode excitations respectively. We use LIA to validate Direct Numerical Simulations (DNS) of a 2D isolated Gaussian-distributed entropy spot interacting with a normal shock. Like turbulence, a Gaussian-distributed perturbation excites a broad range of Fourier modes. Results are presented for the interaction in the presence of a magnetic field aligned parallel and obliquely to the shock normal direction. We show that vorticity is redistributed in the post-shock flow by all magneto-acoustic modes - in hydrodynamics vorticity transport occurs only via the vortical mode, convected downstream at the flow speed. Vorticity is most intense in the slow magneto-acoustic modes for both magnetic field orientations. The structure and propagation of downstream modes remains symmetric about the shock normal for the parallel MHD shock interaction. No clear lines of symmetry are observed in the oblique MHD shock interaction.

[1] Giacalone, J., & Jokipii, J. R. (2007). Magnetic field amplification by shocks in turbulent fluids. The Astrophysical Journal Letters, 663(1), L41.

[2] Ji et al., Mon Not R Astron Soc Lett 463,4 (2016).

[3] Xu, S., & Lazarian, A. (2017). Magnetic field amplification in supernova remnants. The Astrophysical Journal, 850(2), 126.

[4] Shneider, M. N., Macheret, S. O., & Miles, R. B. (2004). Analysis of magnetohydrodynamic control of scramjet inlets. AIAA journal, 42(11), 2303-2310.

Unforced Navier-Stokes solutions with applications to convection and magnetohydrodynamics

Raquel Henriques Vaz
Imperial College London, Department of Mathematics

Steady Boussinesq flow in a weakly curved channel driven by a horizontal temperature gradient is considered. A similarity structure with linear variation in the transverse direction is assumed so that the problem reduces to a system of ordinary differential equations. A series expansion in G, the product of the Grashof number and the square root of the curvature, reveals a real singularity and anticipates hysteresis. Numerical solutions are found using path-continuation and the bifurcation diagrams for different parameter values are obtained. Multivalued solutions are observed as G and the Prandtl number vary. Often the solutions are insensitive to the local boundary conditions and can be regarded as perturbations of the homogeneous problem. Four such unforced solutions are found. In two of these the velocity remains coupled with temperature which, formally, scales as $1/G$ as $G \to 0$. The other two are purely hydrodynamic. The existence of such solutions is due to the unbounded nature of the domain. It is shown that these occur not only for the Dean equations, but constitute previously unreported solutions of the full Navier–Stokes equations in an annulus of arbitrary curvature. Dynamo action is investigated in these circumstances. Magnetic field instabilities with the same spatial structure are sought. The kinematic eigenvalue problem is found to have two growing modes for moderate values of the magnetic Reynolds number, R_m. As $R_m \to \infty$ it is shown that the modes are governed by layers on the outer wall. As the field grows, saturated solutions to the nonlinear dynamo problem are found in three distinct solution families. The complicated bifurcation structure is investigated.

Session B: Multiphase flows and stability theory

Continuous and discontinuous transitions in a toy model for granular media

Rishabh Gvalani
Imperial College London, Department of Mathematics

We study a particle model for granular media[1] in the infinite particle limit. The limit is described by a partial differential equation referred to as the McKeanVlasov equation. The system admits as its trivial stationary solution a spatially homogenous distribution. We show that, given an appropriate choice of interaction, potentially infinitely many nontrivial solutions may exist. Furthermore, we analyse the nature of the transitions from the trivial to the nontrivial solutions. This is joint work with Jos Carrillo, Greg Pavliotis, and Andr Schlichting.

[1] Benedetto, D., Caglioti, E., & Pulvirenti, M. (1997). A kinetic equation for granular media. ESAIM: Mathematical Modelling and Numerical Analysis, 31(5), 615-641.

Population balance modelling and laser diagnostic validation of soot particle evolution in laminar ethylene diffusion flames

Anxiong Liu, Carlos Garcia, Fabian Sewerin, Benjamin Williams, Stelios Rigopoulos
Imperial College London, Braunschweig University of Technology and University of Oxford

Soot is composed of polydisperse particles that are generated during hydrocarbon combustion processes. Besides being an air pollutant, soot also affects combustion efficiency and safety as it can accumulate inside combustors or their components. On the other hand, soot can enhance heat transfer and is thus sought after in equipment such as furnaces, provided that it can be oxidised before exit. Soot formation involves several gas-particle interaction processes: gas-phase chemical kinetics govern the formation rates of precursor species, particle inception rates determine the formation of nuclei from precursors and surface growth and coalescence control the growth of primary particles. In a post-growth stage, moreover, agglomeration and oxidation entail the formation of fractal-shaped aggregates and, possibly, a reduction of the particle volume. In order to understand the mechanisms involved in the evolution of soot formation in practical combustion problems, it is essential to establish predictive models and the experimental validation for simpler configuration such as a laminar jet coflow flame [1]. Population balance equation (PBE) allows for predicting the temporal and spatial evolution of the distribution of particle size or any other properties characterising the particulate population. In this research, a finite-volume PBE approach, discretizing along the property of particle size (volume), is incorporated in a time-dependent in-house CFD code (BOFFIN) [2], with a coverage of the conservation laws of mass, transport equations of momentum, energy, gas phase species and particle number densities at different volumes. Here, several kinetic submodels for the different processes of soot formation: nucleation, surface growth [3], spherical coagulation [4,5] with fractal adjustment and oxidation, are included. Moreover, an estimated primary particle size by the thermophoretic sampling has been applied to the PBE-CFD code in order to describe the fractal aggregates with an accurate morphological model. Accompanying, a set of experiments on a laminar co-flow burner is conducted and various laser diagnostic techniques are used to characterize the flame. OH-PLIF is used to validate the chemical kinetics and mixing, elastic light scattering (ELS) provides information about soot formation and line-of-sight extinction measurements allow us to determine the soot volume fraction. Location-specific samples are extracted with the aid of a thermophoretic sampling device similar to that originally designed by Dobbins [6,7]. Model validation is performed by comparing predicted signals' obtained from the PBE-CFD model with experimental laser diagnostic signals. This approach circumvents the complexities that are commonly met in converting experimental laser diagnostic signals into physically meaningful quantities. The objectives of our work are twofold: first, we present planar OH-LIF and ELS measurements characterizing the flame structure, precursor species and soot appearance. As well, the light extinction measurements are directly correlated with the integrated soot volume fraction and the thermophoretic sampling helps estimate the average primary particle size, a crucial morphological parameter for modelling fractal aggregates. Second, we combine a detailed PBE model for soot with an in-house CFD code, in order to predict the evolution of the soot PSD throughout the flame. Based on predictions of the chemical composition and the soot PSD, we obtain OH-PLIF, ELS and light extinction signals which form the basis for a direct comparison with the experimental signals.

[1] Santoro, R. J., Semerjian, H. G., & Dobbins, R. A. (1983). Soot particle measurements in diffusion flames. Combustion and Flame, 51, 203-218.

[2] Jones, W. P., di Mare, F., & Marquis, A. J. (2002). LES-BOFFIN: users guide. Mechanical Engineering Department, Imperial College London, London.

[3] Appel, J., Bockhorn, H., & Frenklach, M. (2000). Kinetic modeling of soot formation with detailed chemistry and physics: laminar premixed flames of C2 hydrocarbons. Combustion and flame, 121(1-2), 122-136.

[4] Kronenburg, A., Bilger, R. W., & Kent, J. H. (2000). Modeling soot formation in turbulent methaneair jet diffusion flames. Combustion and Flame, 121(1-2), 24-40.

[5] Vemury, S., & Pratsinis, S. E. (1995). Self-preserving size distributions of agglomerates. Journal of Aerosol Science, 26(2), 175-185.

[6] Dobbins, R. A., & Megaridis, C. M. (1987). Morphology of flame-generated soot as determined by thermophoretic sampling. Langmuir, 3(2), 254-259.

[7] Dobbins, R. A., & Megaridis, C. M. (1987). Morphology of flame-generated soot as determined by thermophoretic sampling. Langmuir, 3(2), 254-259.

Linear stability of Poiseuille-Couette flow

Rishi Kumar

Imperial College London, Department of Mathematics

Plane Poiseuille-Couette flow is produced in a channel by the application of a uniform pressure gradient and relative motion of the walls. This type of flow is the linear combination of Poiseuille flow (in which flow is driven by an applied pressure gradient parallel to the walls of the channel) with plane Couette flow (in which flow is produced in a channel when walls slide with same speed (say with) but in opposite directions). The aim of the talk is to investigate the linear stability of plane Poiseuille-Couette flow both at finite Reynolds numbers and in the large Reynolds number limit. To examine the linear stability at finite Reynolds number the Orr-Sommerfeld equation which describes the linear three-dimensional modes of disturbance to this flow is solved numerically using a standard Chebyshev-collocation method. Although there is a unique neutral curve when the sliding speed is zero, it is found that for non-zero sliding speed, multiple neutral curves exist in the two-dimensional problem () for Poiseuille-Couette flow at large Reynolds number consistent with the predictions of [1] exclusively based on a high-Reynolds-number asymptotic approach to this flow. These curves retreat to infinity provided the non-dimensional wall speed (corresponding to the dimensional speed where is the maximum speed of the Poiseuille component of the flow) is approximately 0.34 in line with Reynolds & Potter's (1967) conclusions. We will compare the large asymptotes of these various neutral curves with asymptotic theory. The asymptotic analysis of the lower branch of the main neutral curve for and where will be discussed thoroughly. Finally, we consider a qualitative comparison of finite Reynolds number computations of the two-dimensional problem and the corresponding solutions to the lower branch eigenrelation for a specific sliding velocity.

[1] Cowley, S. J., & Smith, F. T. (1985). On the stability of Poiseuille-Couette flow: a bifurcation from infinity. Journal of Fluid Mechanics, 156, 83-100.

[2] Reynolds, W. C., & Potter, M. C. (1967). Finite-amplitude instability of parallel shear flows. Journal of Fluid Mechanics, 27(3), 465-492.

Session C: Biological and environmental fluid dynamics

Linking individual and collective dynamics of sperm in suspension

Simon Schoeller and Eric Keaveny

Imperial College London, Department of Mathematics

Suspensions of sperm cells can display a diverse array of collective behaviour ranging from turbulence-like dynamics [1] to coherent motion of tightly packed groups. Collective dynamics of swimming suspensions are often investigated using models that treat cells as point particles or rigid objects that exert steady dipolar forces on the surrounding fluid. While these models can reproduce the turbulence-like state, through their waving flagella, sperm exert time-dependent, beyond-dipolar forces on the fluid. When these details are included, interactions through the fluid lead to attraction and flagellar synchronisation of neighbouring cells.

Using simulations [2] of up to 1000 cells immersed in a Stokes fluid, we investigate how time-varying features at the scale of the individual affect the collective dynamics on a larger scale. Resolving the fluid flow, steric interactions and elasticity of the flagella, we test how hydrodynamic interactions and the variability of the swimmers' undulation frequency affect collective dynamics. If the frequencies of all swimmers are nearly the same, we find that swimmers will form growing clusters. If there is sufficient variation in the undulation frequencies, however, the suspension develops vortices and swirls much larger than the size of the individual cells. We also find that the onset of the swirling state depends strongly on the cell density. Further, a quantitative analysis of the swirling collective dynamics reveals that the flows generated by flagellum undulations contribute substantially to the overall energy in the fluid.

[1] Creppy, A., Praud, O., Druart, X., Kohnke, P. L., & Plourabou, F. (2015). Turbulence of swarming sperm. Physical Review E, 92(3), 032722.

[2] Schoeller, S. F., & Keaveny, E. E. (2018). From flagellar undulations to collective motion: predicting the dynamics of sperm suspensions. Journal of The Royal Society Interface, 15(140), 20170834.

Steady Streaming as a Method for Drug Delivery to the Inner Ear

Laura Sumner and Tobias Reichenbach

Imperial College London, Department of Bioengineering

Background: The human ear converts pressure waves into electrical signals which are then relayed to the brain. This mechanotransduction occurs in the Organ of Corti situated on the basilar membrane inside the spiral-shaped temporal bone. The motion of the basilar membrane due to sound causes hair cells inside the Organ of Corti to open ion channel gates and trigger action potentials in attached auditory-nerve fibers. The hair cells can be damaged due to noise exposure or age, however. Mammalian hair cells cannot be regenerated, leading to sensorineural hearing loss. Sensorineural hearing loss can potentially be prevented or treated through drugs, but delivering the drugs to the hair cells remains one of the important problems. The inner ear is encased in the body's hardest bone, and drugs can only be injected through the round window or oval window at its base (Salt and Plontke 2009). Here we seek to investigate if steady streaming may be employed to distribute drugs from the base of the cochlea across its longitudinal extent. Steady streaming is indeed present in fluid systems with a fluctuating flow and results in a non-zero mean flow (Riley 2001). In the cochlea, the fluctuating flow results from the motion of the basilar membrane. This membrane has a spatially-varying impedance which allows to spatially segregate frequencies (Reichenbach and Hudspeth 2014). At a particular frequency the basilar membrane re-

sponds maximally at a frequency-specific location. Distributing drugs through steady streaming may thus employ a series of sounds of different frequencies to transport drugs from the high- to the low-frequency region.
Method: We model the basilar-membrane motion at a particular frequency through the WKB approximation. We then use this motion as input to a computational fluid-dynamics (CFD) simulation that we implement using OpenFoam. The simulation uses a dynamic mesh solver combined with a particle tracking solver to enable the coordinates of individual particles across the entire domain to be determined and post processed to find the steady streaming velocities.
Results: Results for the longitudinal streaming at 8kHz show a reasonable agreement with the theoretically proposed values. These values are also large enough such that therapies which utilise this effect are feasible (a time period of a few hours would be required). The boundary layer thickness is also an almost exact match to theory, meaning validation of the simulation.
Conclusions: We show that the simulation predicts considerable steady streaming over timescales reasonable for therapies, and plan to use it to investigate different types of sound stimulation for efficient drug transport.

The role of potential vorticity in coastal outflows

Sean Jamshidi
Imperial College London, Department of Mathematics

The long-wave, reduced-gravity, shallow-water equations (the semi-geostrophic equations) are used to study the outflow of a river into the ocean. While previous models have studied dynamics driven by gradients in density, the focus here is on the effects of potential vorticity. The river water has the same density as a finite-depth upper layer of oceanic fluid, but the two fluids have different, uniform, potential vorticities. The governing equations can be reduced to two first-order, nonlinear hyperbolic PDEs which are then integrated numerically for a prescribed outflow velocity and oceanic-layer depth. Results are found to depend strongly on the sign of the potential-vorticity anomaly (PVa), with all fluid turning downstream (in the direction of Kelvin wave propagation) when the river water has positive PVa and anticyclonic flow upstream of the river mouth when the PVa is negative. In all cases, a nonlinear Kelvin wave propagates at finite speed ahead of the river water. A range of behaviours is observed, including flows that develop shocks and flows that continue to expand offshore. The qualitative behaviour of the outflow is strongly correlated with the value of a single dimensionless parameter that expresses the ratio of the speed of the flow driven by the Kelvin wave to that driven by image vorticity.

Session D: Turbulent Flows

Direct simulation of turbulent plumes in a crossflow

Owen Jordan and Maarten van Reeuwijk
Imperial College London, Department of Civil & Environmental Engineering

Direct Numerical Simulation of buoyant plumes in a uniform crossflow is performed with the aim of investigating the effect of the crossflow velocity on the dynamics of the plume evolution. A detailed comparison is made with the classical theory of bent-over plumes. The simulations indicate that the forcing associated with pressure differences across the cross-section of the plume is a leading order quantity, which is important since this term is generally discarded in the plume theory. We explore how this quantity can be parameterized with the help of DNS data. Furthermore, we investigate how the crosswind affects the evolution of the plume geometry.

Finite or infinite predictability horizon?

Adrian Leung
University of Reading, Mathematics of Planet Earth CDT

It is well-accepted that the chaotic nature of atmospheric dynamics imposes an inherent finite limit of predictability. The idea originated from a paper by Edward Lorenz in 1969, in which he presented a theoretical argument suggesting that the predictability horizon cannot be indefinitely extended by reducing the initial error to any-thing above zero. In his derivation the 2D barotropic vorticity model was used, and the error growth behaviours with kinetic energy spectral slopes of -5/3 and -7/3 were compared. Extrapolating from his results, Lorenz hypothesised that the predictability would become infinite if the spectral slope steepened to -3 without providing any direct numerical or theoretical evidence. We have performed direct numerical simulation of a forced-dissipative version of the spectral 2D barotropic vorticity model with a -3 spectral slope. Contrary to Lorenzs hypothesis, our results suggest finite predictability. Repeating Lorenzs derivation for the case of a -3 spectral slope, we discuss the possible reasons leading to the disagreement of our results with his hypothesis.

Entrainment and Mixing properties of plasma-controlled turbulent jets using implicit Large-Eddy Simulations

Vasilis Ioannou
Imperial College London

In this work, implicit Large Eddy Simulations of turbulent jets are performed to investigate the effect of Dielectric Barrier Discharge (DBD) plasma actuators located inside the jets nozzle. To achieve this, the high-order flow solver Incompact3d is used. This finite-difference solver is based on sixth-order schemes on a Cartesian mesh for accuracy, a 2D domain decomposition library for scalability, a spectral Poisson solver for efficiency and an Immersed Boundary Method to model a solid geometry inside the computational domain. A convergent nozzle is included inside the computational domain in order to obtain a realistic flow field inside the nozzle. A good agreement with experimental data is obtained for the non-controlled case and a Reynolds number equal to 460,000. In this presentation, various arrangements of an array of 8 DBD plasma actuators radially distributed are used to trigger instabilities inside the nozzle. More specifically, an axisymmetric case and azimuthal modes 1 and 2 cases with the actuators pulsating at a Strouhal number of 0.32, close to the jet preferred frequency, are investigated and the results are compared with a non-controlled case. The first investigation deals with the mixing properties of the jet, with a study of the length of the potential core, jet spreading and the convection of a passive scalar field and associated probability density functions. An entrainment analysis is also performed, calculating the entrainment coefficient, and analyse the existence of coherent structures in the near field downstream of the nozzle exit.

List of Posters

1 **Reaction Mechanisms for Material Systems with Complex Potential Energy Surfaces**

ALIKI TSOPELAKOU, PETER LINDSTEDT
Imperial College London, Department of Mechanical Engineering

The chemical interactions between surfaces and adjoining fluid phases are often poorly understood and detailed chemistry modeling across scales, starting from the fluid-material surface boundary, is arguably essential for explaining the resulting events. Our aim is to shed light on the development of advanced computational tools that ultimately enable the quantification of interactions between chemical species resulting from advanced engine operation modes and selected metal surfaces (Pt, Rh). The rate parameters for the surface chemistry were derived using a systematic application of Variational Transition State Theory (VTST), for adsorption reactions coupled with two dimensional (2D) collision theory for reactions occurring on the surface. The VTST approach removes the need for the experimental determination of surface sticking coefficients and the associated major uncertainties. The barrier heights were determined using the Unity Bond Index-Quadratic Exponential Potential (UBI-QEP) method. Finally, a thorough analysis of the performance of the detailed chemical mechanisms developed by Kraus and Lindstedt [1,2] and Vincent et al. [3] has been conducted, under fuel-rich conditions over a platinum coated surface in presence and absence of CO co-feed. The reactions of CO over Pt are well-known to be controversial and the current conditions have not been previously studied.

[1] Kraus, P., & Lindstedt, R. P. (2016). Variational Transition State Theory-Based Surface Chemistry for the C2H6/H2/O2/Pt System. Energy & Fuels, 31(3), 2217-2227.

[2] Kraus, P., & Lindstedt, R. P. (2017). Microkinetic Mechanisms for Partial Oxidation of Methane over Platinum and Rhodium. The Journal of Physical Chemistry C, 121(17), 9442-9453.

[3] Vincent, R. S., Lindstedt, R. P., Malik, N. A., Reid, I. A. B., & Messenger, B. E. (2008). The chemistry of ethane dehydrogenation over a supported platinum catalyst. Journal of Catalysis, 260(1), 37-64.

2 **Investigating inertial particle transport for application in tracking plastic litter in the ocean**

BIRGIT STZL
Imperial College London, Department of Civil Engineering

Motivated by the study of floating plastic debris in the ocean, we investi-gated a model for inertial particle transport in the ocean. In most numerical studies so far, plastic objects are treated as passive particles with negligible size, and their movement purely follows the ocean flow. Inertial particles in contrast are viewed as objects with finite size and mass, which interact with ambient fluid and experience flow drag. We implemented a simplified version of the most commonly used approx-imation of inertial particle transport, the MaxeyRiley equation (Maxey and Riley 1983), into a Lagrangian particle tracking model, coupled with an idealised, but dynamically consistent and highly turbulent midlatitude double-gyre ocean circulation model. Simulations of the inertial particle transport model showed that inertial particle transport can significantly dif-fer from purely flow driven passive particle transport in the double-gyre ocean circulation framework, and only for inertial particles a significant particle accumulation occurs. Key parameters that influence the inertial particle movement are the particle density and size. Lighter particles, with lower density than the fluid, are attracted to the inte-rior of the ocean gyres, particularly to the regions around the eastward ex-tension of the western boundary currents, where the eddying flow is most energetic. Heavier particles, with higher density than the fluid, are attracted outwards of the gyres and towards the boundaries of the domain. Numerical simulations of plastic particles slightly lighter than water show a similar tendency to accumulation in the western, more turbulent regions of the inte-rior and interface of the ocean gyres.

This is consistent with observational data and data-driven simulations, which show plastic particle accumulation inside ocean gyres and higher density of particles off the gyre centres. This suggests that particle inertia could play a role in in determining the denser plastic accumulation zones and should be taken into account for oceanic plastic transport modelling.

3 Attractive-repulsive hydrodynamics for consensus in collective behaviour

Sergio P. Perez
Imperial College London, Departments of Chemical Engineering and Mathematics

This poster summarises and illustrates the main qualitative properties of hydrodynamics models for collective behaviour. These models include a velocity consensus term together with attractive-repulsive potentials leading to non-trivial flock profiles. The connection between the underlying particle systems to the swarming hydrodynamic equations is performed through kinetic theory modelling arguments.We focus on Lagrangian schemes for the hydrodynamic systems showing the different qualitative behaviour of the systems and its capability of keeping properties of the original particle models. We illustrate known results concerning large time profiles and blow-up infinite time of the hydrodynamic systems to validate the numerical scheme.

4 Interactions between the Subharmonic and Harmonic Content of Self Excited Combustion Instabilities of Lean Propane Mixtures

E. Karlis, Y. Hardalupas, A. M Taylor
Imperial College London, Department of Mechanical Engineering

Thermoacoustic instabilities under lean operation in gas turbine burners hinder the development of lean premixed combustion mode of operation, thus limiting the potential to decrease NOx emissions. In this work we focus on the interaction between the fundamental frequency and its harmonic and subharmonic frequencies observed in spectra of heat release and acoustic pressure signals under limit cycle combustion instabilities of a propane/air premixed swirl stabilized model gas turbine combustor. We studied experimental cases of self-excited oscillations under four equivalence ratios (0.50, 0.55, 0.60, and 0.65) and 12 Reynolds numbers (from 15000 to 26000 at increments of 1000) to cover a broad range of operational regimes. It was observed that for equivalence ratios above 0.55 a subharmonic frequency emerges in both heat release and acoustic spectra. The dynamics underwent a period doubling bifurcation with the limit cycle transitioning into two distinct toroidal structures in phase space. The emergence of the heat release subharmonic frequency and its relationship with the fundamental affected the dependence of the harmonics on the fundamental. The subharmonic heat release amplitude initially increased linearly with that of the fundamental heat release up to an inflection point after which a decreasing trend was observed. The amplitude of the acoustic fundamental increased linearly with the heat release fundamental up to a saturation point that coincided with the inflection point. At this point the power law dependence between the fundamental and the first harmonic of the acoustic pressure signal is no longer held. This indicated a strongly nonlinear behaviour as well as energy dissipation from the acoustic fundamental to the first harmonic. The emergence of the heat release subharmonic frequency was manifested on the phase averaged flame shape over a period of the instability. Before period doubling occurred, the flame propagated on the boundary layer along the combustor wall assuming a V shape. After the period doubling occurrence the flame anchored on the shear layers surrounding the internal recirculation zone established by the vortex breakdown of the swirling motion of the flow assuming an M shape.
We attempt to explain the flame shape transition by calculating the maximum strain imposed on the flame via high speed phase averaged PIV vector fields. The ratio of the strain imposed on the flame over the flame extinction strain rate scaled close to one, at the region of the inner shear layers

for M flame shapes. The same ratio was significantly greater than unity for V shape flames along the inner shear layers hence the flame was not able to further anchor upon them. Linear acoustic analysis of our geometry demonstrated that the subharmonic was not an excited acoustic mode, hence we conjecture its emergence was related to hydrodynamic phenomena. To explicitly depict the hydrodynamic structure associated with the subharmonic frequency we applied the Dynamic Mode Decomposition (DMD) on both heat release rate and PIV datasets. We show that the flame is modulated by vortex merging along the shear layers and subsequent vortex rollup of the flame surface at the subharmonic frequency. We suggest that the observed decrease of the subharmonic amplitude after the defined deflection point in the heat release rate power spectra was related to attenuation of these vortical structures. This effect is related to an increase in viscosity in this region due to the fest anchoring of the flame in the shear layer region for the highest of the equivalence ratios examined.

5 Mixing and energetics of a gravity current on a slope

E. A. Dieu, M. van Reeuwijk, G. O. Hughes
Imperial College London, Department of Civil & Environmental Engineering

Turbulent buoyancy-driven flows arise commonly in both the natural and the built environments, and most previous studies have focused on cases where these flows are aligned with either the vertical or horizontal directions. However, buoyancy-driven flows are often confined by sloping boundaries, and the governing dynamics differ as soon as the slope forms an angle with the horizontal of order a degree or more. Turbulence is typically energised along the interface of a slope current, leading to entrainment and mixing with the ambient fluid. Previous studies have attempted to characterise entrainment in a slope current, but a robust understanding and parameterisation of this behaviour is yet to emerge.

In this study, we adopt a new approach to characterise mixing in slope gravity currents by understanding their energy budget. We analyse data from direct numerical simulations of an evolving dense current for a range of slope angles, and quantify both the resultant mixing and the energy consumed by stratified turbulence. We discuss the consequences for the dynamics and parameterisation of mixing in slope gravity currents.

6 Efficient design for additive manufacturing: topology optimisation for heat transfer

Marco Pietropaoli
Imperial College London, Department of Aeronautics

Additive manufacturing has given rise to previously unthinkable possibilities. Complex mechanical components can now be produced directly from digital models avoiding the limits imposed by traditional manufacturing techniques. Despite this great flexibility, aeronautical, mechanical and biomedical industries keep using old and relatively inefficient designs for their products. A paradigm shift in the design approach is needed. Biological structures such as bones and veins are really efficient and nearly perfect. We developed a design method able to emulate natural evolution and automatically design super efficient components for aircraft and cars. Not surprisingly, those new components bear striking resemblance to bronchial tubes. Fluid and structure interaction problems for aeronautics are at the core of our research. A key example: in order to survive under the huge thermal stresses due to fuel combustion, gas turbine blades are internally cooled by flowing coolant air. Bio-inspired designs for the internal structures of the blades increase the efficiency of the thermal exchange. Such an improvement would potentially double the life of the engine, increase the overall efficiency and reduce the emissions. In terms of maintenance and fuel, there would be an overall saving of hundreds of millions of pounds per aeroplane. On benchmarks, design solutions obtained using our optimisation routines are up to 40% more efficient compared to the state of the art techniques. Moreover, the versatility of the method makes it applicable also to automotive, chemical and energy industry.

7 Direct Numerical Simulation of Three-dimensional jets

Ricardo Constantea, Lyes Kahouadjia, Andre Nicolleb, Jalel Cherguic, Damir Juricc, Seungwon Shind, Omar K. Matar
Imperial College London, BP, LIMSI, CNRS, Hongik University

Direct Numerical Simulations resolves the whole range of spatial and temporal scales of turbulence. We carry out direct numerical simulations of liquid jet dynamics and breakup using the BLUE code, which tracks the interface using a hybrid front-tracking/Level Contour Reconstruction Method; it defines the interface position through a marker function and a local triangular Lagrangian mesh.
Liquid jet breakup is an example of interfacial complexity associated with multiphase flows because of the formation of ligaments and their pinch off to give rise to droplet formation. We consider the atomisation of a liquid jet released into either stagnant gas or other liquid phase where the velocity is stimulated sinusoidally to promote the growth of Kelvin-Helmholtz instabilities, thus forming a flow system characterized by complex interfaces. The influence of the Reynold number and frequency imposed on the sinusoidal velocity has been analysed. The spread of cylindrical water jet into coflowing air is also considered (essentially, a replication of the Marmottant and Villermaux experimental work [1]).

[1] Marmottant, P., & Villermaux, E. (2004). On spray formation. Journal of fluid mechanics, 498, 73-111.

8 DALES-URBAN: Towards an integrated large-eddy simulation model for the urban climate

Tom Grylls, Ivo Suter, Harm Jonker, Jasper Tomas and Maarten van Reeuwijk
Imperial College London, Delft University of Technology

Over half of the worlds population now lives in urban areas, meaning the interaction between the urban population, built environment and climate is increasingly important in terms of the sustainability of life in a city. Global challenges such as air quality and the urban heat island effect necessitate advancements in our capabilities to computationally model and resolve the fundamental physics and processes occurring in the urban climate. The heterogeneity and scales involved in real cities renders this a complex research problem. To this end, we are developing a high fidelity simulation tool that can resolve flows in the urban environment with unprecedented precision. The areas of development of the model are outlined below, aiming to encompass the range of diverse phenomena prevalent within real cities.

9 Mathematical models of vitreoschisis in the eye: Derivation of an appropriate slip condition to model the resultant motion of the vitreous

Laura Bevis
Imperial College London, Department of Bioengineering

As the body ages, the gel-like vitreous humour in the back of the eye degrades, forming pockets of aqueous fluid that migrate towards the circumference of the eye. Vitreoschisis occurs when this liquefied vitreous becomes trapped between the layers of the vitreous cortex, which surrounds the vitreous humour. This condition is thought to increase the risk of retinal detachment, which affects about 1 in 300 people and can cause blindness if not treated immediately. In order to model the motion of the vitreous humour as a result of vitreoschisis, previous work has modelled a sphere filled with fluid, representing the vitreous humour, with a slip condition over the region of the vitreoschisis to represent the condition [1]. They investigated the dynamics of this fluid under small, periodic eye rotations and computed the stress on the wall of the sphere, to give insight as to how the condition affects the retina. Here we compute the correct mathematical form of the slip condition for a given vitreoschisis by considering layers of different incompressible fluids above an oscillating, flat plate representing the retina. The layers representing the central vitreous and vitreous cortex are treated as viscoelastic using a simple non-Newtonian fluid model, while the layer representing the liquefied vitreous is treated as Newtonian. This allows us to derive an analytical expression for the slip condition required to model the full dynamics. This slip condition could be implemented with previous work to model the full system in a sphere, in order to give a more accurate computation of the stress on the retina. Further work includes modelling a finite case of vitreoschisis and including the effects of retinal curvature. It would also be of interest to consider the motion of the vitreous under a sudden acceleration, rather than periodic motion.

[1] Isakova K. (2016). Mathematical models of fluid motion in the vitreous chamber of the human eye. PhD thesis.

10 Understanding the scaffold of turbulence in high Reynolds number flows

ELEANOR JOHNSTONE
Imperial College London, Department of Mathematics

It is thought that one transatlantic flight can add as much to your carbon footprint as a typical year's worth of driving. Why are planes so inefficient? Well, part of the reason is turbulence, which causes drag on the aircraft. Therefore, one of the most important questions to ask is about laminar flow control: how can we control the flow in specific areas so turbulence is less prevalent? Not only could this save the aerospace industry millions of pounds in fuel, but it would also have a huge impact on the sustainability of air travel.
Despite turbulent motions appearing completely random to the naked eye, research in the past half century has determined an underlying coherent structure in turbulent flows, caused by interactions between vortices and waves. This poster will examine this scaffold of the turbulence in more detail, including how the structure sustains itself without external perturbation, and the implications of such interactions in the freestream to the basic boundary-layer flow. Recent work in the field of compressible flows will be shown.